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Mastering Mechanical Desktop Release 3

Surface, Parametric, and Assembly Modeling

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Contents

Preface

Mechanical Desktop is a computer-aided design application for constructing Non-Uniform Rational B-Spline (NURBS) surface models, 3D parametric solid models, assemblies and subassemblies, and associative engineering drawings.

This book is designed to give you an opportunity to practice applying Mechanical Desktop to construct 3D NURBS surface models and 3D parametric engineering designs, to construct an assembly from a set of solid parts, to output 2D engineering drawings, and to become familiar with the utilities provided. It is assumed that you have installed the Mechanical Desktop application properly in your computer. This book consists of five chapters and an appendix. Chapter 1 offers a brief introduction to the Mechanical Desktop application. Chapter 2 shows you how to use Mechanical Desktop to construct 3D wires from which you can make various kinds of surfaces, compose 3D surface models, and interoperate surfaces with solids. The surface modeling tool of the Mechanical Desktop is a NURBS surface modeling system, which allows you to use NURBS mathematics to construct NURBS surfaces and free-form surfaces. Because Mechanical Desktop is a member of the AutoCAD family, objects constructed are fully compatible with AutoCAD native solids. You can use surfaces to cut these solids and incorporate free-form surface features into them. You can also convert a solid to a set of surfaces.

Chapter 3 concerns parametric solids. Within the framework of the Autodesk CAD system, there are two kinds of solids: Mechanical Desktop solids and AutoCAD native solids. Both are 3D solid models, and they are fully compatible with each other. You can convert a native solid to a static parametric solid or convert a parametric solid to a native solid. You can use a surface to cut a solid. Thus, you can incorporate a free-form surface into a solid model. You can also convert a parametric solid model to a set of NURBS surfaces.

After learning how to construct 3D parametric solids in Chapter 3, you will work on virtual assemblies in Chapter 4. You will put solid parts together to form assemblies. Making assemblies from a set of solid parts involves applying constraints to features of paired solid parts. After making assemblies, you will construct assembly scenes in which you will explode and tweak the solid parts apart to form exploded views.

In Chapter 5, you will construct 2D engineering drawings from solid parts, assemblies, and surface models. The 2D drawings are associative to the 3D objects. For solid parts in particular, the drawing and the 3D objects are bidirectionally associative. Changing a parametric dimension in a drawing modifies the 3D solid and modifying the 3D solid amends the drawing.

Finally, the appendix offers a brief explanation of all the commands provided by Mechanical Desktop.

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Ron K. C. Cheng

Chapter 1

Introduction

Aims and Objectives

The aims of this chapter are to introduce the four major design tools of Mechanical Desktop, to outline the key functions of the design tools, and to familiarize you with the Mechanical Desktop user interface. After studying this chapter, you should be able to

- Describe the four major design tools of Mechanical Desktop
- State the key functions of these design tools

Overview

Mechanical Desktop is a computer-aided design (CAD) application for making 3D Non-Uniform Rational B-Spline (NURBS) surface models, 3D parametric solid models, assemblies and subassemblies of solid models, and associative engineering documents of 3D surfaces, solids, and assemblies. It has four sets of design tools: surface modeling, part modeling, assembly modeling, and associative drafting. Because Mechanical Desktop runs on top of AutoCAD, all the design and drafting tools of AutoCAD are available. (See Figure 1.1.)

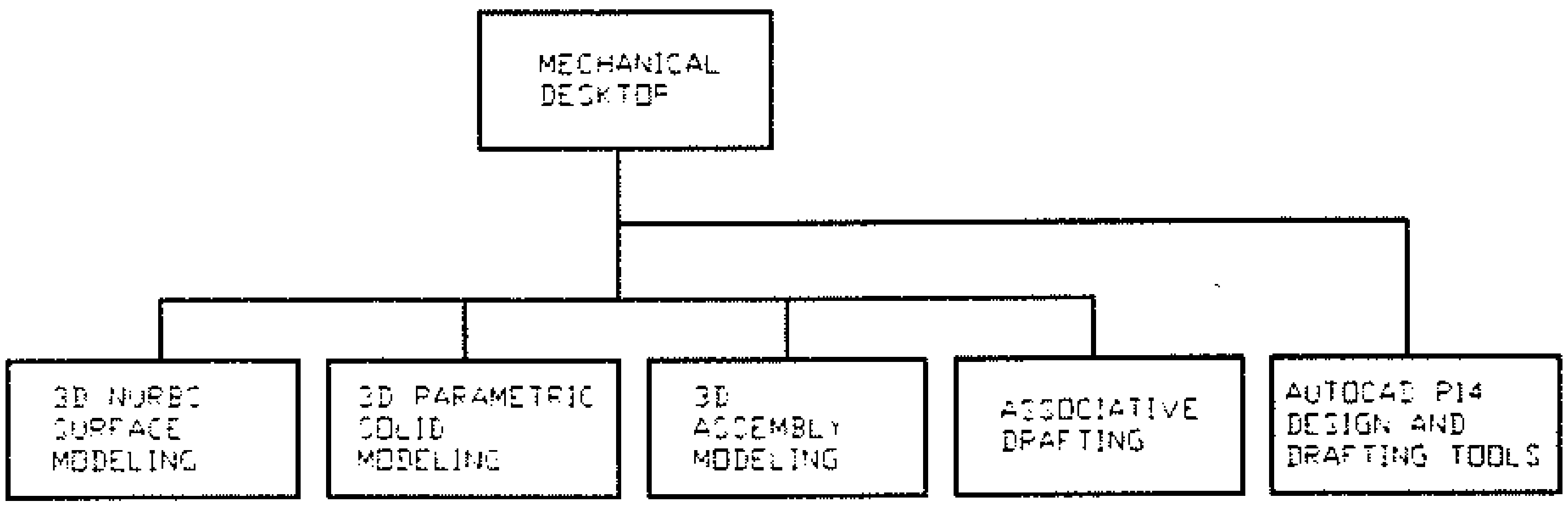


Figure 1.1 Four sets of Mechanical Desktop design tools together with AutoCAD tools

1.1 Engineering Design Tools

Surface Modeling Tool

The surface modeling tool (Chapter 2) is a NURBS surface modeling system for design and manufacturing. You can use it to construct free-form surfaces in a computer. In addition, you can use its supplementary tools to construct and edit wires for constructing and editing NURBS surfaces. Figure 1.2 shows the surface model of a 1/10 scale model car together with all the mechanical parts constructed by using the parametric solid modeling tool (Chapter 3) and the assembly modeling tool (Chapter 4).

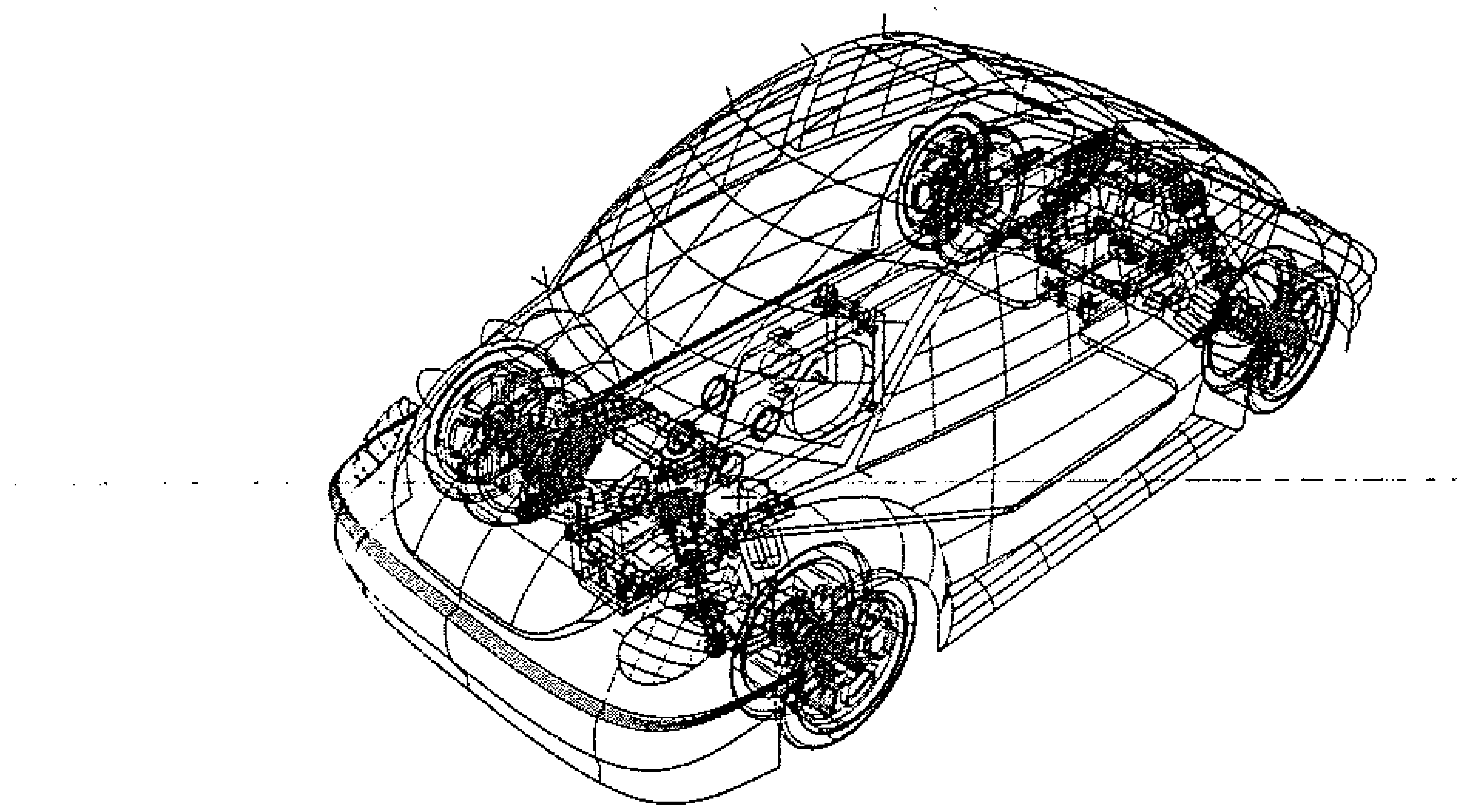


Figure 1.2 Surfaces and mechanical parts of a 1/10 scale model car

Apart from making surface models, you can use a NURBS surface to cut an AutoCAD native solid or a Mechanical Desktop parametric solid. Working in the opposite direction, you can convert an AutoCAD native solid or a Mechanical Desktop solid to a set of NURBS surfaces. Figure 1.3 shows the solid model of a bevel gear constructed by cutting a solid with a set of surfaces.

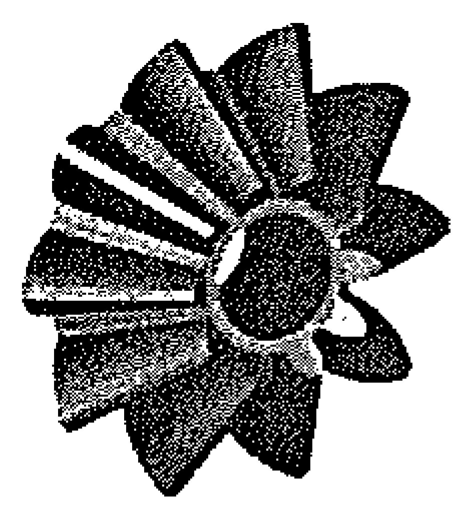


Figure 1.3 Bevel gear constructed by using surfaces to cut a solid

Part Modeling Tool

The part modeling tool (Chapter 3) is a parametric feature-based solid modeling system for construction and subsequent editing of a dimension-driven 3D solid model. Solids are constructed from rough sketches and can be modified. Surfaces can be included. In addition, you can convert a parametric solid to a set of surfaces, convert a parametric solid to a native solid, and convert a native solid to a static parametric solid on which you can add parametric solid features. Figure 1.4 shows a parametric solid model of the suspension arm of a 1/10 scale model car.

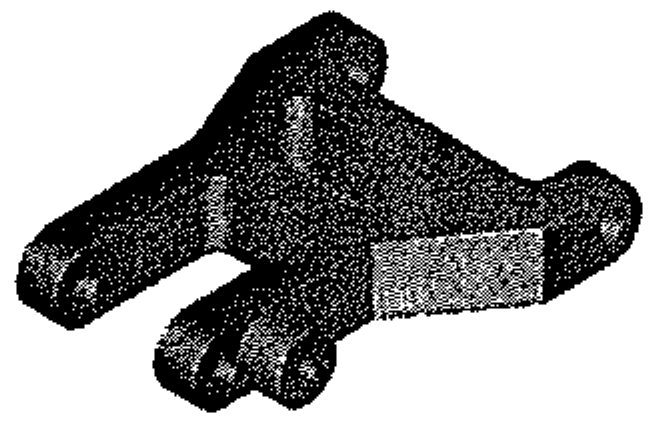


Figure 1.4 Parametric solid of the suspension arm of a 1/10 scale model car constructed in Chapter 3

Assembly Modeling Tool

The assembly modeling tool (Chapter 4) is a design tool for assembling a set of solid parts to form assemblies and subassemblies. Assembly modeling involves the application of constraints to selected pairs of features of the solid parts. With an assembly, you can check interference and set up a number of assembly scenes in which you can explode the solid parts. Figure 1.5 shows the assembly of a scale model car.

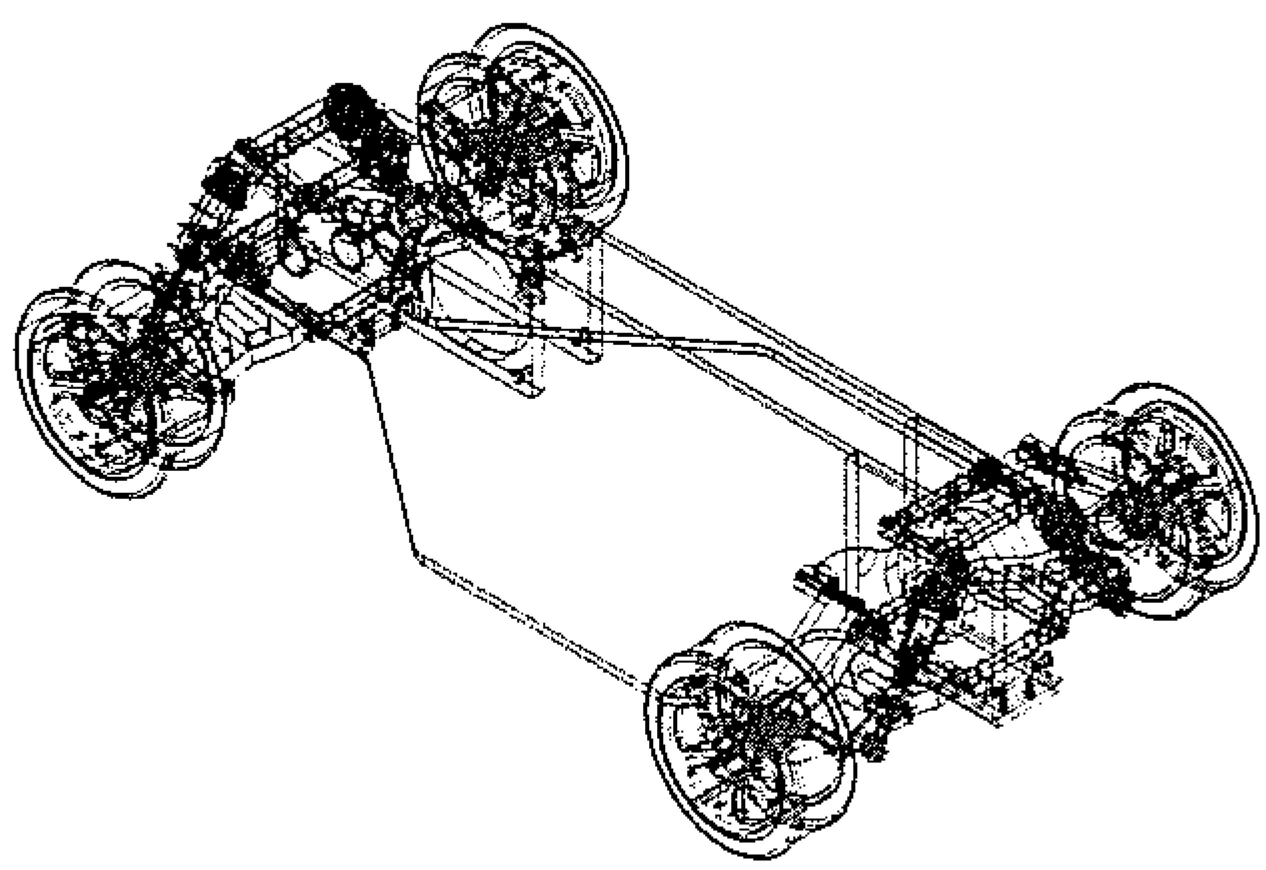


Figure 1.5 Assembly of the solid parts of a 1/10 scale model car

Associative Drafting Tool

The associative drafting tool (Chapter 5) is used for generating 2D engineering drawings from 3D objects, surfaces, parametric solids, and assemblies. You can construct orthographic, isometric, auxiliary, detailed, and broken drawing views. The engineering drawings and the 3D solid parts are associated with each other. Changes in either the engineering drawing or the 3D solids cause automatic changes in the other. Figure 1.6 shows the associative drawing of a U-bracket of a 1/10 scale model car.

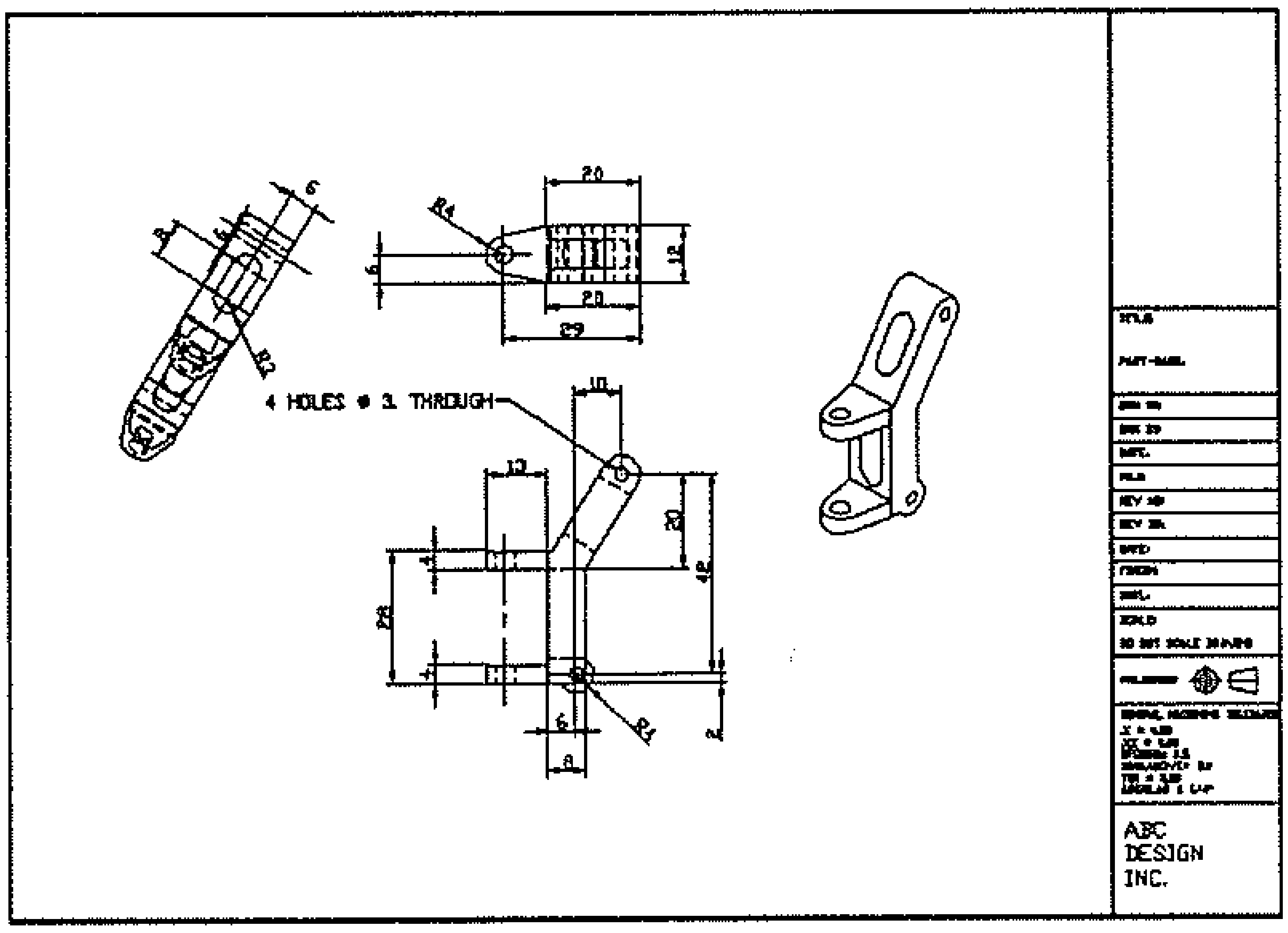


Figure 1.6 Associative engineering document of a U-bracket of a 1/10 scale model car

Compatibility and Interoperation

Mechanical Desktop solids and surfaces and AutoCAD native solids are fully compatible. You can interoperate them. You can change a parametric solid to a native solid or convert a native solid into a static base solid feature. You can use a NURBS surface as a surface feature to cut a parametric solid or a native solid. You can also convert a parametric solid and an AutoCAD native solid to a set of NURBS surfaces. (See Figure 1.7.)

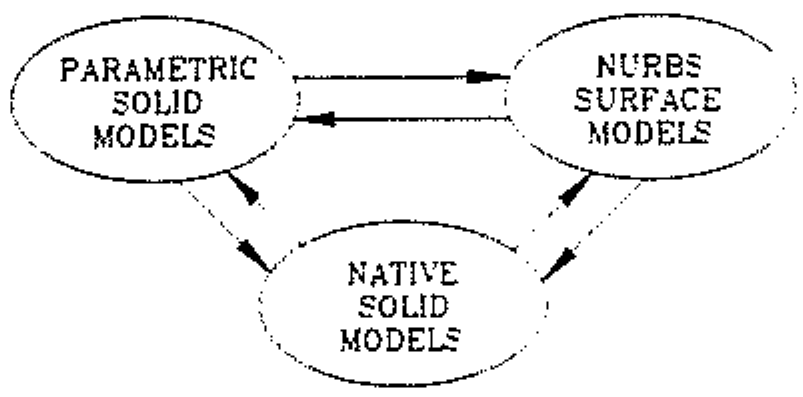


Figure 1.7 Interoperation

1.2 Mechanical Desktop Application Window

When you start Mechanical Desktop, your screen layout will resemble Figure 1.8. The Mechanical Desktop user interface is similar to that of AutoCAD. It has additional browser and shortcut keys. At the top of your screen, you will find 12 pull-down menu items. In particular, the Surface, Part, Assembly, and Drawing pull-down menus concern the four major design tools of Mechanical Desktop. If you do not find these pull-down menus, you can use the MENULOAD command (Figure 1.9) at the command line interface.

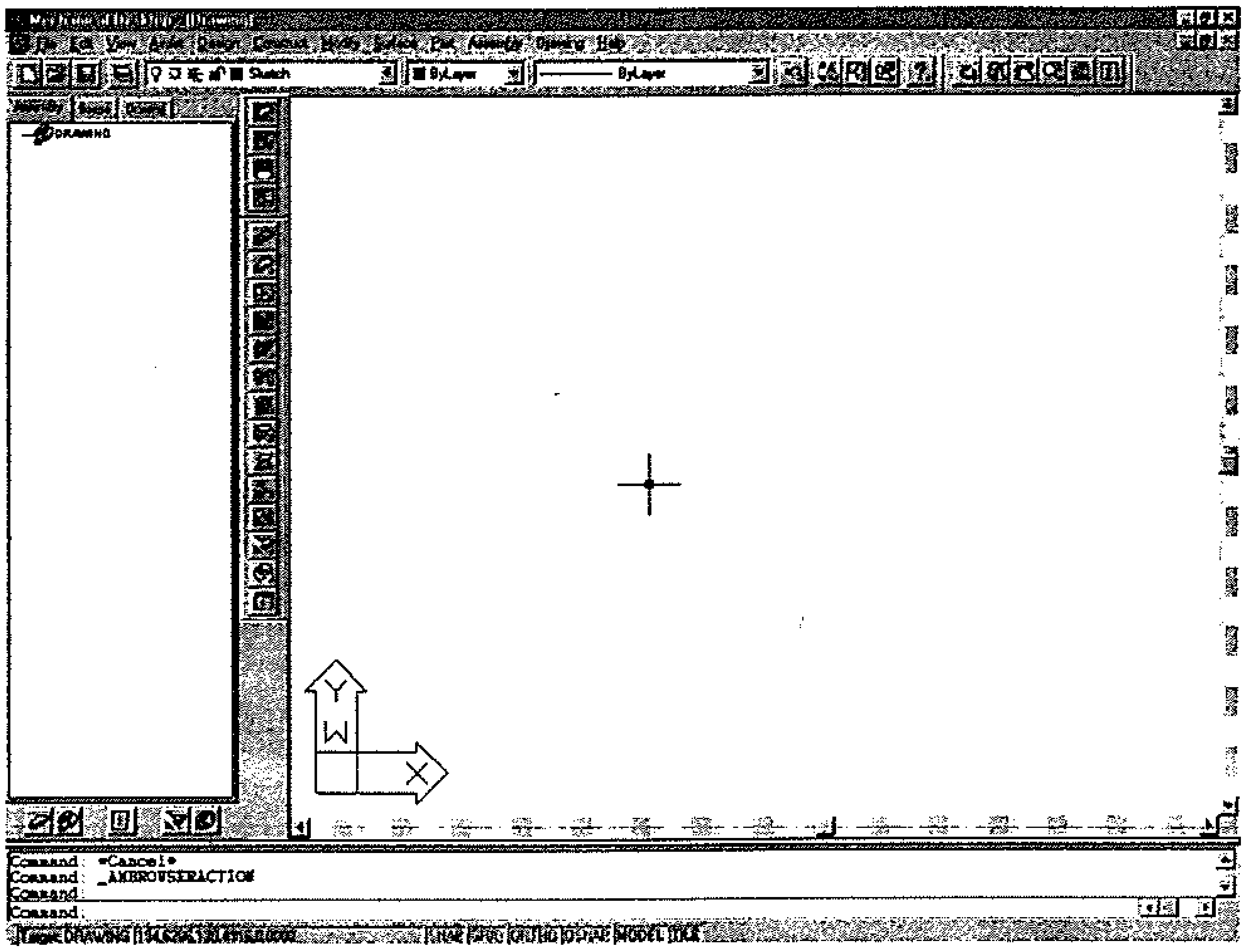


Figure 1.8 Mechanical Desktop application window

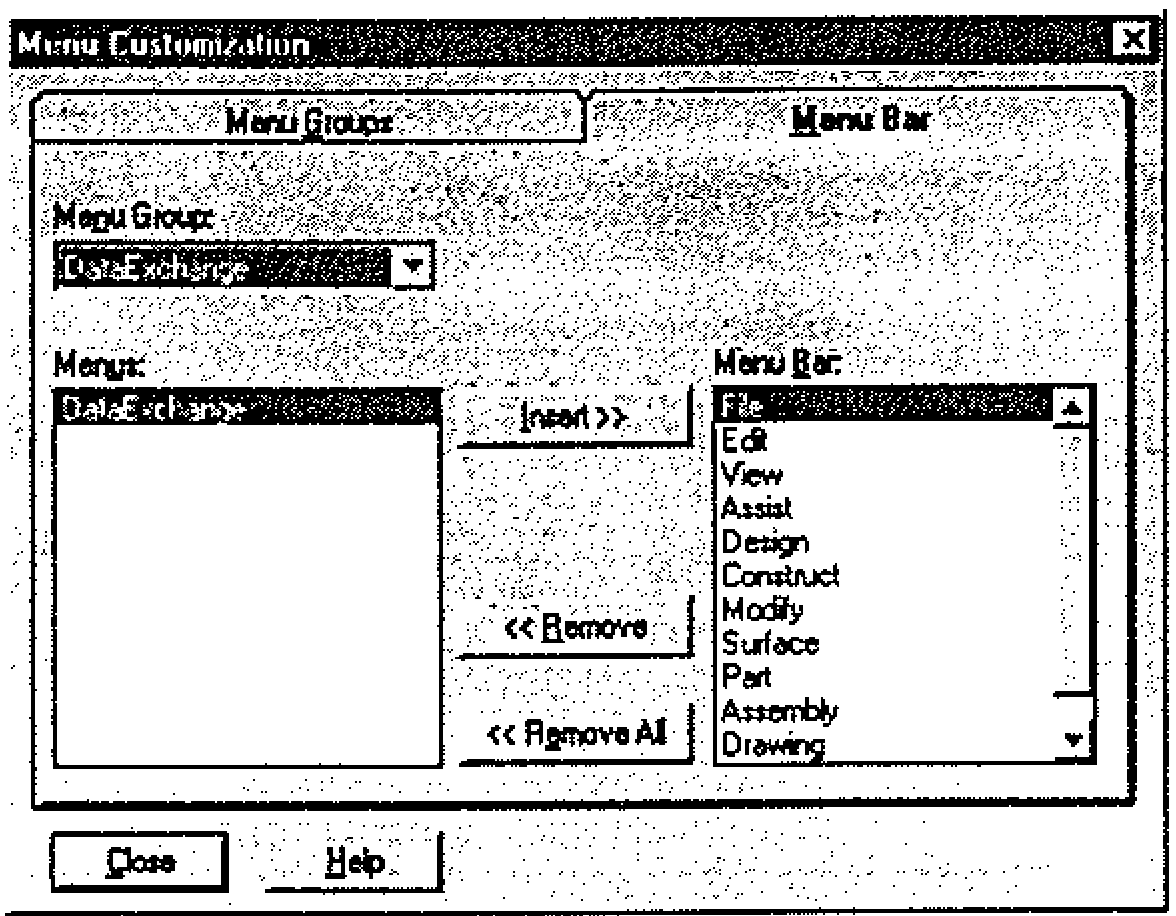


Figure 1.9 Menu Customization dialog box

If the Desktop Browser is not displayed, you can use the AMBROWSER command by selecting the Desktop Browser item of the View pull-down menu. (See Figure 1.10.)

<View> **<Desktop Browser>**[1]

[1] In the explanation that follows, **<View>** **<Desktop Browser>** will mean selecting the **<View>** pull-down menu, then selecting the **<Desktop Browser>** item.

Command: **AMBROWSER**

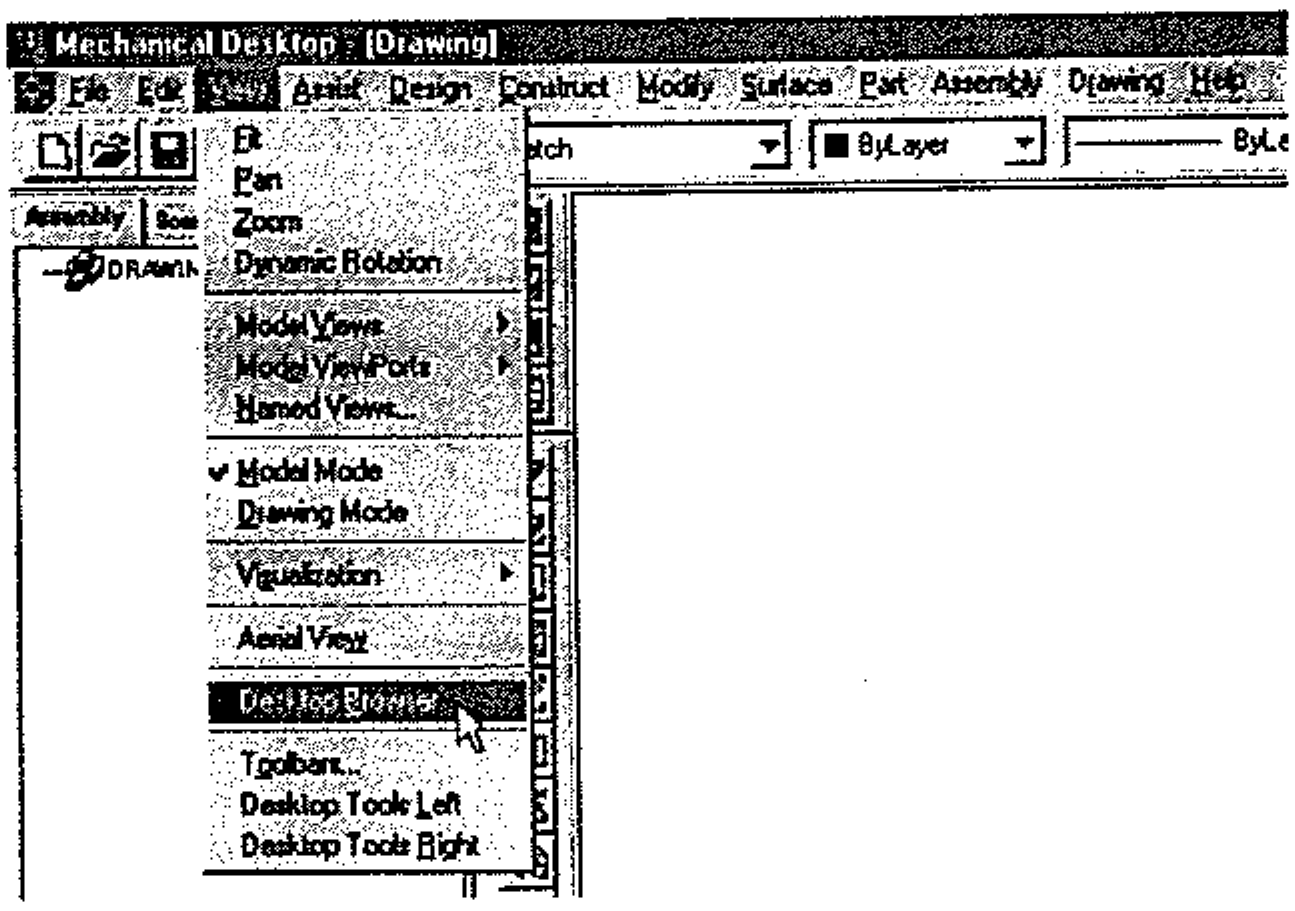


Figure 1.10 Selecting the Desktop Browser item from the View pull-down menu

To display toolbars, you can use the TOOLBAR command. (See Figure 1.11.) By checking the appropriate boxes in the Toolbars list box, you can call out the relevant toolbars. Figure 1.12 shows the Surface Modeling toolbar.

<View> **<Toolbars...>**

Command: **TOOLBAR**

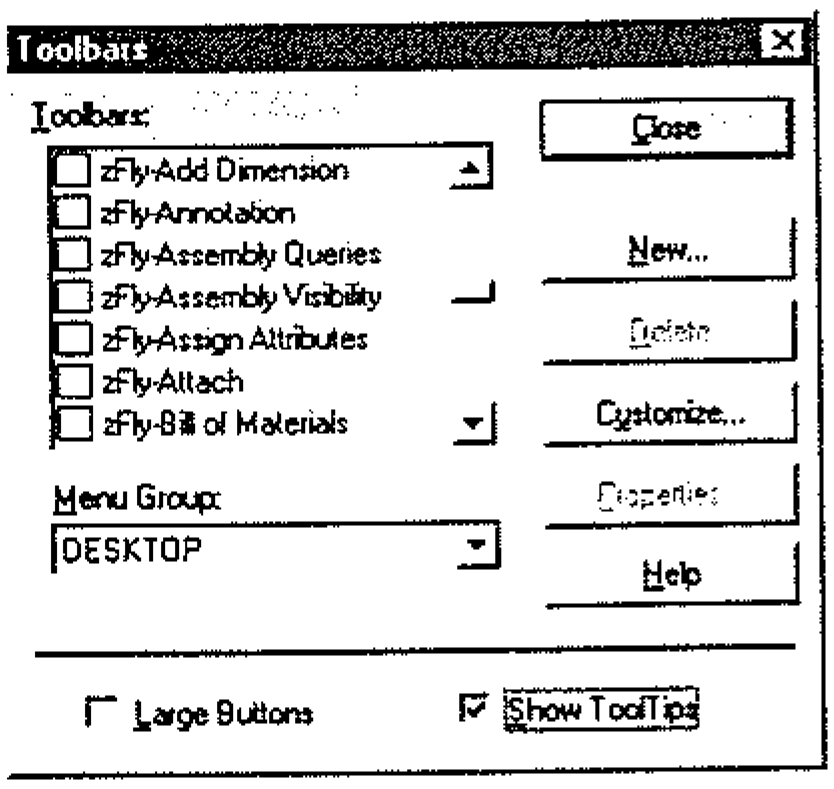


Figure 1.11 Toolbars dialog box

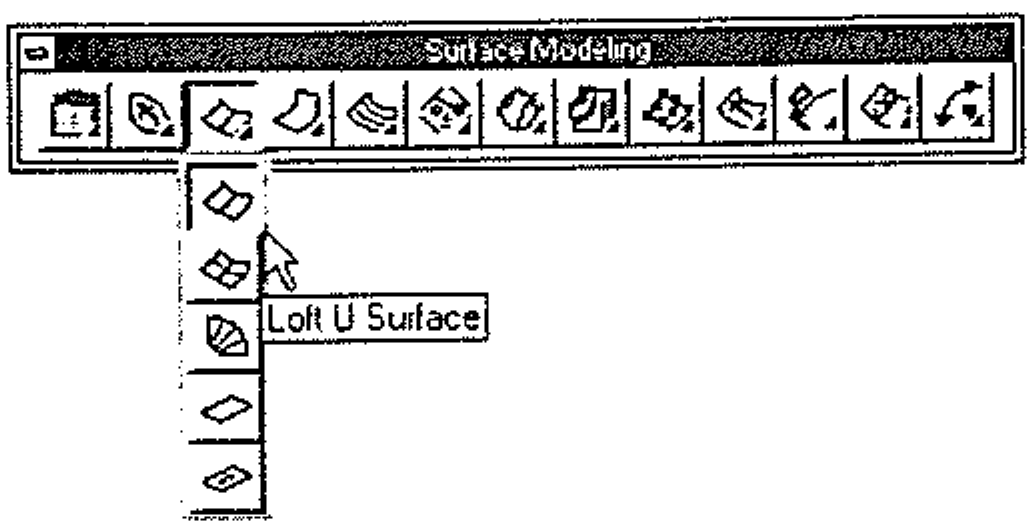


Figure 1.12 Surface Modeling toolbar

As illustrated in Figure 1.12, selecting the Loft U Surface item of the Surface Modeling toolbar runs the AMLOFTU command.

[Surface Modeling] **[Loft U Surface]**[2]

Command: **AMLOFTU**

1.3 Command Interaction

With the pull-down menu and toolbars in their proper positions, you can use Mechanical Desktop commands in several ways:

- You can select an item from the pull-down menu or the cascading menu from the pull-down menu. (See Figure 1.13.)
- You can select an icon from the toolbars. (See Figure 1.14.)
- You can type the command name at the command line interface. (See Figure 1.15.)
- You can use the right mouse button to bring out the shortcut menu from the Desktop Browser. (See Figure 1.16.)

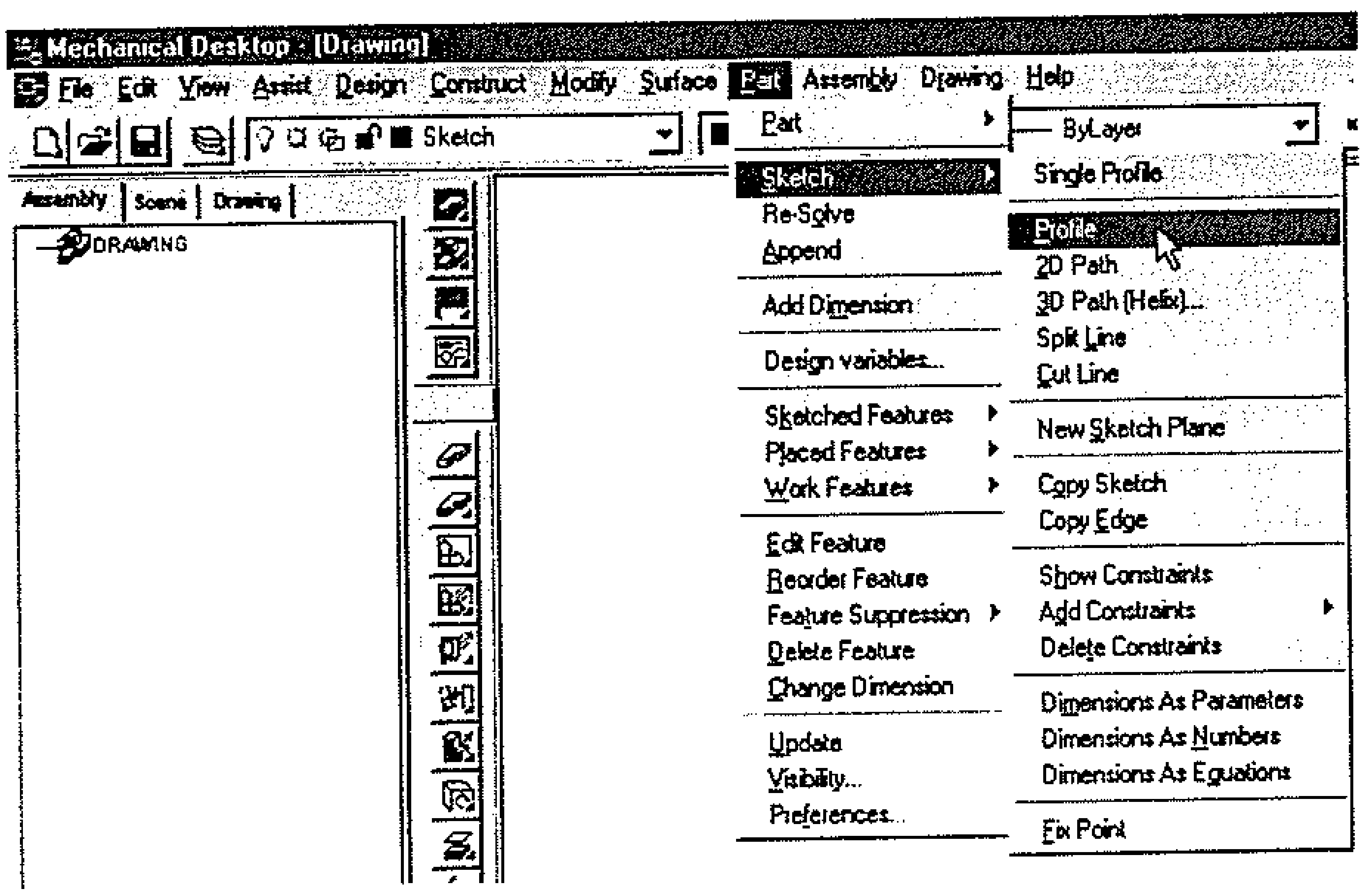


Figure 1.13 Selecting an item from the pull-down menu

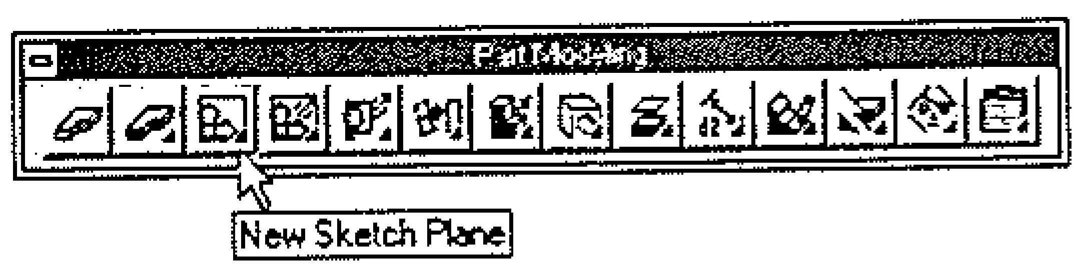


Figure 1.14 Selecting an icon from a toolbar

[2] In the explanation that follows, **[Surface Modeling]** **[Loft U Surface]** will mean selecting the **[Surface Modeling]** toolbar, then selecting the **[Loft U Surface]** icon.

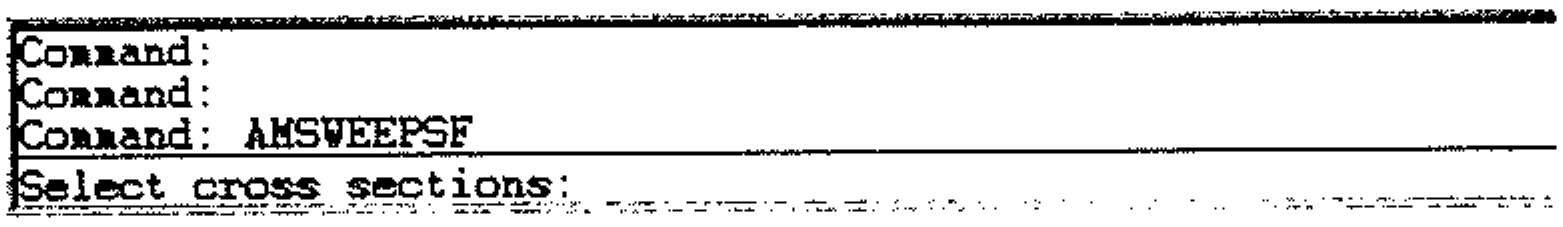


Figure 1.15 Inputting the command name at the command line

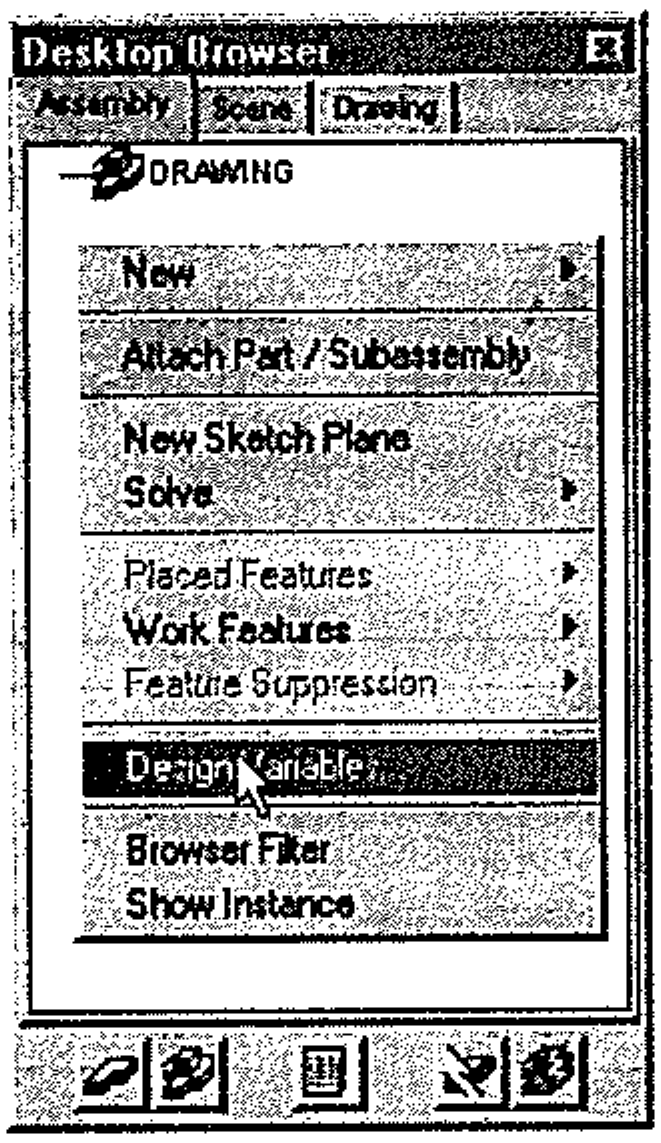


Figure 1.16 Bringing out the shortcut menu from the Desktop Browser

1.4 Key Points and Exercises

Mechanical Desktop is an engineering design tool that runs on top of AutoCAD R14. It has four powerful design tools: a surface modeling tool, a parametric solid modeling tool, an assembly modeling tool, and an associative engineering drawing construction tool.

Basically, the application window is much the same as that of AutoCAD R14, with a few differences. It has additional pull-down menu items. It has a Desktop Browser that provides command shortcuts. And AutoCAD pull-down menu items are rearranged.

Exercise 1.1

What are the four design tools of Mechanical Desktop? State their prime functions.

Exercise 1.2

How many ways can you run a Mechanical Desktop command? List them.

Chapter 2

NURBS Surface Modeling

Aims and Objectives

The aims of this chapter are to explain the key concepts of surface modeling, to delineate various methods of constructing and editing 3D wires and 3D surfaces, to let you master the techniques of using Mechanical Desktop to construct free-form NURBS surface models, to outline various surface modeling utility tools, and to show the ways to incorporate free-form surfaces into solid models. After studying this chapter, you should be able to

- Describe the key concepts of surface modeling
- Build and modify 3D wires for making 3D surfaces
- Construct and edit various kinds of 3D surfaces
- Use surface modeling utilities in engineering design
- Incorporate free-form surface features into solid models
- Apply surface modeling tools in design projects

Overview

In our daily lives, we encounter many objects with free-form surfaces. Some examples are the handle of a razor, the casing of a computer pointing device, the casing of a mobile phone, and the body panels of an automobile. To construct these objects as 3D models in the computer, we need surface modeling tools.

In this chapter, you will focus on NURBS surface modeling tools and work on a number of projects through which you will learn how to construct and edit 3D wires and 3D surfaces, use various surface modeling utilities, and use free-form surfaces to cut native solids. After learning how to construct parametric solid models in the next chapter, you will use a free-form surface to cut a parametric solid as well.

2.1 Surface Modeling Concepts

A surface in a computer is a mathematical expression that represents a 3D shape with no thickness. NURBS mathematics is the most advanced tool in surface modeling. It allows the implementation of multipatch surfaces with cubic surface mathematics. And, most important, it maintains full continuity control even with trimmed surfaces.

Mechanical Desktop surface modeling tools use NURBS mathematics for the construction of splines and surfaces and offer additional sophisticated tools for construction and editing wires for subsequent generations of NURBS surface models. Using Mechanical Desktop import utilities, you can construct 3D surfaces from IGES files or digitized data. Given the shell thickness, you can evaluate mass properties from a surface model. To integrate free-form surfaces in solid modeling, you can use a NURBS surface to cut an AutoCAD native solid or a Mechanical Desktop parametric solid. Working in the opposite direction, you can convert an AutoCAD native solid or Mechanical Desktop solid to a set of NURBS surfaces.

Surface Construction

Using Mechanical Desktop, you can construct three major kinds of NURBS surfaces: primitive surfaces, free-form surfaces, and derived surfaces. In addition, you can trim a surface to obtain the boundary of a particular shape, and convert AutoCAD objects and Mechanical Desktop solid objects into sets of surfaces.

Primitive Surfaces

As the name implies, primitive surfaces are basic geometric surface shapes. They are the conical surface, the cylindrical surface, the spherical surface, the toroidal surface, and the planar surface. (See Figure 2.1.) To produce a primitive surface, you specify a geometric shape and state its dimensions and location. For example, if you want to produce a cylindrical surface, you need only specify the location of the center of the base, the diameter, and the height. (These primitive surfaces, although easy to create, have very limited use.)

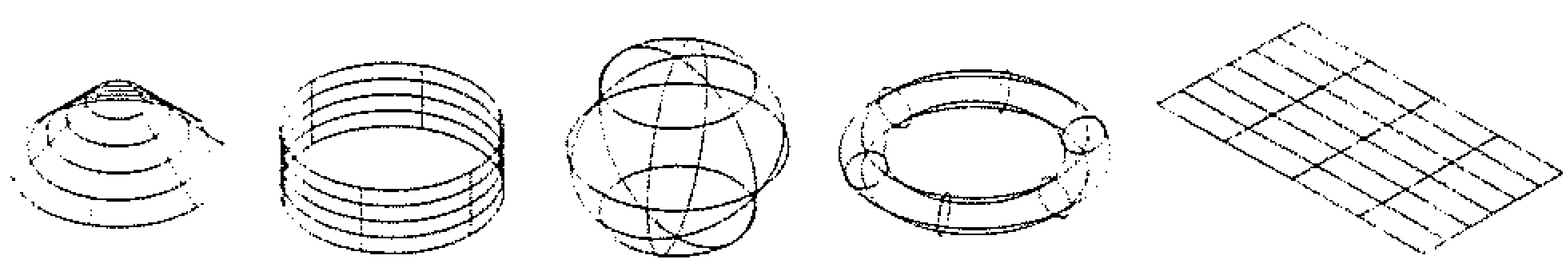


Figure 2.1 Conical, cylindrical, spherical, toroidal, and planar surfaces

In addition to making individual primitive surfaces, you can make a tubular surface with straight sections and circular elbows from a series of connected cylindrical and toroidal surfaces. To make a tubular surface, you construct a 3D polyline with a number of straight line segments. After that, you specify the diameter of the tube and the radii of the bends. (See Figure 2.2.)

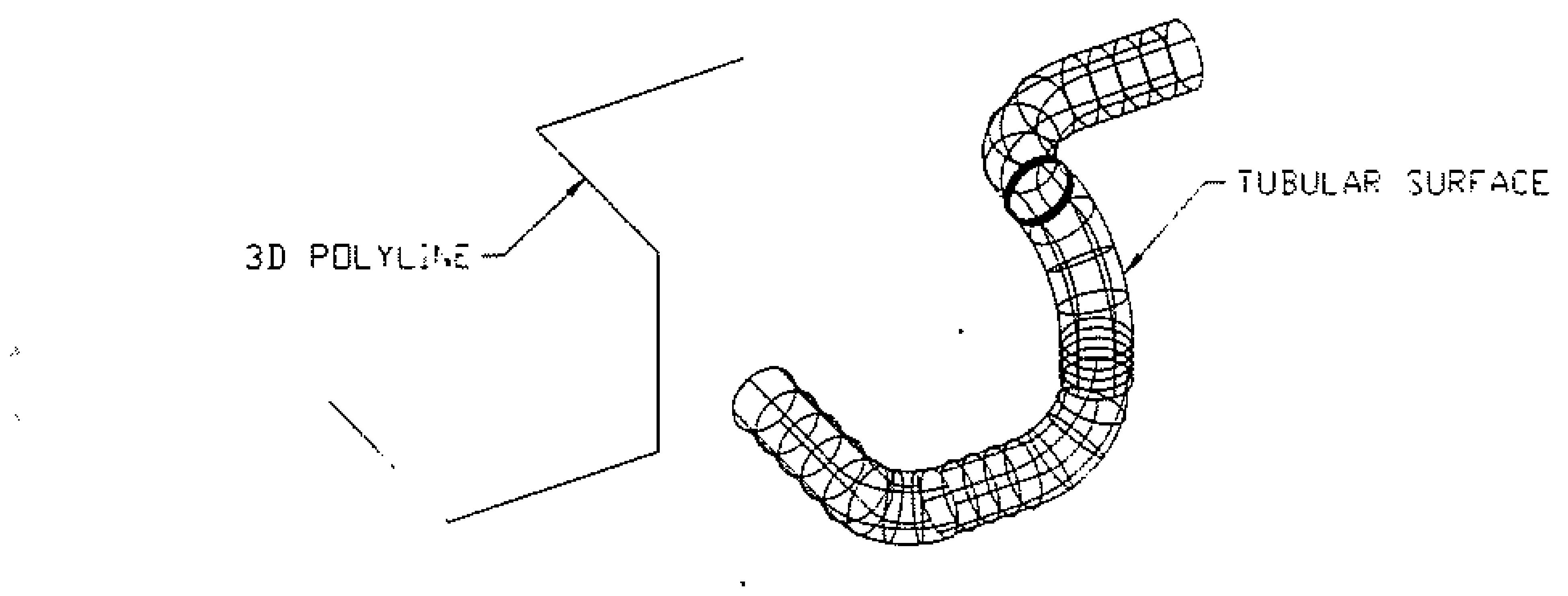


Figure 2.2 Tubular surface

Free-Form Surfaces

There are six kinds of free-form surfaces: the revolve surface, the extrude surface, the rule surface, the loft u surface, the sweep surface, and the loft uv surface. To make a free-form surface, the first step is to construct a set of wires that define the profiles and silhouettes of the surface. Based on the wires, the computer then computes and constructs a set of surface data.

In general, a free-form surface needs two sets of wires in two orthogonal directions to define its profile and silhouette. To distinguish these two directions from the X axis and the Y axis, they are called the U direction and the V direction. Wires in these directions are called U-wires and V-wires, respectively.

The revolve surface and the extrude surface are the simplest types of free-form surfaces, because each needs only a single wire to define its shape. For a revolve surface, you construct a wire that defines the cross section along the axis of revolution. To make the surface, you specify a section wire, an axis of rotation, and an angle of rotation. Figure 2.3 shows a revolve surface constructed by rotating a wire 180° about an axis.

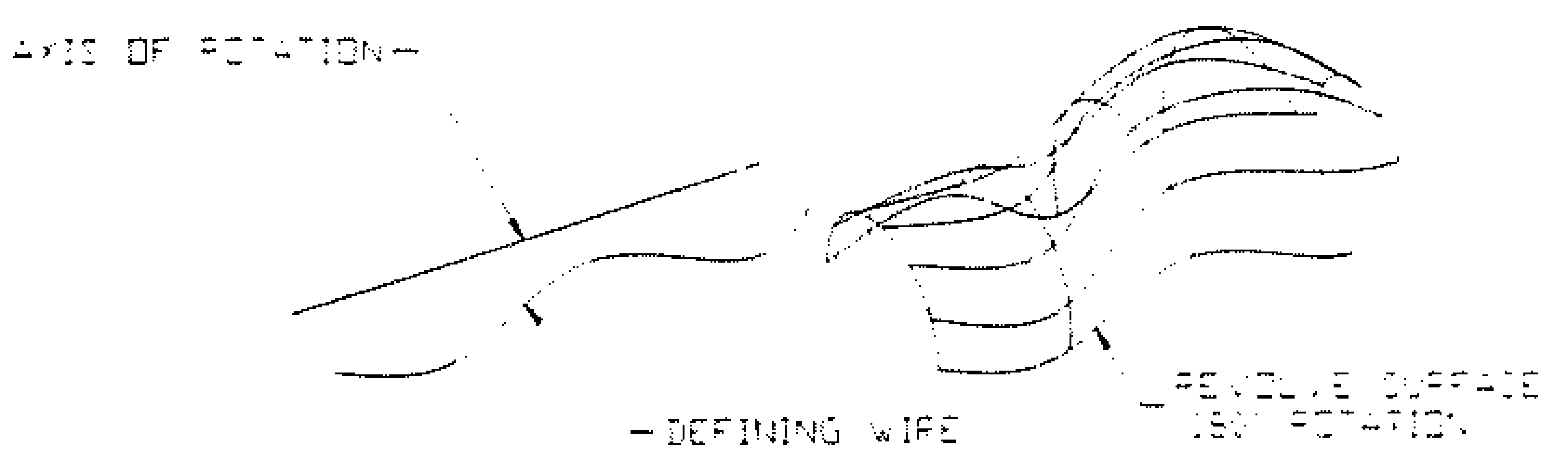


Figure 2.3 Revolve surface

Similar to a revolve surface, an extrude surface also needs a single wire to define its cross section. To make the extrude surface, you specify a section wire, a direction of extrusion, and a taper angle. (See Figure 2.4.)

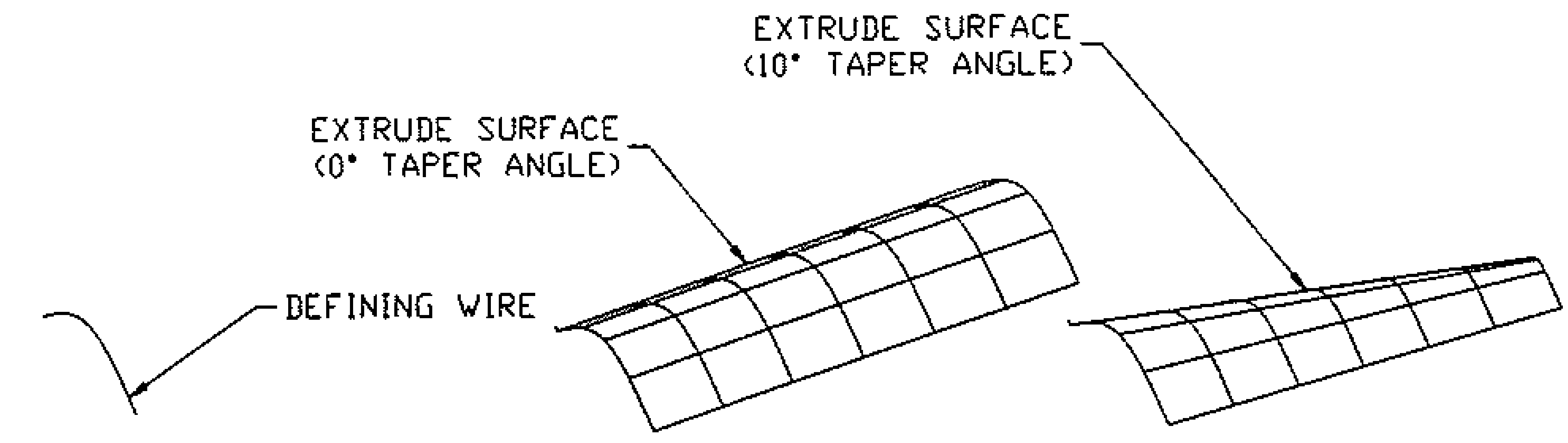


Figure 2.4 Extrude surfaces

To construct a surface that varies linearly in cross section from one edge to the other, you can supply two U-wires. The resulting surface is called a rule surface. It changes in cross section uniformly from the first wire to the second wire in one direction. In the other direction, the cross sections are straight lines. Figure 2.5 shows a rule surface.

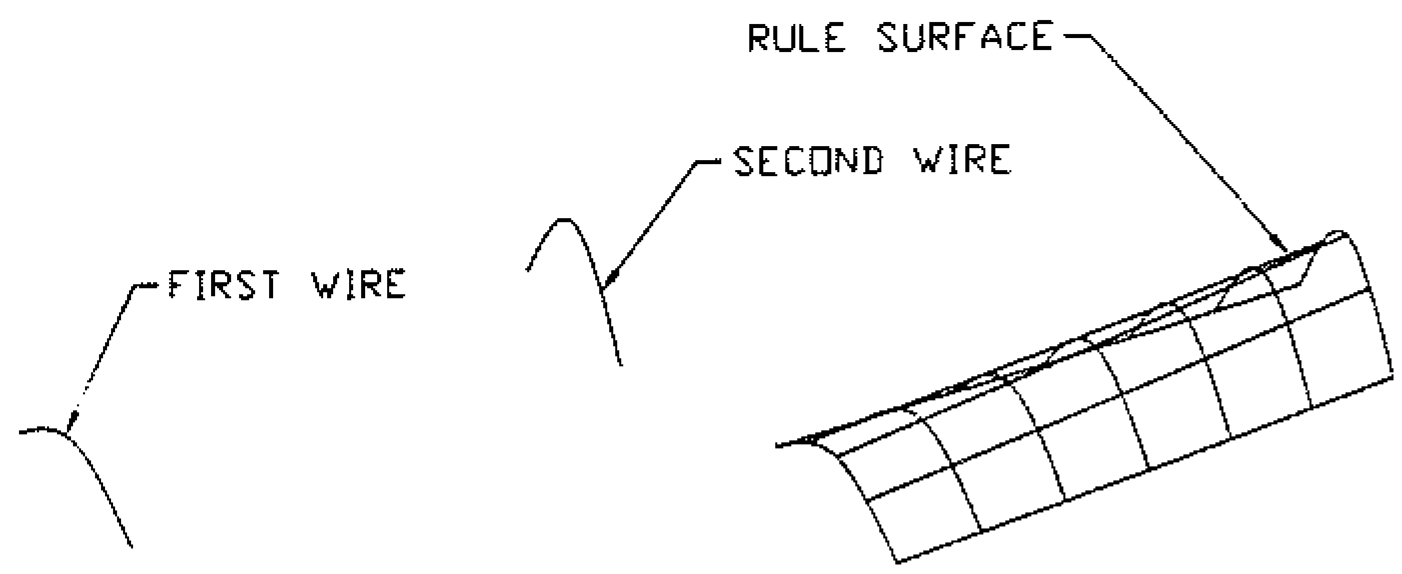


Figure 2.5 Rule surface

To make a surface that has a variable cross section, you use three or more U-wires as input wires. The resulting surface is called a loft u surface. Its cross section in the U direction will change smoothly from the first wire to the second, and then from the second wire to the third. If you have a fourth wire, the cross section changes from the third wire to the fourth as well. As a result, the cross section in the V direction also changes smoothly from one edge to the other. Figure 2.6 shows a loft u surface constructed from three U-wires.

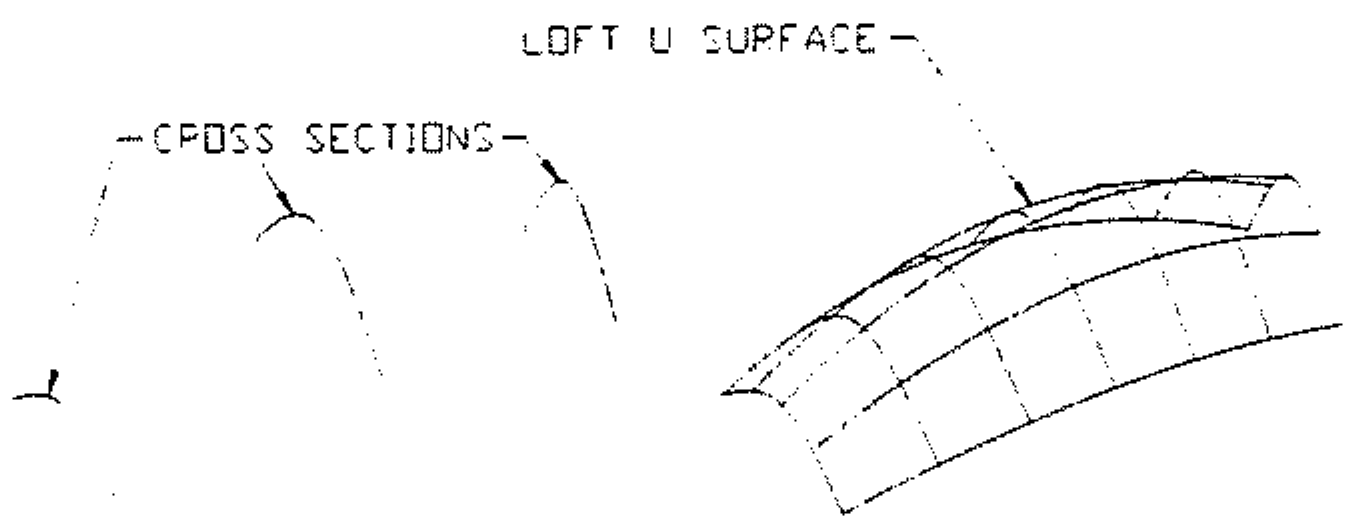


Figure 2.6 Loft u surface

In Figure 2.6, the cross sections of a loft u surface in the V direction are smooth spline curves that pass through the U-wires. Their shapes are determined by the ways the U-wires change from one section to another. To exercise more control over the contours of the surface in the V direction, you specify one or two rails for the U-wires to transit. The surface is called a sweep surface. Figure 2.7 shows a sweep surface with three cross sections and a single rail.

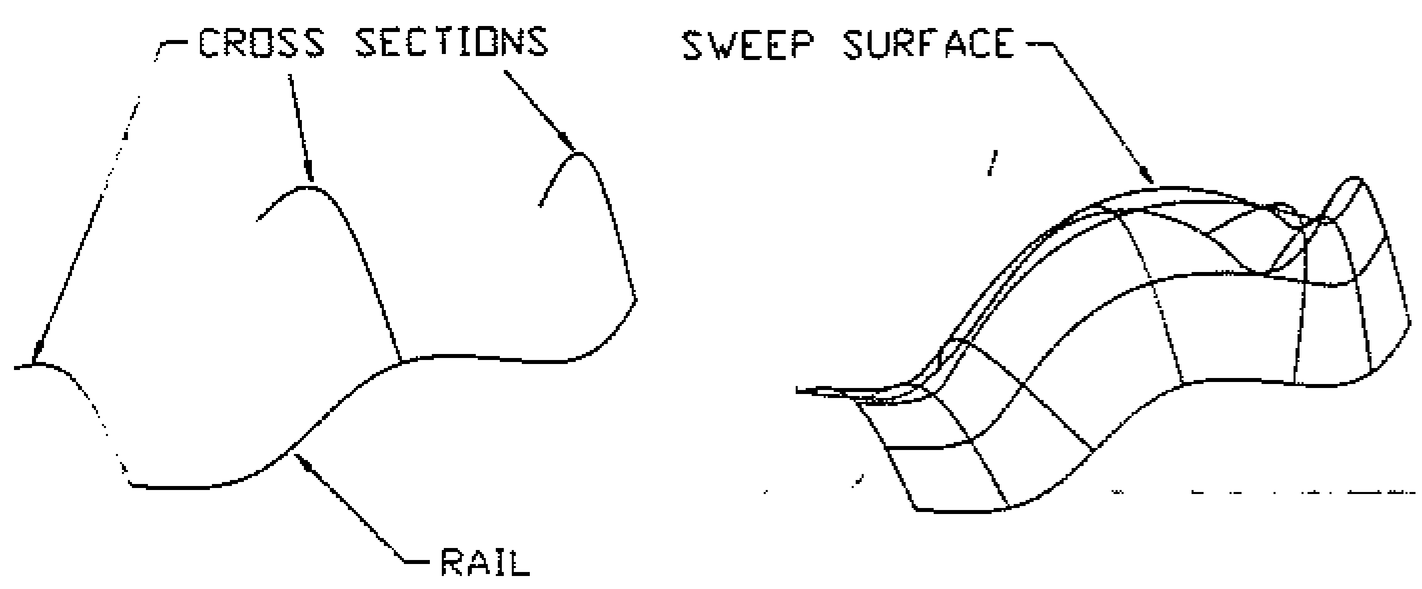


Figure 2.7 Sweep surface with a single rail

For the surfaces shown in Figure 2.6 and Figure 2.7, the U-wires are the same but the shapes are different. Here, you can see how the rail controls the transition of the U-wires as they change from one section to another. In Figure 2.7, a single rail is used to control one end of each U-wire. To control both ends, you use two rails. When you use two rails, the U-wires' cross sections change in two ways. First, they change from one section to another. Second, they change shape according to the distance between the rails. Figure 2.8 shows a sweep surface with two rails. Compare Figure 2.8 with Figure 2.7 to see the difference.

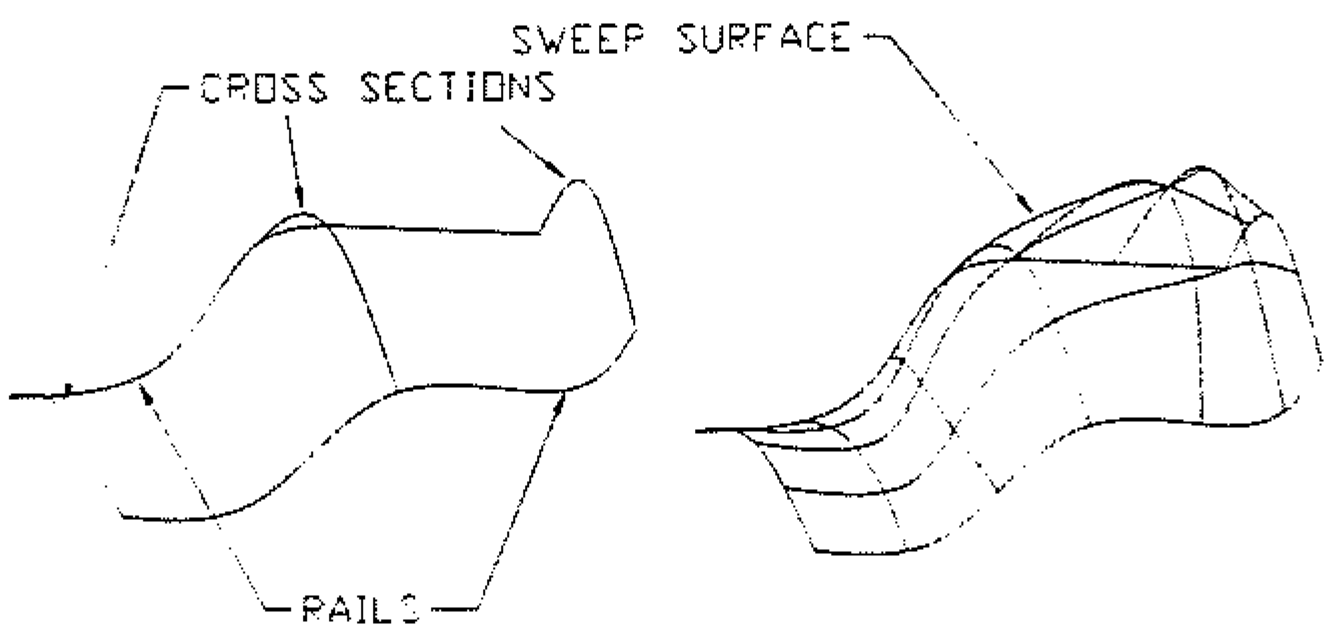


Figure 2.8 Sweep surface with two rails

In a sweep surface, the rails control the cross sections in the V direction. To control the V direction cross sections fully, you specify two sets of wires, one set of U-wires and one set of V-wires. The resulting surface is called a loft uv surface. (See Figure 2.9.)

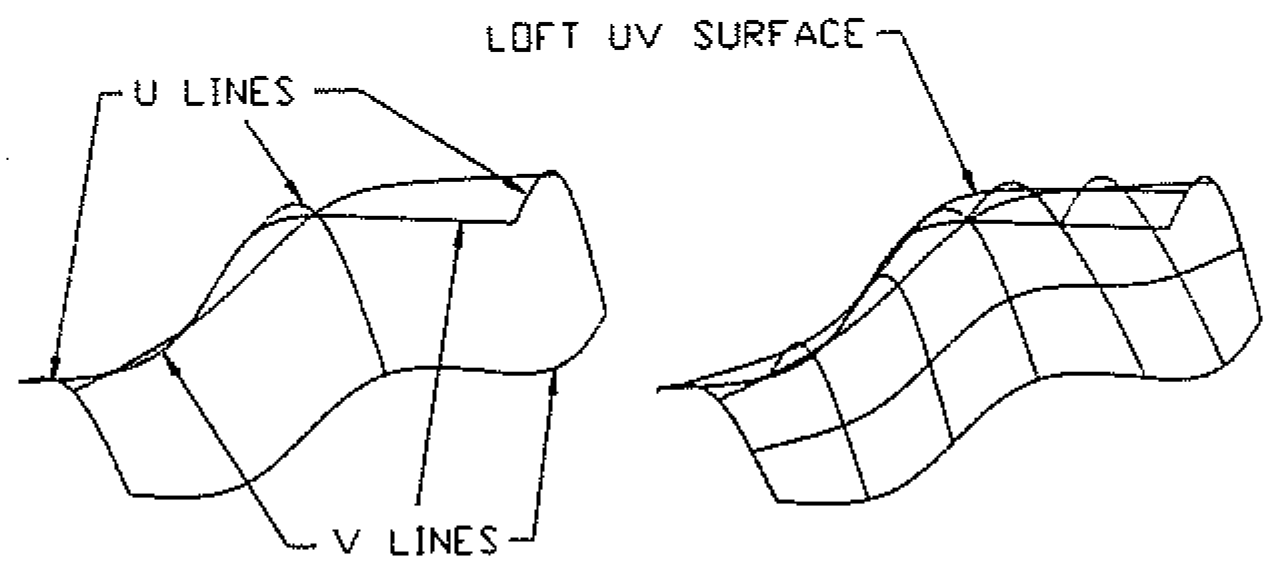


Figure 2.9 Loft uv surface

Derived Surfaces

Derived surfaces include the fillet surface, the corner surface, the blend surface, and the offset surface.

A fillet surface treats the edges of two intersecting surfaces by providing a connecting surface with a circular cross section. Figure 2.10 shows a fillet surface of constant radius formed between two planar surfaces.

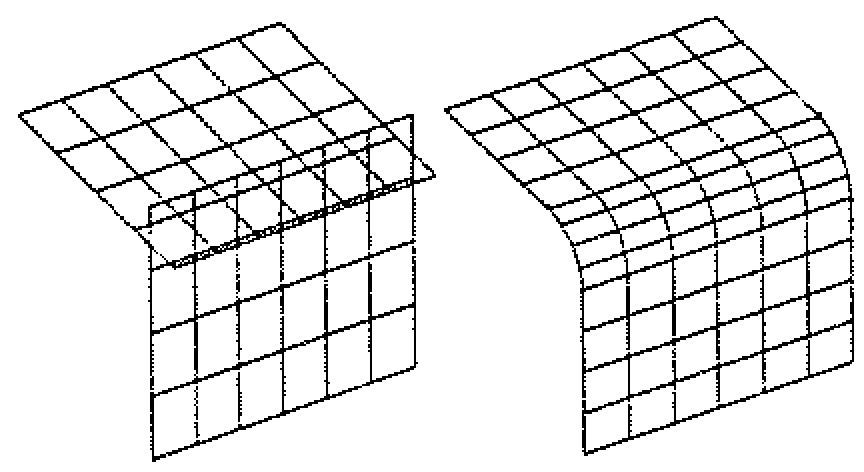


Figure 2.10 Constant-radius fillet surface

When you make a fillet surface, you can set the fillet radius to vary linearly or cubically. In a linear variable fillet surface, the fillet radius changes linearly from one set value to another set value. In a cubical variable fillet surface, the fillet radius changes cubically from one set value to another. (See Figure 2.11.)

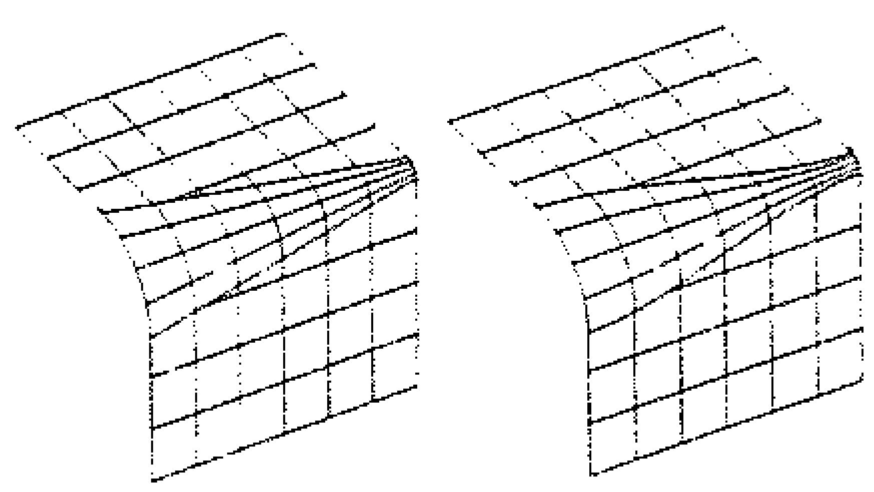


Figure 2.11 Linear variable fillet (left) and cubical variable fillet (right)

A fillet surface is used to treat two intersecting surfaces. If you have three intersecting surfaces to treat, you can form fillets in pairs, then treat the intersecting fillets with a corner fillet. (See Figure 2.12.)

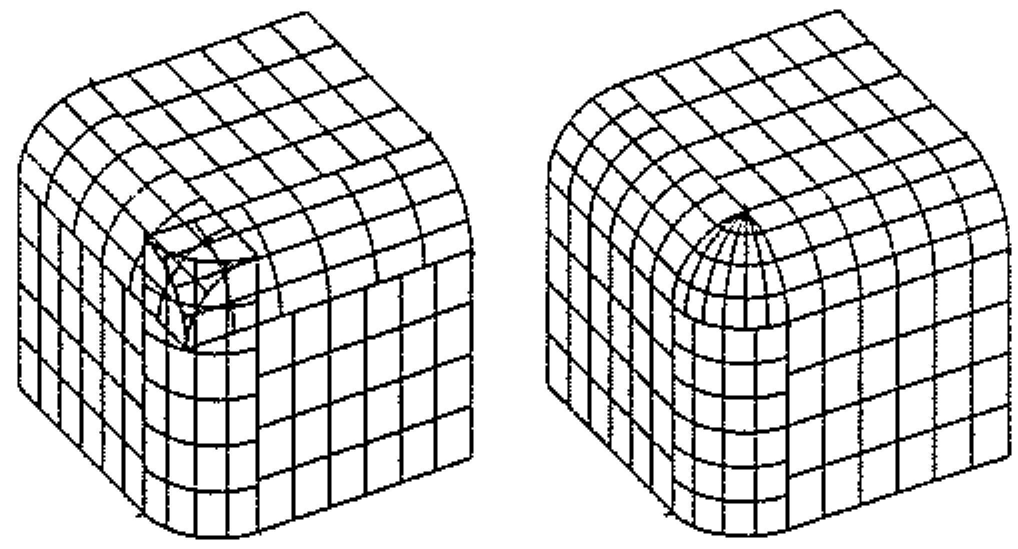


Figure 2.12 Intersecting fillet surfaces (left) and corner fillet surface (right)

When you make a surface model, two adjacent surfaces do not always have to intersect. To treat the joint between two nonintersecting surfaces, you can fill in the gap between them by blending. Figure 2.13 shows a blend surface formed between two nonintersecting surfaces.

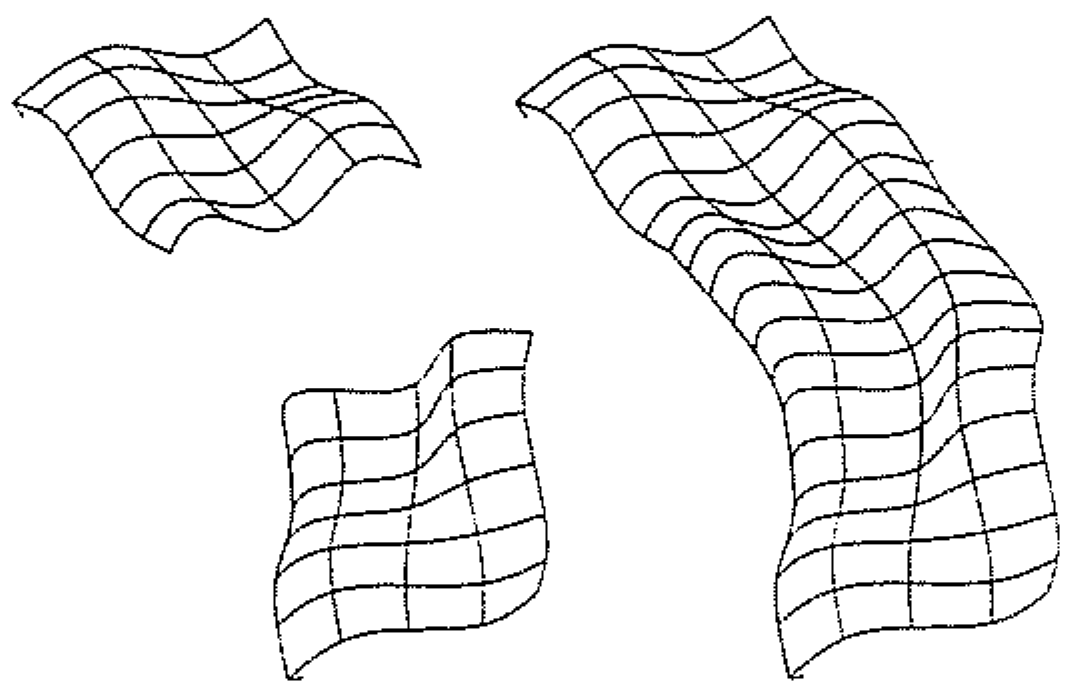


Figure 2.13 Nonintersecting surfaces (left) and the blend surface formed between them

You can also blend three or four surfaces. (See Figures 2.14 and 2.15.)

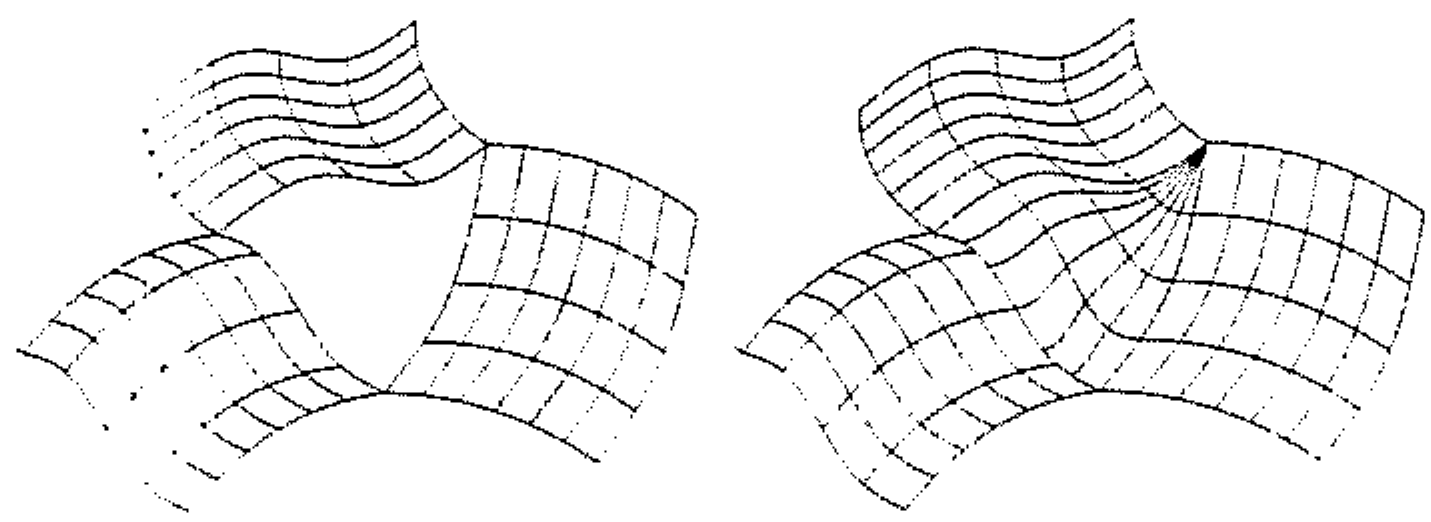


Figure 2.14 Blend surface among three edges

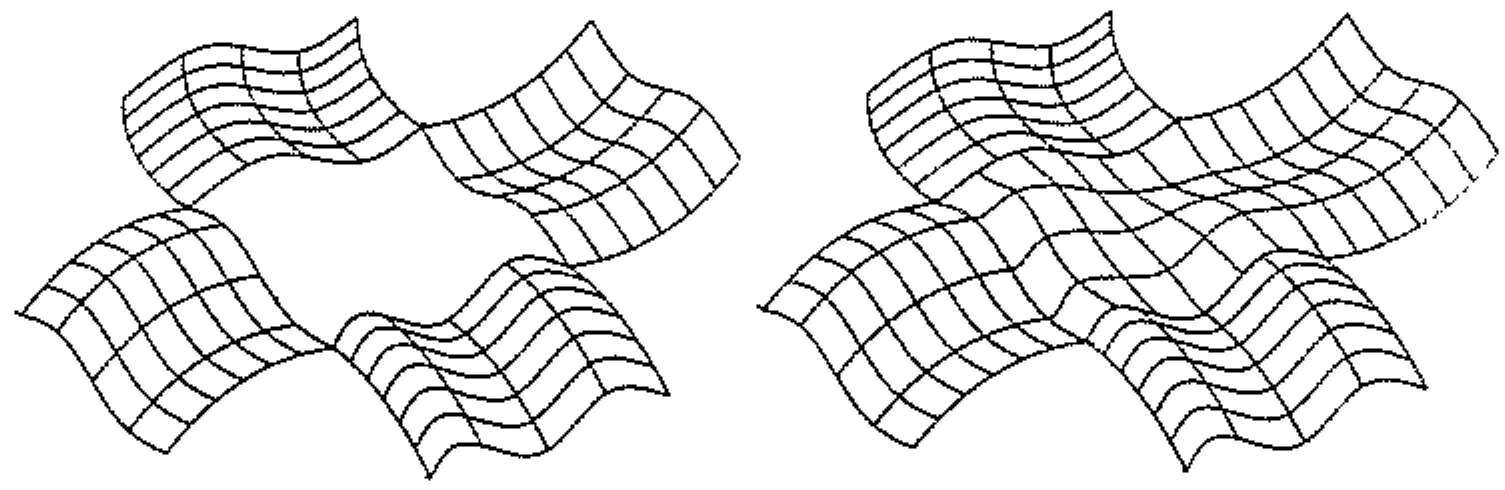


Figure 2.15 Blend surface among four edges

Sometimes you may have to construct a surface to run at a constant distance from another surface. To make such a second surface, you simply derive an offset surface from an existing surface. Figure 2.16 shows an offset surface constructed to form the earpiece of a walkie-talkie.

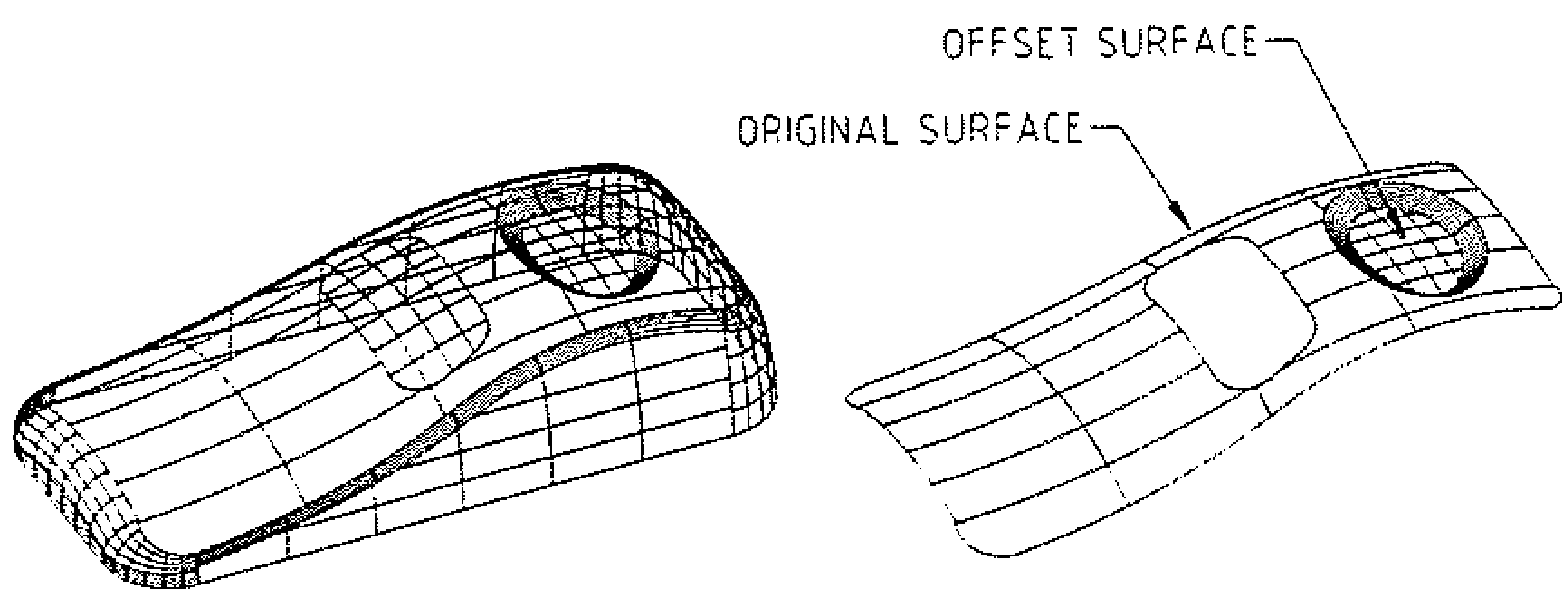


Figure 2.16 Offset surface

Trimmed Surfaces

To produce a smooth surface, it is necessary to use smooth defining wires and smooth boundary lines. However, most of the surfaces that we use to compose a design do not necessarily have smooth boundaries, although they have smooth profiles.

Figure 2.17 shows the surface model of an automobile body panel. This is a smooth surface, but its boundary is irregular.

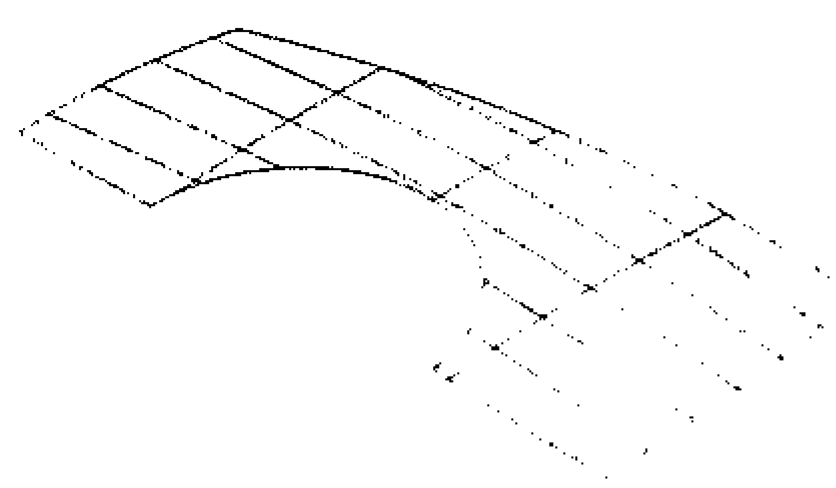


Figure 2.17 Surface model of an automobile body panel

Given this problem, you might intuitively use the boundary curves that you see as the defining wires to construct the surface model. If you did, you would probably get an irregular surface like the one in Figure 2.18.

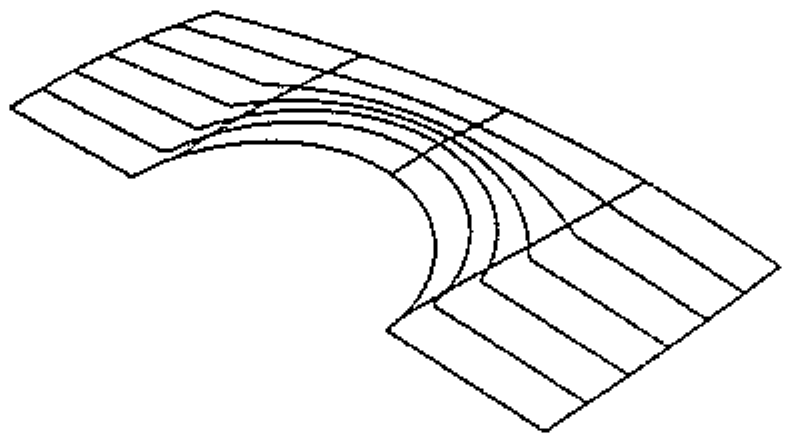


Figure 2.18 Irregular surface defined by irregular boundaries

Obviously, the surface shown in Figure 2.18 is not the one we want (shown in Figure 2.17). What has gone wrong? The answer is that a set of irregular curves will generate an irregular surface. Unless the boundary lines are smooth wires, they cannot be used as defining curves for the surface.

To obtain a smooth surface with an irregular boundary, you have to perform two steps. You use a set of smooth wires to produce a smooth surface that is much larger than the required surface. This is called the base surface. Then you use the irregular boundary curve to trim the smooth surface. In the computer, the resulting surface will consist of the original untrimmed smooth surface with smooth boundaries and the irregular boundaries. Although both of these are saved in the database, only the boundary and the remaining part of the trimmed surface are displayed. As a result, we obtain a smooth free-form surface with irregular boundaries. This is called a trimmed surface.

To produce a free-form surface that is large enough for subsequent trimming, you must define a set of curves that encompass the required surface. To make such wires, you need to be able to visualize the defining curves that are outside the required surface.

In Figure 2.19, the construction of the smooth automobile body panel starts from a set of smooth wires. From the smooth wires, a smooth surface that is much larger than the required surface is made. To obtain the required surface, an irregular boundary is used to trim the large smooth surface.

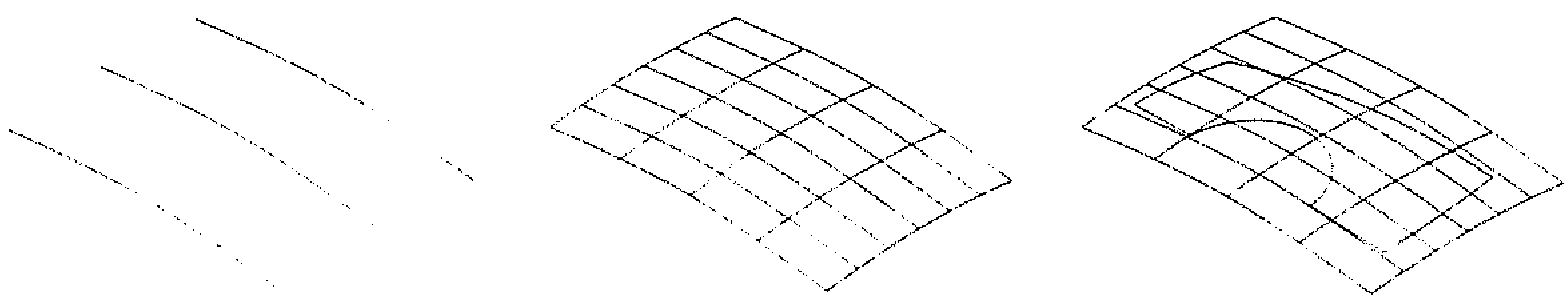


Figure 2.19 Defining curves (left), the untrimmed surface (center), and the irregular boundary (right)

To reiterate, a smooth surface needs to be constructed from smooth defining wires. Smooth surfaces with irregular edges occur in many designs. If you use the irregular edges to construct the surface directly, you will get a surface with many sudden changes in curvature. The surface will not be smooth at all. To obtain a smooth surface with irregular edges, you build a larger surface from smooth wires. Then you trim the smooth surface with the irregular edges. The resulting surface is a trimmed surface.

To sum up, a trimmed surface retains its original smooth defining boundaries in the database while possessing a new trimmed boundary. The original surface is called the base surface, and the trimmed boundary is called the trim edges. The integrated data definition of the new trimmed surface contains both the base surface and the trim edges.

Converted Surfaces

In addition to making primitive surfaces, free-form surfaces, derived surfaces, and trimmed surfaces, you can construct surfaces by converting existing arcs, circles, lines, and polylines with thickness, AutoCAD solids, and Mechanical Desktop solid parts.

Although the conversion tool is very handy, it is not advisable to start making surfaces by conversion. However, you may find it easier to construct the basic shape of a surface model from a complex AutoCAD native solid or a Mechanical Desktop parametric solid. Figure 2.20 shows a native solid used as the starting point to make a surface model. To fully utilize a converted object, you can add details by using other surface modeling tools.

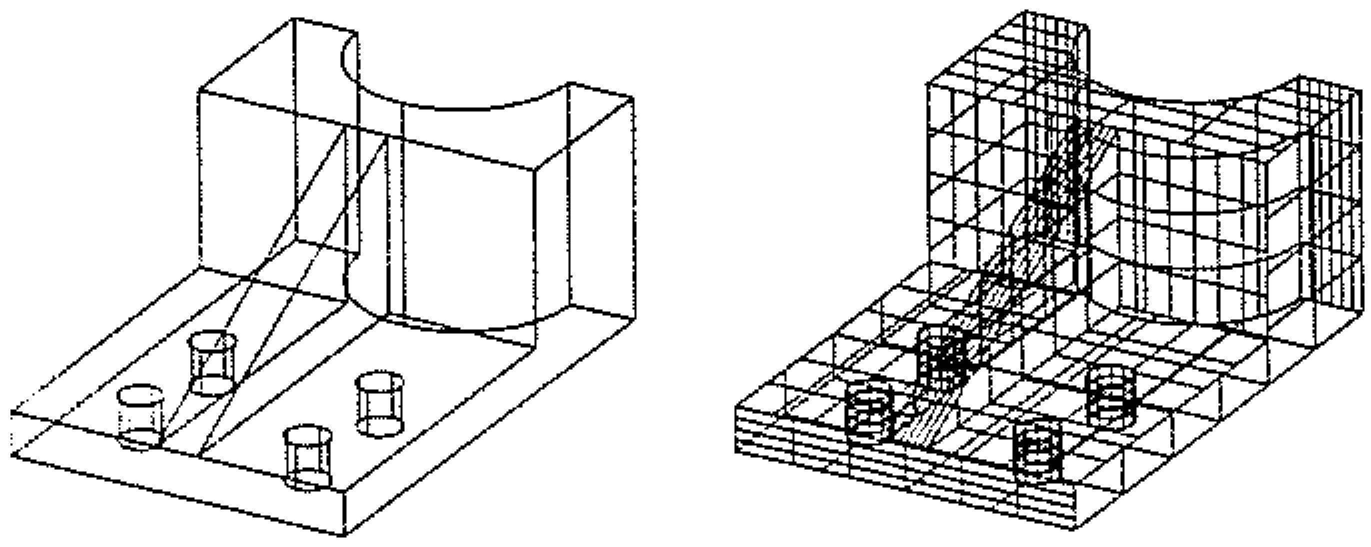


Figure 2.20 Complex native solid converted to a set of surfaces

Surface Editing

In making surface models, you have to edit as well as to construct. Editing processes that you can perform on a surface are

- Breaking a surface
- Joining two or more surfaces
- Lengthening a surface
- Scaling a surface
- Intersecting two surfaces
- Projecting and trimming a surface
- Untrimming a surface
- Truncating a surface
- Refining a surface
- Changing the grip points of a surface
- Modifying the span of a surface
- Flipping the normal direction of a surface

Breaking a Surface

You can break a surface into two surfaces along a selected line. After breaking, the surfaces still maintain the original continuity. In Figure 2.21, the profiles and silhouettes of the broken surfaces (right side) will be the same as those of the original surface (left side).

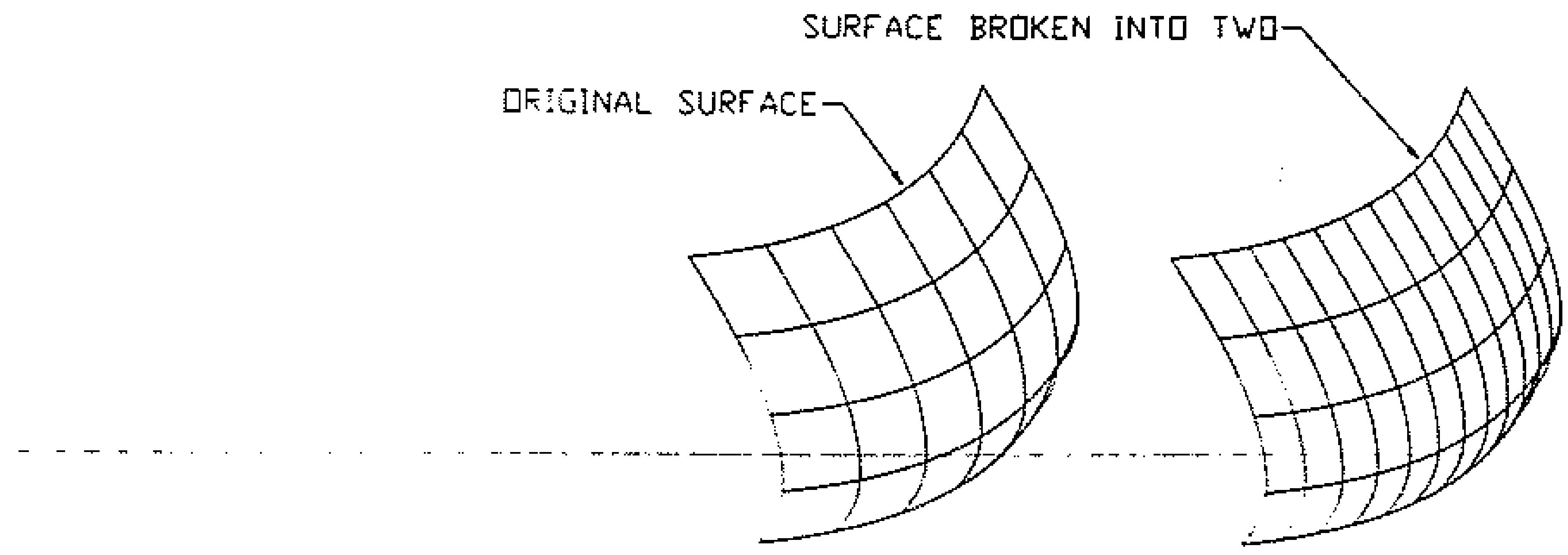


Figure 2.21 A single surface broken into two surfaces

Joining Two or More Surfaces

You can also join two or more surfaces together to form a single surface so that you can handle them together. (See Figure 2.22.)

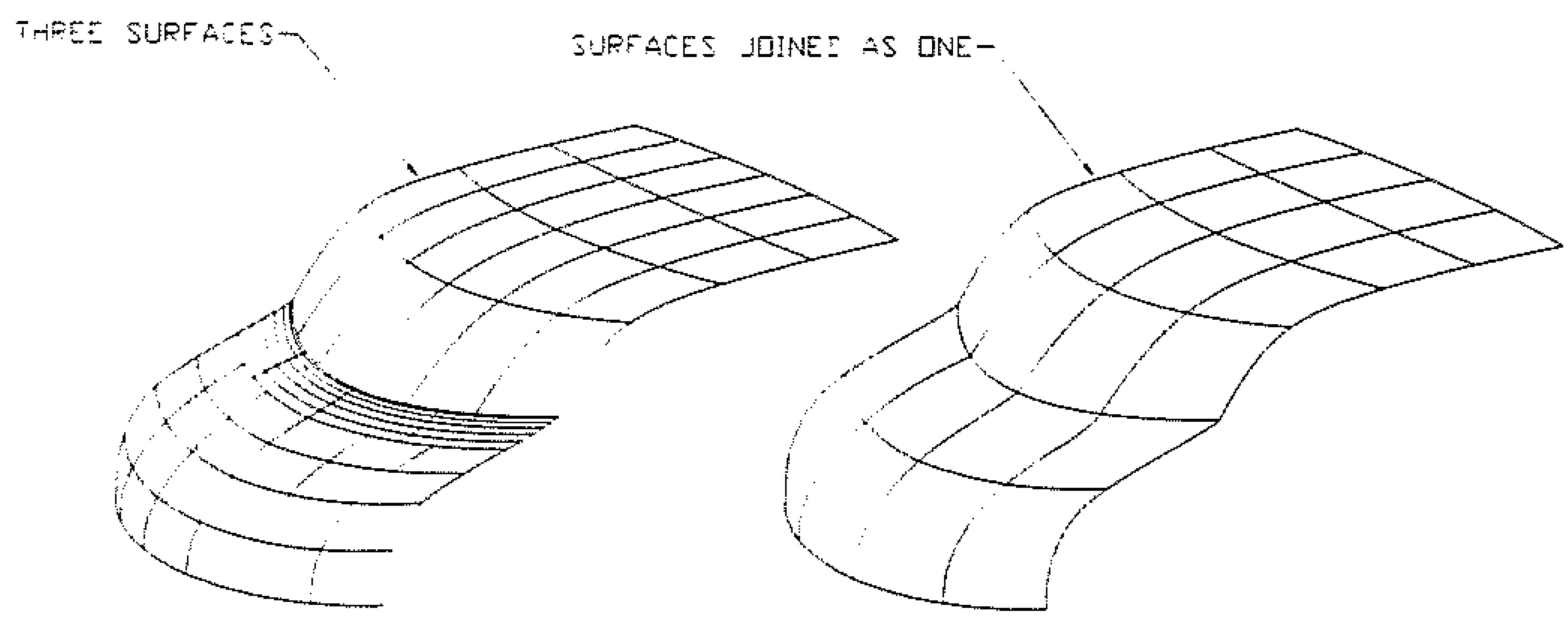


Figure 2.22 Three surfaces joined together to form one single surface

Lengthening a Surface

If you find a surface is too small, you can enlarge it by lengthening it along its untrimmed edge. (See Figure 2.23.)

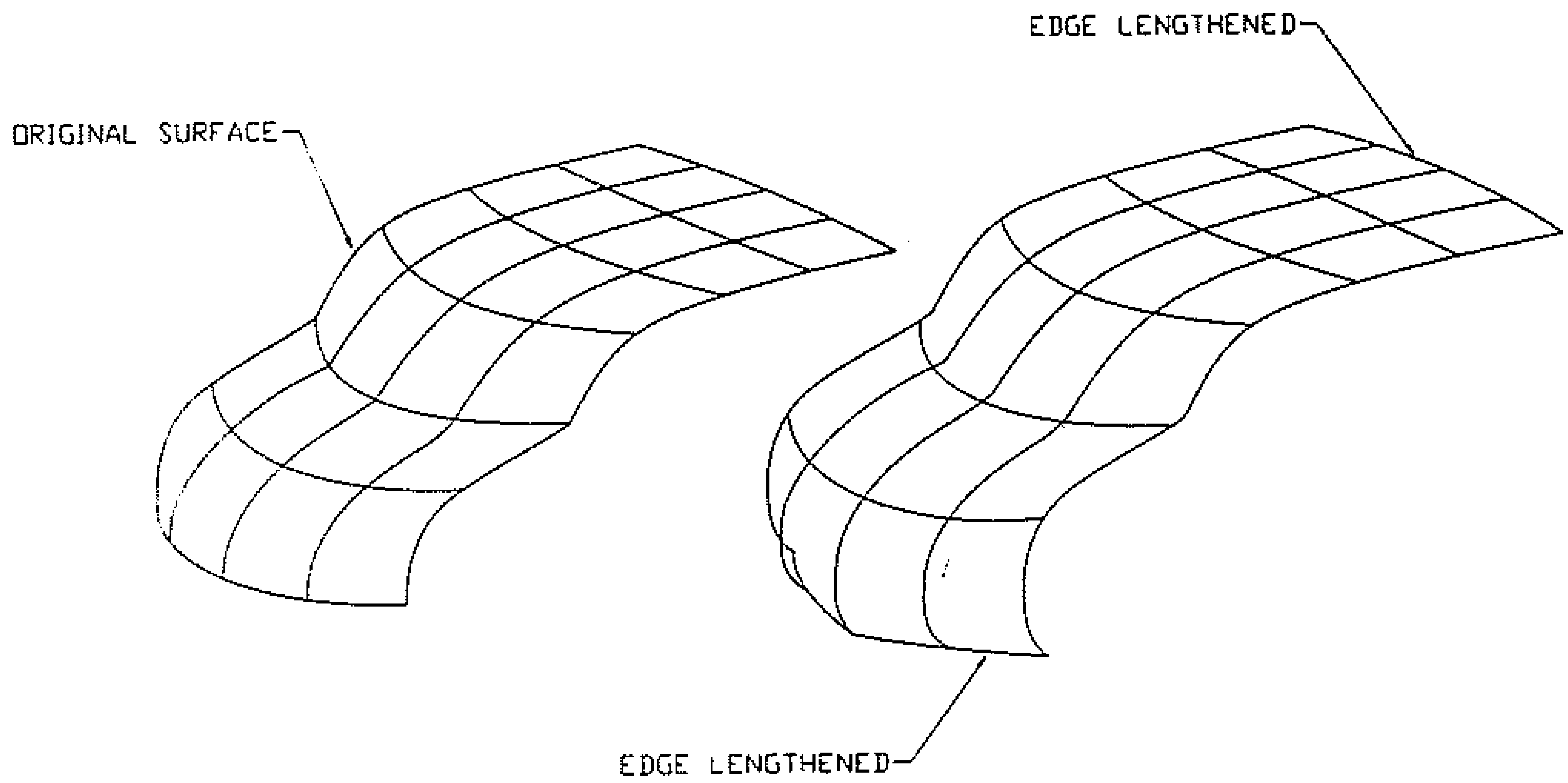


Figure 2.23 Untrimmed edges of a surface lengthened

Scaling a Surface

To change the size of a surface and yet maintain the overall proportions of the profiles and contours, you can scale it in 3D. (See Figure 2.24.)

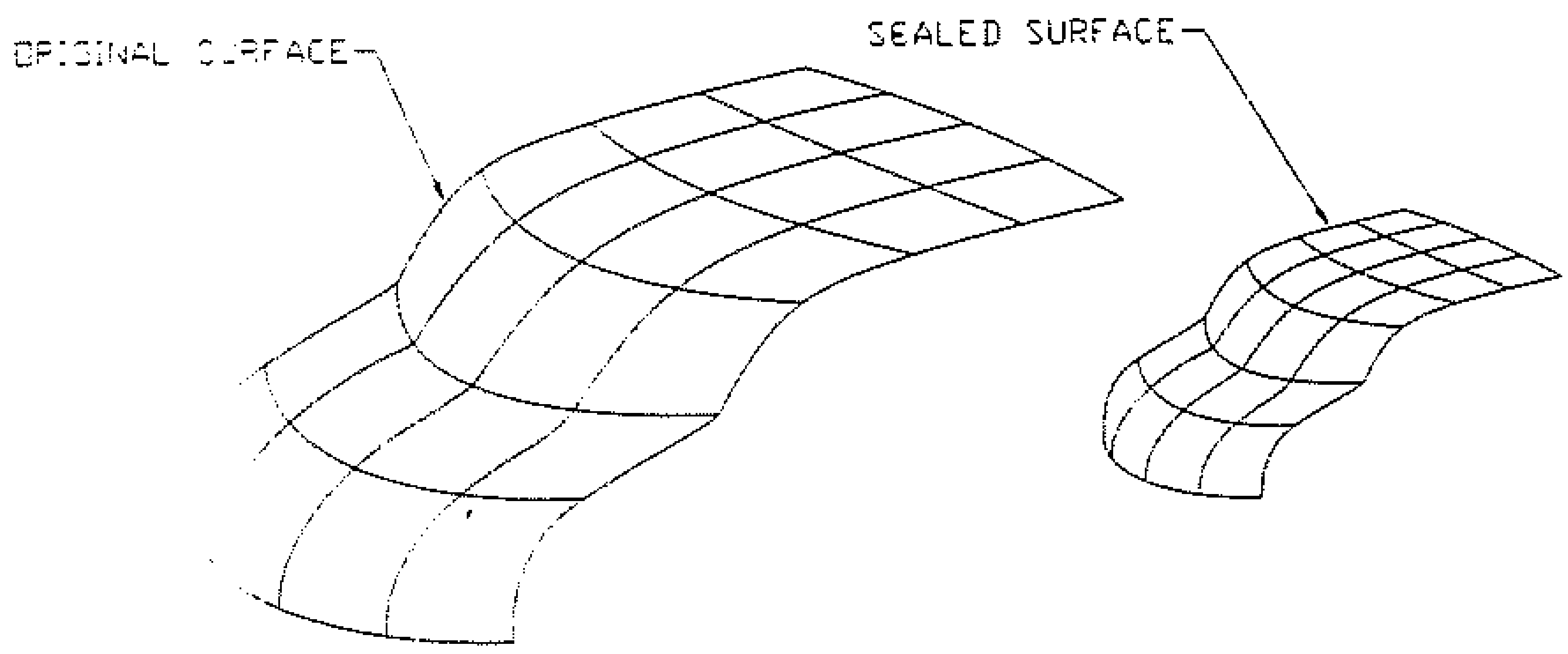


Figure 2.24 Surface scaled

Intersecting Two Surfaces

For two intersecting surfaces, you can trim the unwanted portions of the surfaces away to form a sharp edge at the intersection. (See Figure 2.25.)

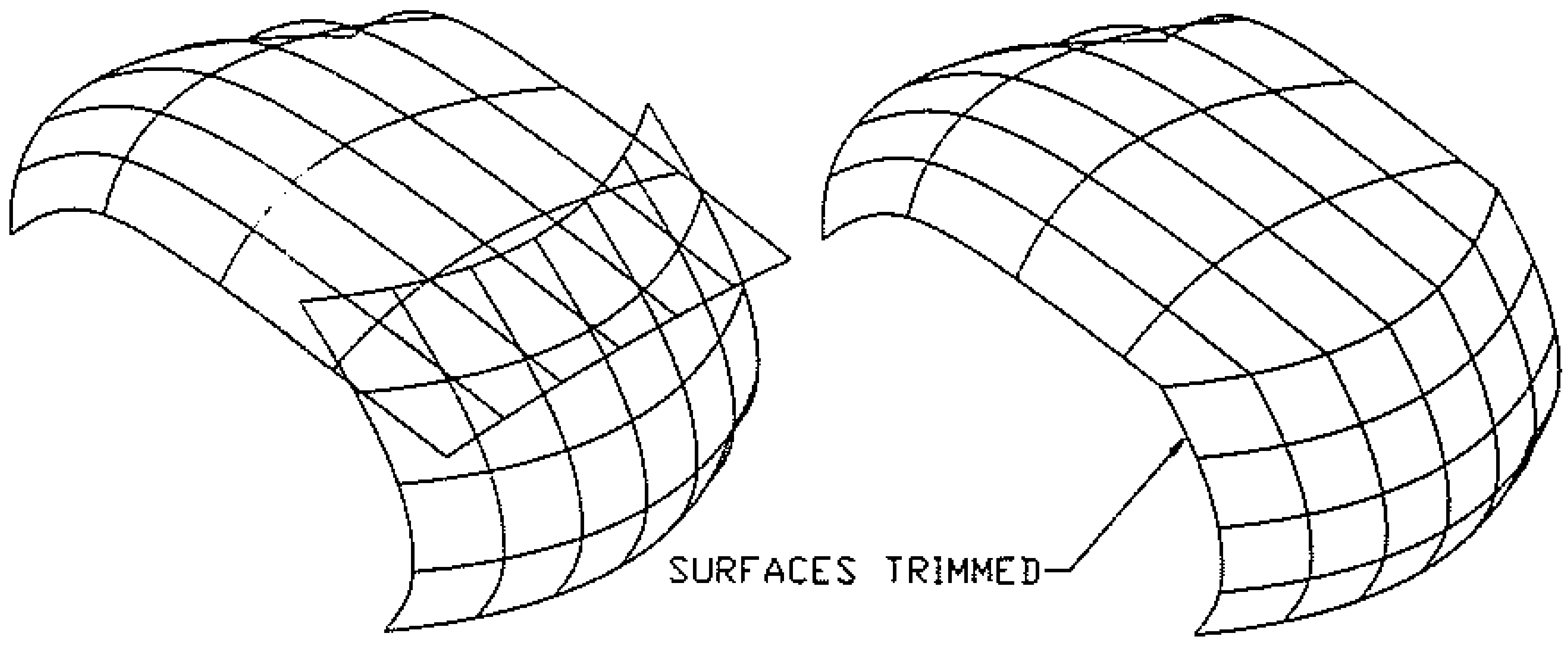


Figure 2.25 Two intersecting surfaces trimming each other

Projecting and Trimming a Surface

To cut a hole in a surface or to give an irregular edge to a surface, you can project a 2D or 3D wire and trim the surface. This was explained in detail earlier. (See Figure 2.26.)

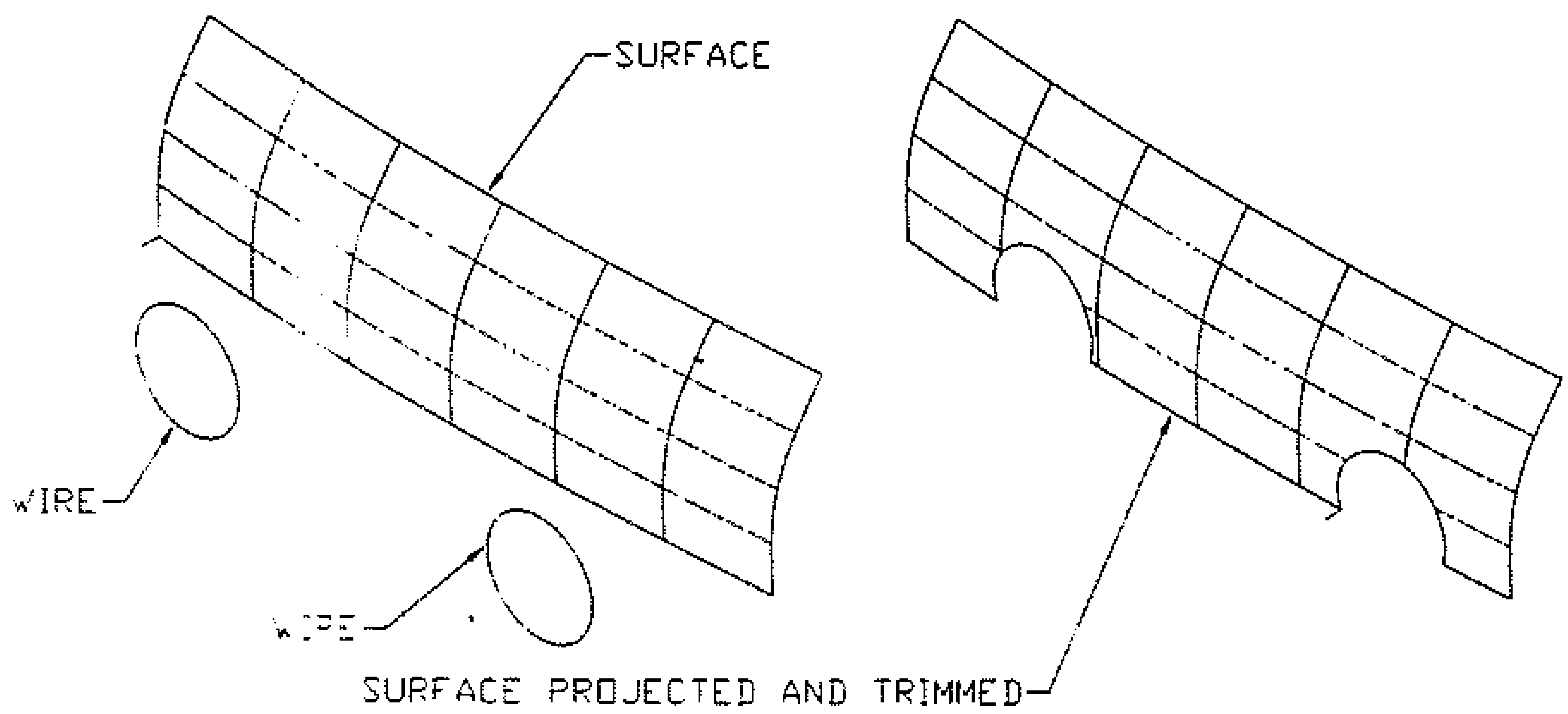


Figure 2.26 Surface trimmed by projected wires

Untrimming a Surface

We all make mistakes and sometimes want to change our minds. You may need to change a trimmed surface back to its original untrimmed state. Because a trimmed surface in the computer consists of the original base surface and the trimmed edge, you can remove its trimmed boundary to change it back to its untrimmed state. (See Figure 2.27.)

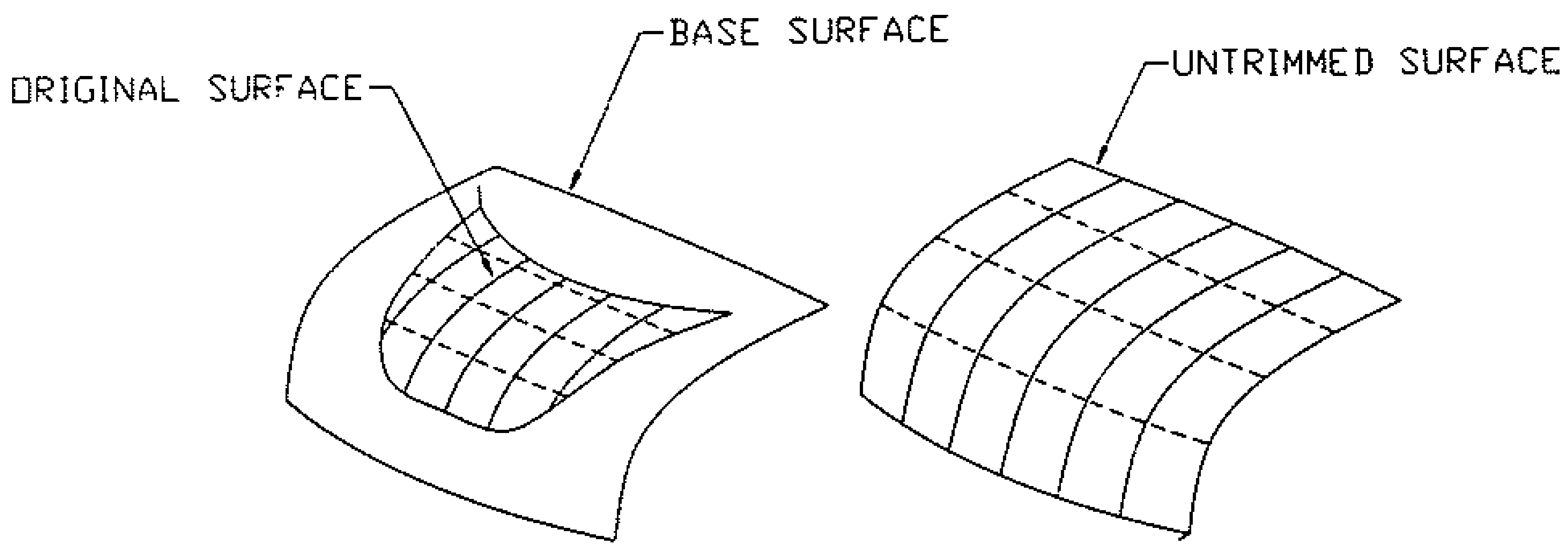


Figure 2.27 Removal of trimmed boundary

Truncating a Surface

To reiterate, a trimmed surface consists of a base surface and a trimmed boundary. To make a smooth surface with an irregular edge, you construct a smooth base surface that is large enough to encompass the trimmed edge. Sometimes you may use a base surface that is much larger than required. If so, unnecessary memory space is wasted to store the unwanted part of the base surface. To reduce the memory used, you can truncate the base surface of a trimmed surface. Figure 2.28 shows the original base surface and the truncated base surface.

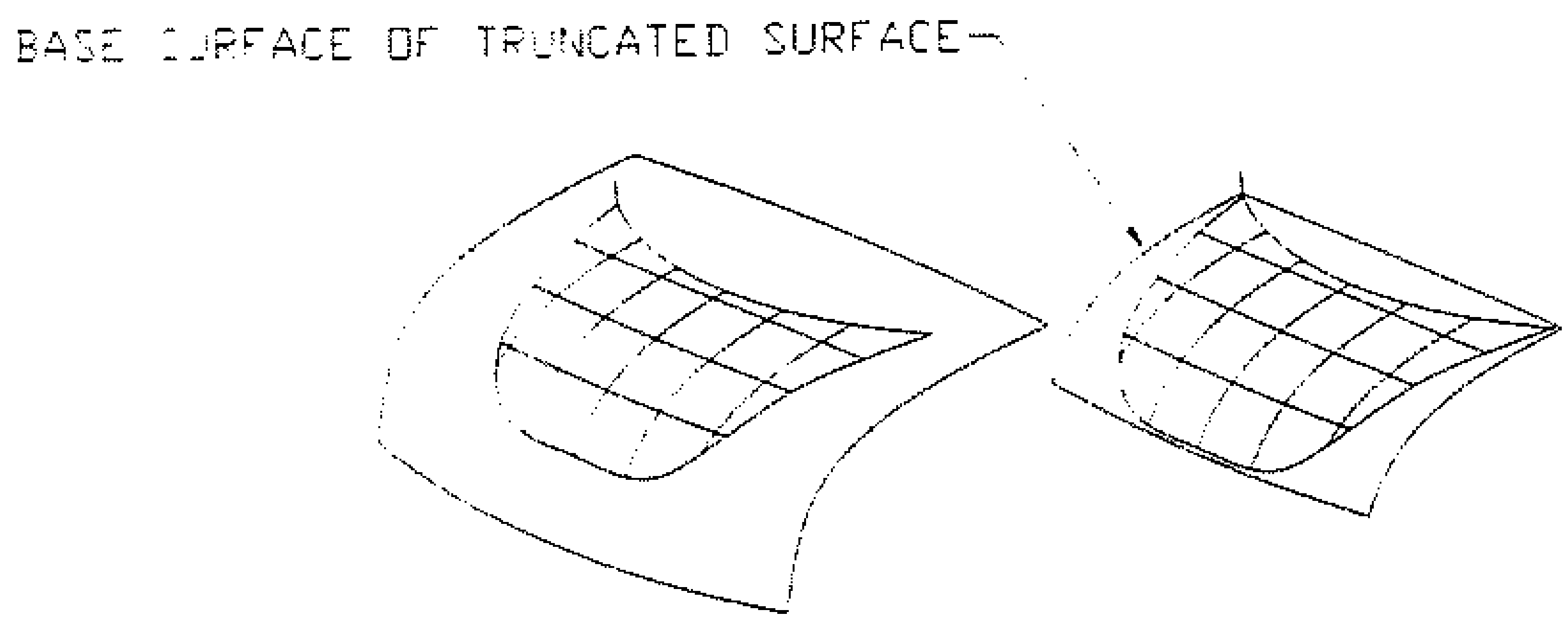


Figure 2.28 Truncation of base surface

Refining a Surface

The accuracy of a surface is determined by the number of UV patches. You can refine a surface by changing its UV patches. Reducing the number of patches decreases the accuracy of a surface. Figure 2.29 shows how surface accuracy in relation to the wires is affected by the number of UV patches.

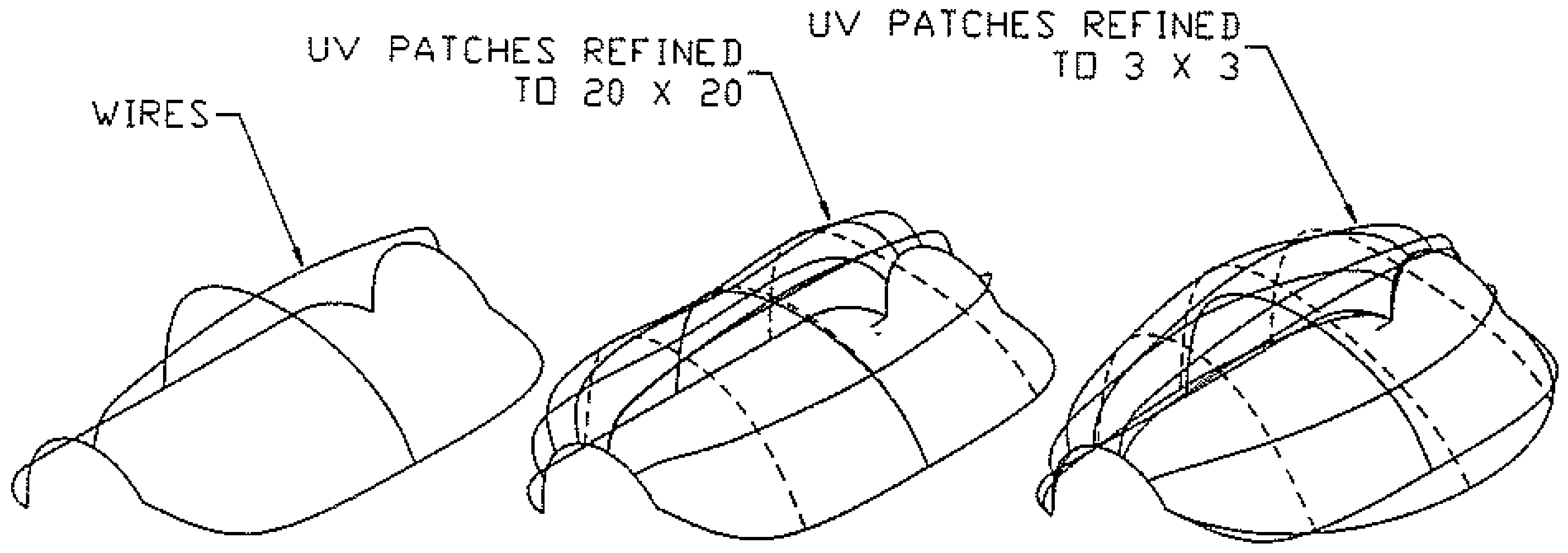


Figure 2.29 Accuracy of a surface in relation to the number of UV patches

Changing the Grip Points of a Surface

Grip points on a surface are locations where you can grip and pull in order to change the surface's profiles and silhouettes. Figure 2.30 shows two different grip point settings.

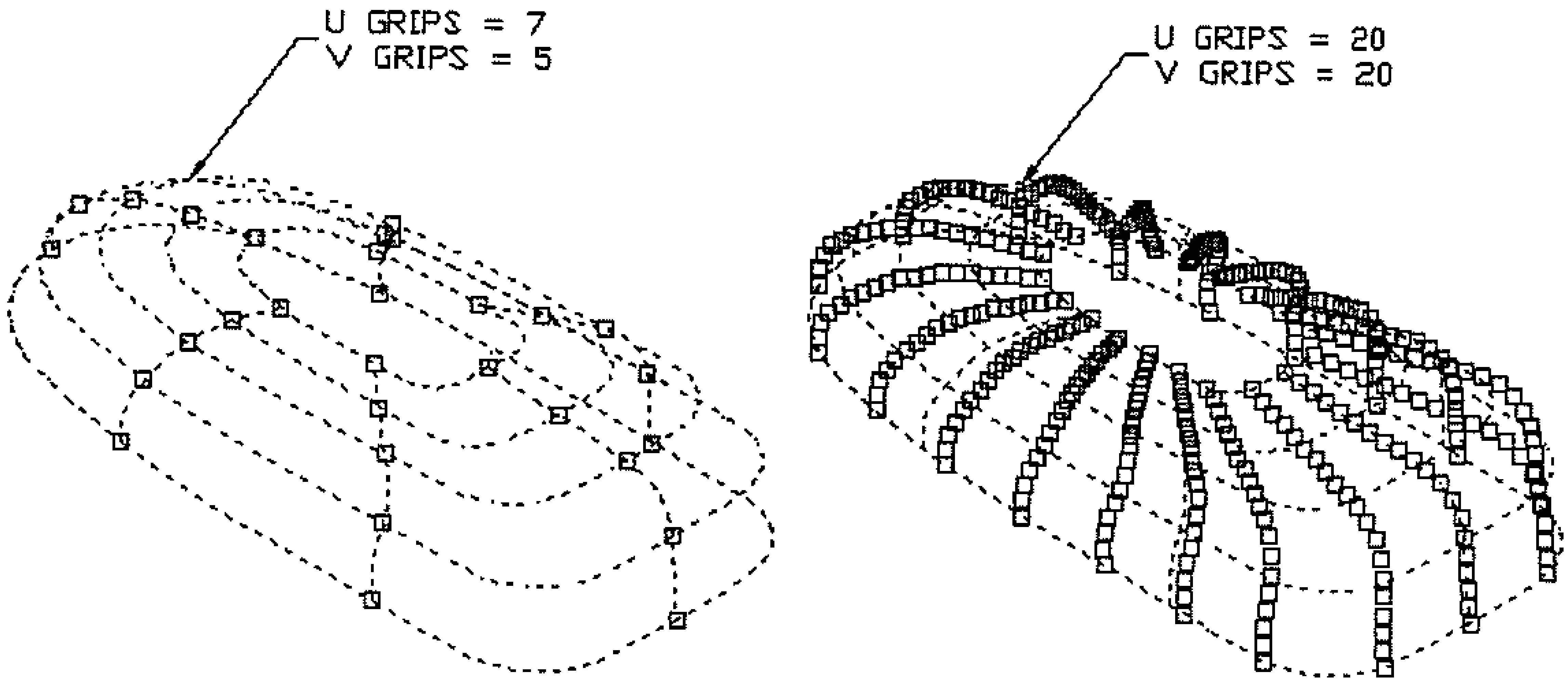


Figure 2.30 Grip point settings

Modifying the Span of a Surface

The span of a surface defines a circular area that a surface will deform when a grip point of the surface is pulled. Changing the span affects the way the surface is deformed when a grip point is pulled. (See Figure 2.31.)

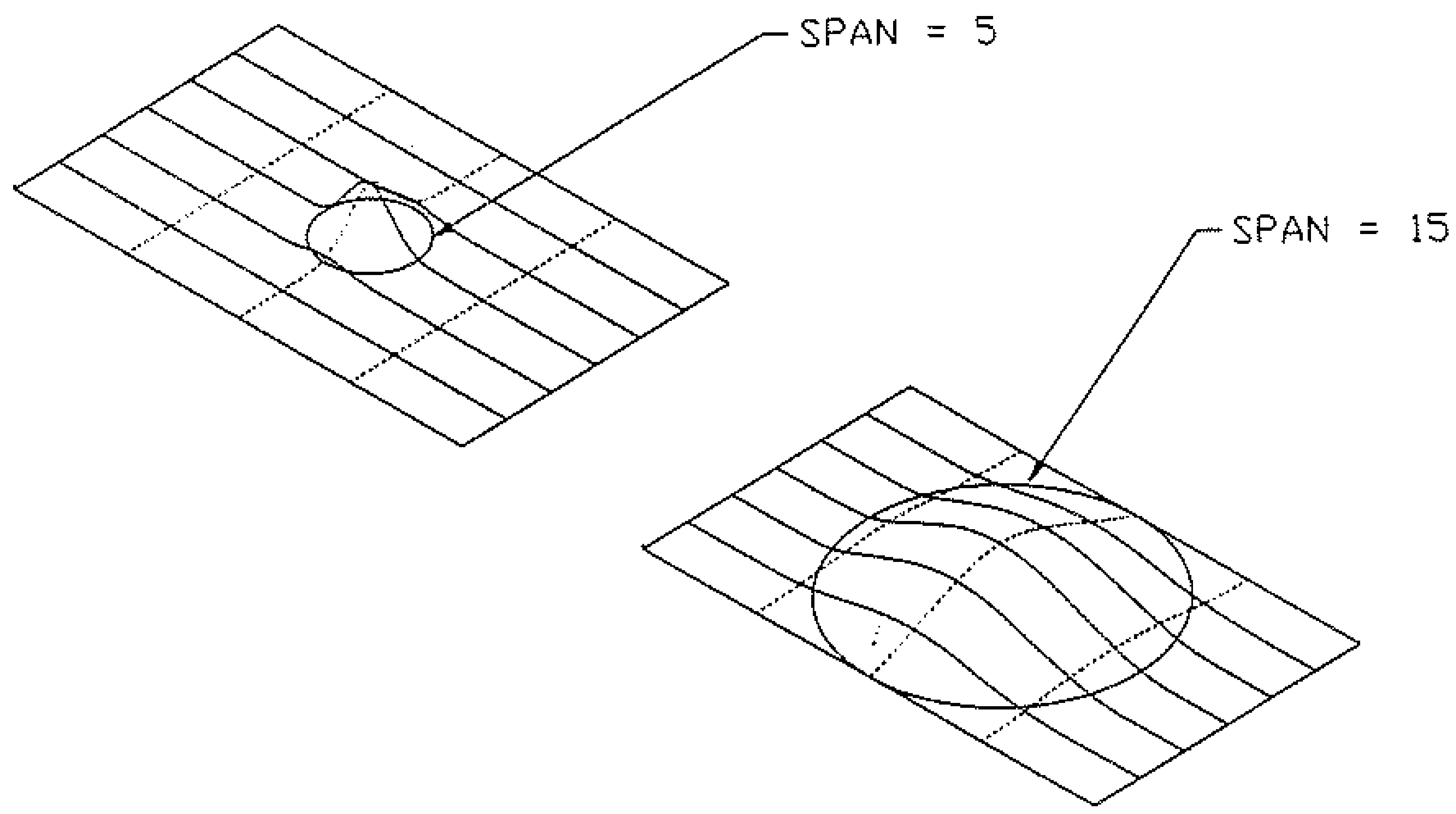


Figure 2.31 Span size and the surface affected when a grip point is pulled

Flipping the Normal Direction of a Surface

A surface has no thickness. To represent a 3D object in a computer, you need a number of surfaces. For the computer (and later applications) to recognize which side of the surface represents a void and which side represents a volume, a normal vector is used. Figure 2.32 shows the vector normal flipped.

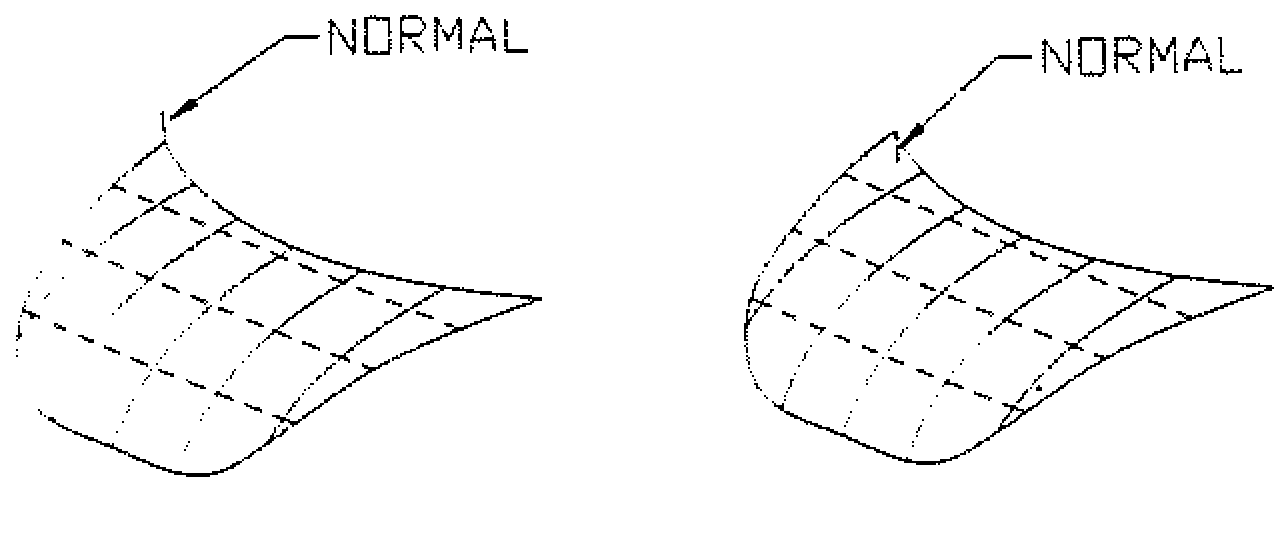


Figure 2.32 Normal direction flipped

Constructing a Surface Model

A surface model in a computer is a representation of a 3D object by a set of surfaces. To construct the surface model of a given object, you should first determine what kinds of surfaces are needed. You may consider using primitive surfaces, free-form surfaces, derived surfaces, trimmed surfaces, or even converted surfaces (if you already have such objects in your drawing file).

Among the various kinds of surfaces, free-form surfaces are the ones most commonly used. As we have said earlier, all free-form surfaces have one thing in common: They all need to be constructed from smooth wires. Therefore, the first task that you need to tackle in surface modeling is to think about what kinds of wires are needed and how they can be constructed. After making the wires, you then let the computer generate the required surfaces.

While you are designing and making a surface model, it is intuitive to start thinking about the surfaces but not the wires. However, the computer constructs surfaces from defining wires. Therefore, you need to analyze the surface to determine what wires are required. After that, you construct the wires and let the computer generate the surfaces from the wires. In the surface modeling projects of this chapter, you will learn to construct 3D wires as well as 3D surfaces. Here, the particulars of all the wires are given to you. While working on these projects, you should try to relate the 3D wires to the 3D surfaces. It is hoped that you can reverse the process, seeing the wires when a surface is given.

2.2 Wire Construction Tools

To make free-form surfaces, derived surfaces, and trimmed surfaces, 3D wires are needed. To construct 3D wires, you can use AutoCAD commands. In addition, you can use Mechanical Desktop commands to perform the following tasks:

- Constructing augmented lines
- Joining wires to form augmented lines, polylines, or splines
- Fitting wires into splines
- Changing a spline back to a polyline by unsplining
- Copying the edges of existing surfaces
- Obtaining flow lines from existing surfaces
- Generating a parting line
- Cutting a series of section lines
- Constructing a wire at the intersection of two surfaces
- Projecting a wire onto a surface to obtain another wire
- Offsetting 3D wires
- Filleting 3D wires
- Changing the direction of a wire
- Refining a wire

Constructing Augmented Lines

An augmented line is a special kind of wire along which normal vectors are placed at regular intervals.

You can make use of the normal vectors of the augmented lines to control machines that operate in 4-axis or 5-axis. You can also take an augmented line as a rail and use its normal vector to guide the transition of the section wires for making a sweep surface.

Figure 2.33 shows an augmented line generated along the edge of a free-form surface.

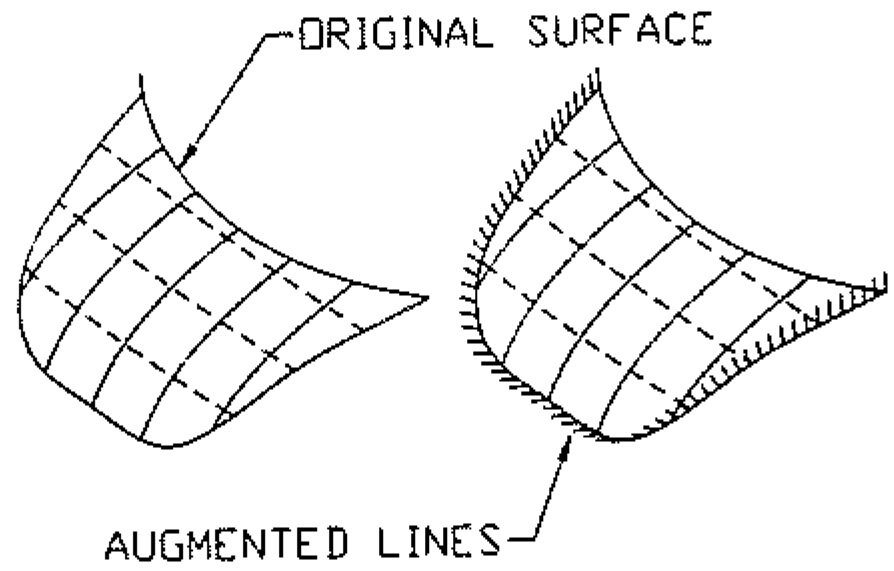


Figure 2.33 Augmented lines constructed along the edge of a free-form surface

Joining Wires to Form Augmented Lines, Polylines, or Splines

To construct free-form surfaces, you need wires. To make a wire in accordance with any specific shape you have in mind, you may find it easier to construct lines, arcs, and circles than to construct splines. To use connected wires of this kind, you can join them together to form a 3D polyline, spline, or augmented line.

Fitting Wires into Splines

Instead of joining wires together, you can fit a polyline/line/arc into a spline. Both joining and fitting create splines from existing wires. The difference between them is that joining constructs a single spline from connected wires and fitting constructs a spline for each selected wire.

Changing a Spline Back to a Polyline by Unsplining

You can also change a spline back to a polyline. Some CAM (computer-aided manufacturing) systems do not accept splines as input wires, though this is quite rare. To overcome this problem, you can change a spline back to a polyline that approximates the shape of the spline.

Copying the Edges of Existing Surfaces

Boundary edges of existing surfaces can be useful in making other surfaces. You can copy them so that they become wires. (See Figure 2.34.)

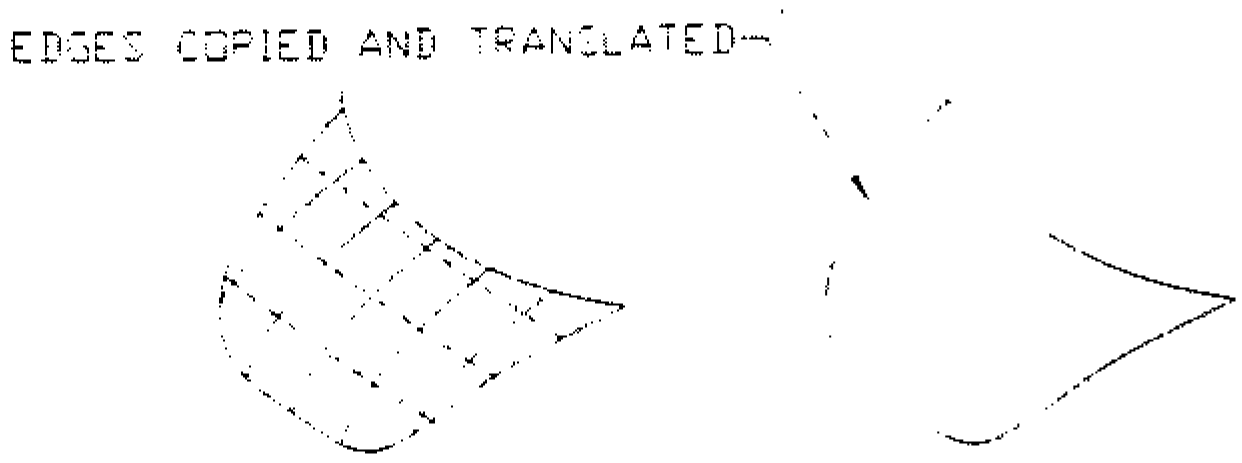


Figure 2.34 Edges copied from a surface

Obtaining Flow Lines from Existing Surfaces

On the computer display, lines along the U and V directions are used to depict the profile and silhouette of a surface. They are called flow lines. You can construct 3D wires from them, and use them as tool paths for manufacturing purposes. Figure 2.35 shows a set of flow lines constructed from an existing surface and translated to the right.

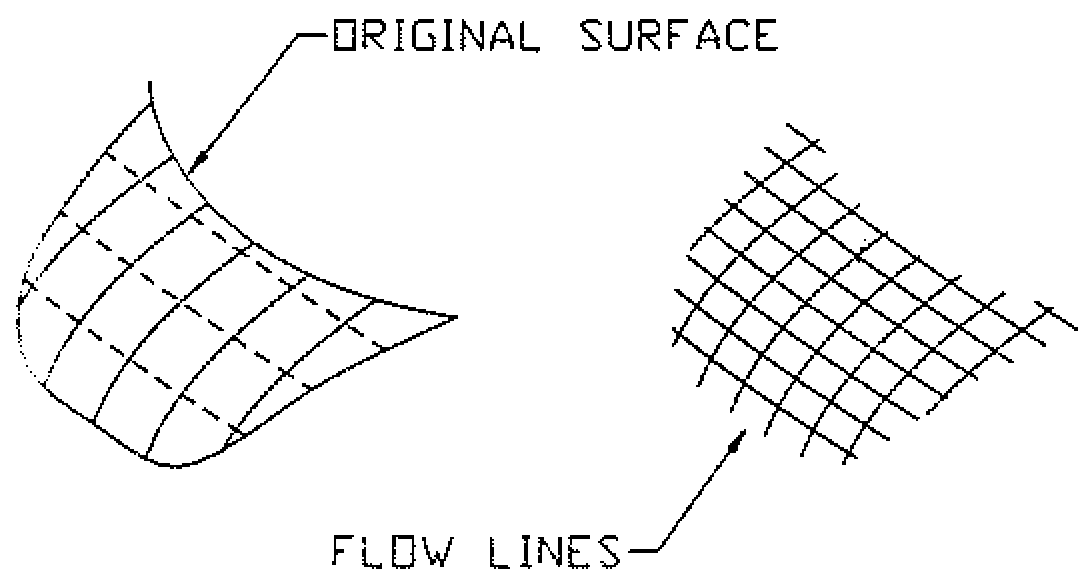


Figure 2.35 Flow lines constructed on NURBS surface and translated

Generating a Parting Line

To make a mold from a surface model, you need a parting line. Figure 2.36 shows a parting line generated on a surface model and translated to the right.

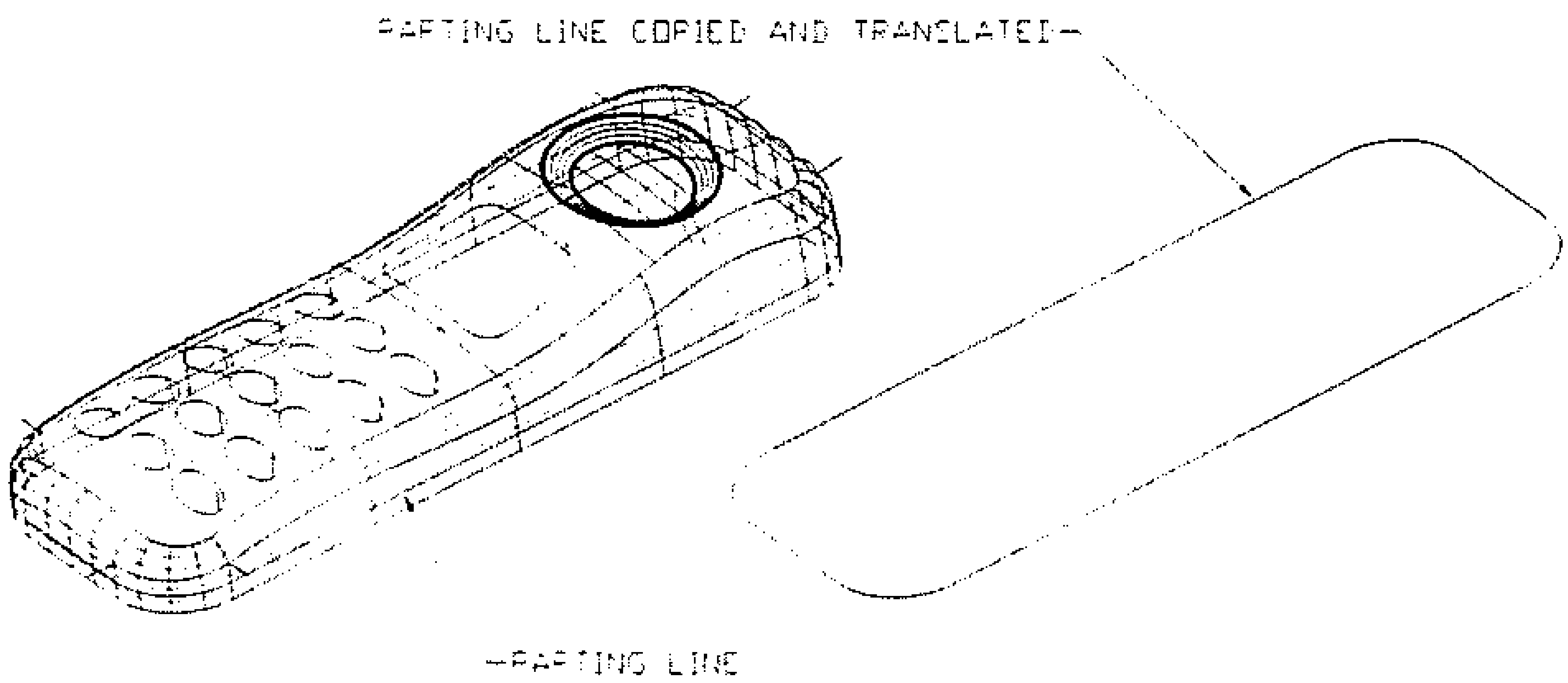


Figure 2.36 Parting line generated on a surface model

Cutting a Series of Section Lines

To visualize and inspect a 3D surface model, you can generate section lines from it. Apart from visualization and inspection, you can use the section lines as tool paths for machining. Figure 2.37 shows a set of section lines generated on a surface model.

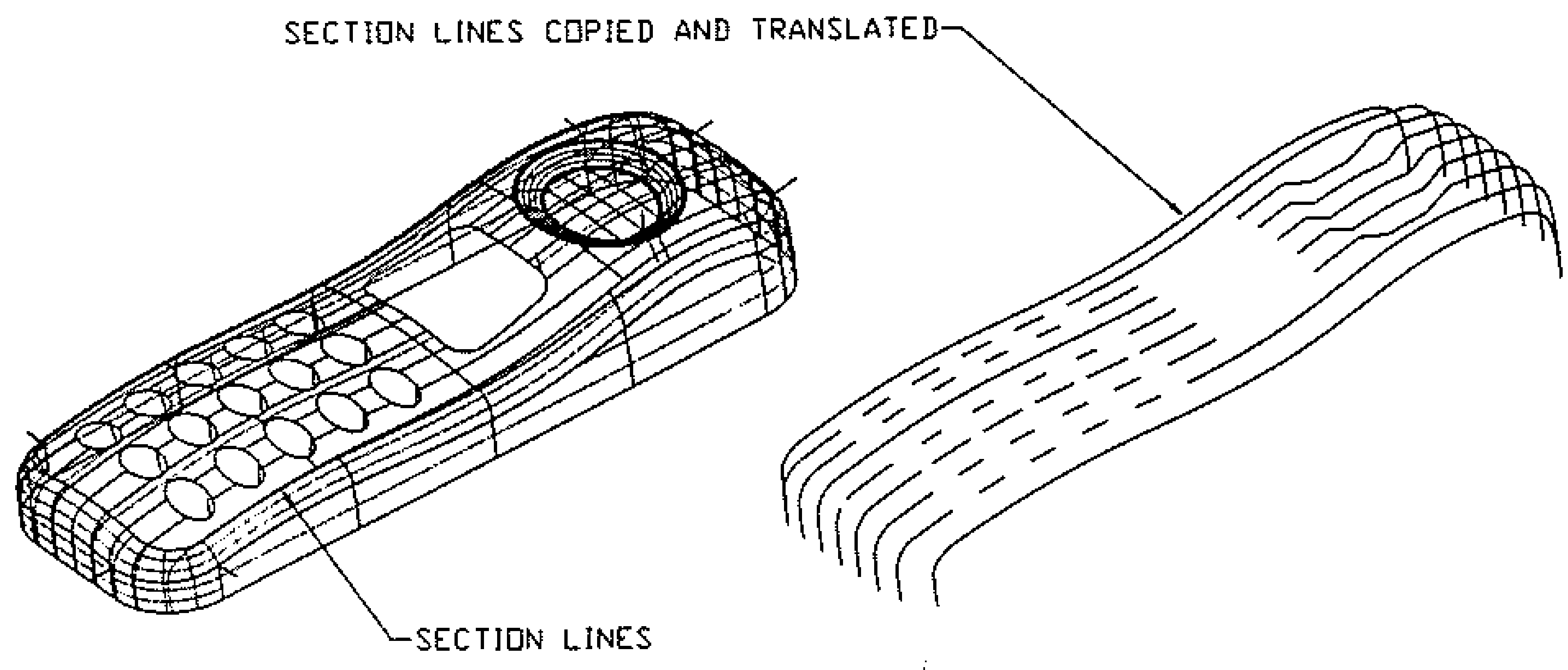


Figure 2.37 Section lines generated on a surface model

Constructing a Wire at the Intersection of Two Surfaces

At the intersection of two intersecting surfaces, an edge is formed. You can obtain a wire from this edge. (See Figure 2.38.)

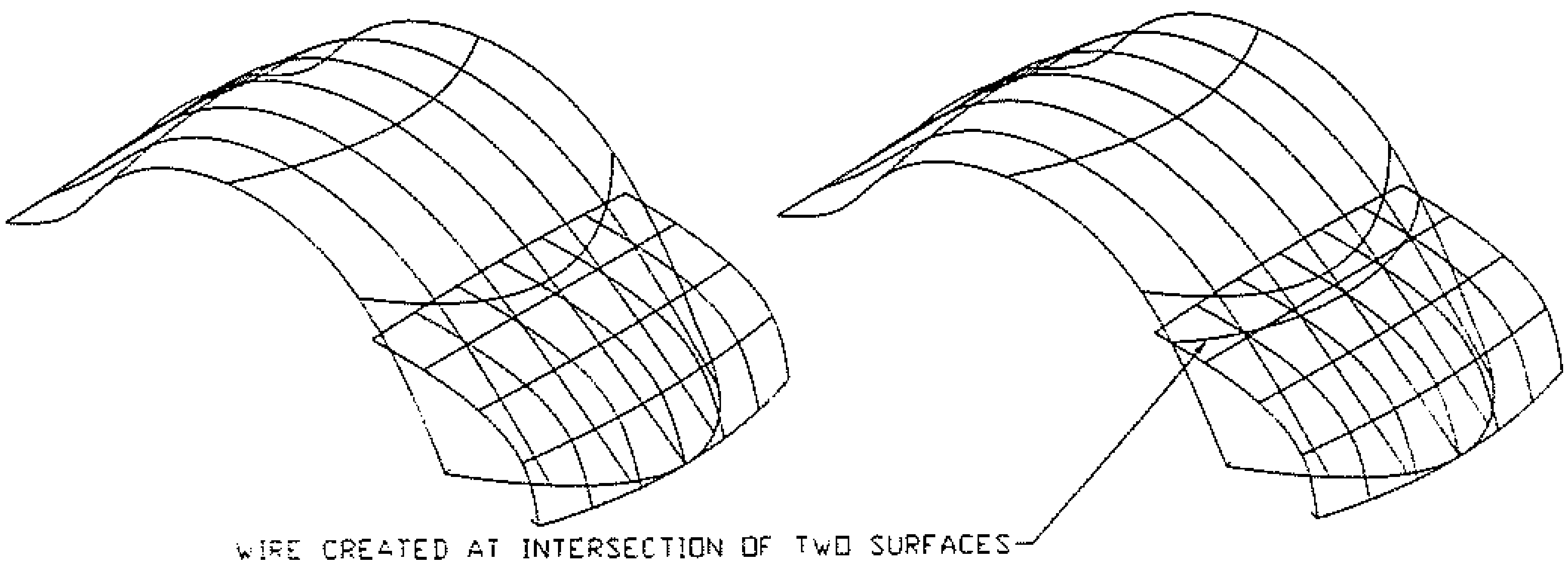


Figure 2.38 Wires constructed at the intersection of two surfaces

Projecting a Wire onto a Surface to Obtain Another Wire

Constructing 3D wires on a curved surface is more difficult and time-consuming than making 2D wires on any specific construction plane (User Coordinate Systems). To make 3D wires on a surface, you can construct 2D wires and then project them onto the surface. (See Figure 2.39.)

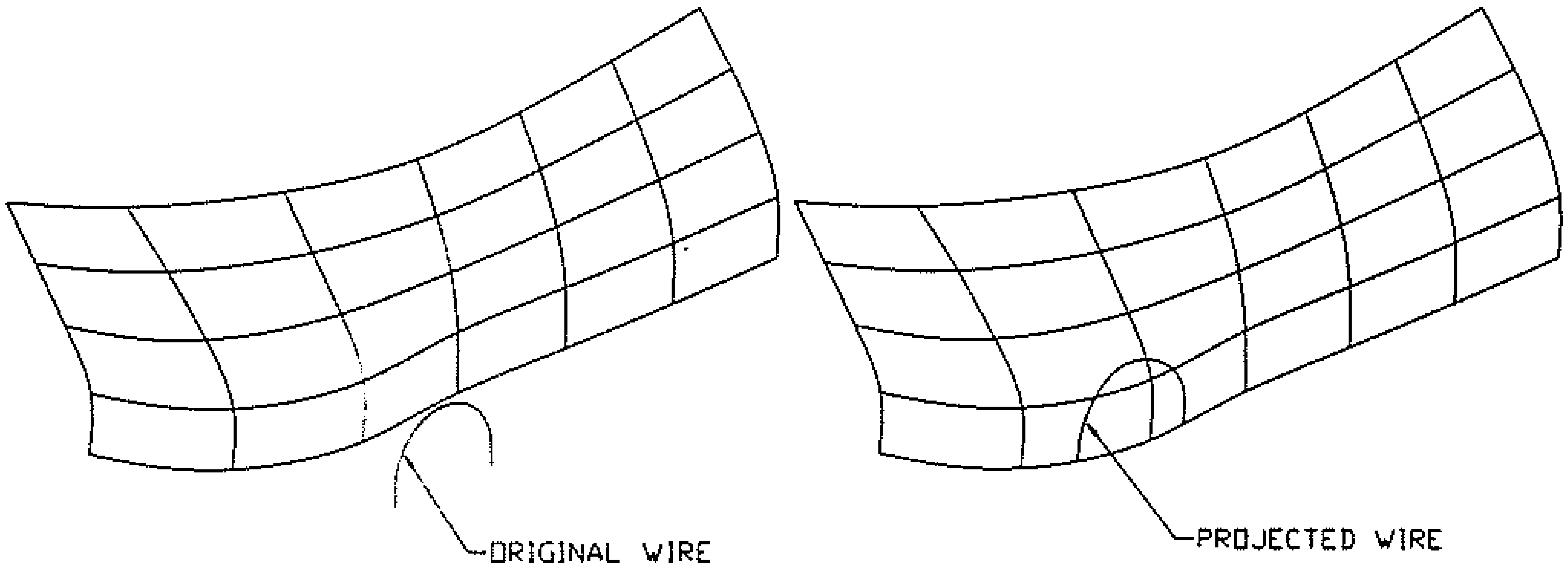


Figure 2.39 Wire projected on a surface

Offsetting 3D Wires

You can offset a 3D wire from a 3D wire. The new wire is constructed at an offset distance that is normal to the selected wire, and the offset is relative to the current display view. (See Figure 2.40.)

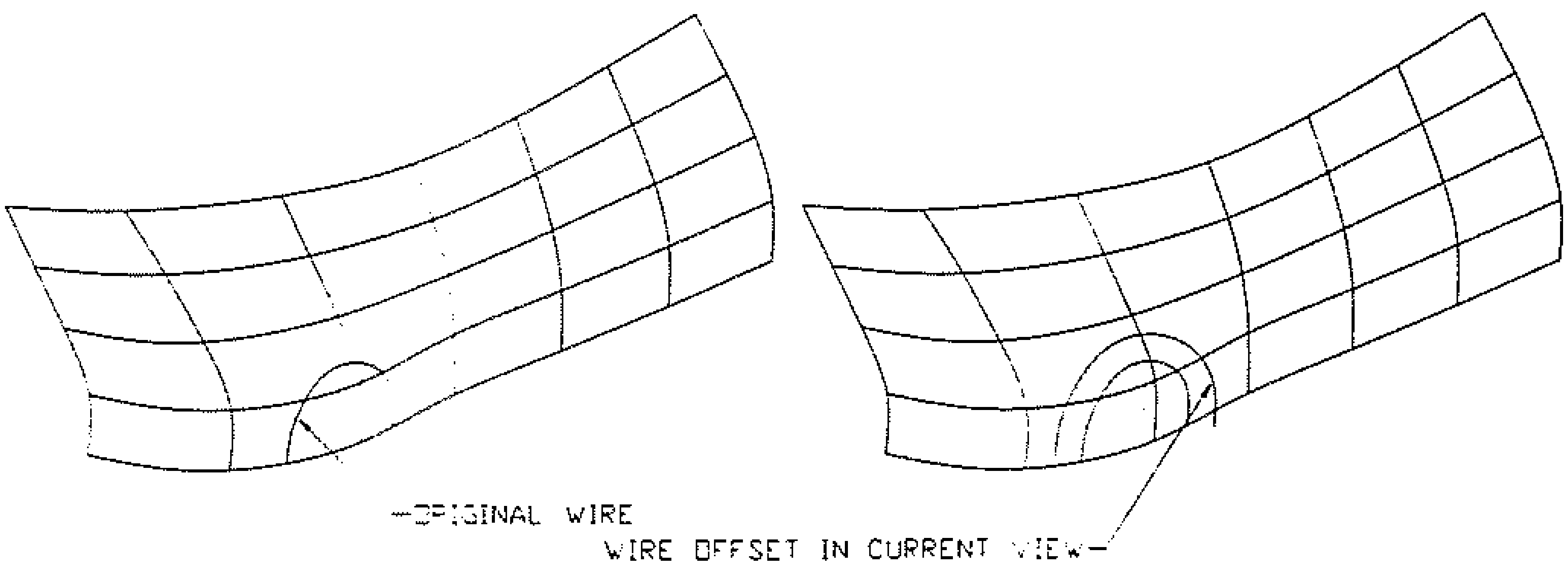


Figure 2.40 Wire offset in the current view

Filleting 3D Wires

Regardless of the UCS location, you can construct a rounded corner between two coplanar wires. While filleting, you can choose to trim or not to trim the selected wires.

Changing the Direction of a Wire

Wires have direction. A wire has a starting point and an ending point. The profiles and silhouettes of a surface constructed from a given set of wires are affected by the directions of the individual wires. To control the surface as it is constructed from a set of wires, you may have to change the directions of the wires. Figure 2.41 shows how the wire directions affect the profile of the surface constructed.

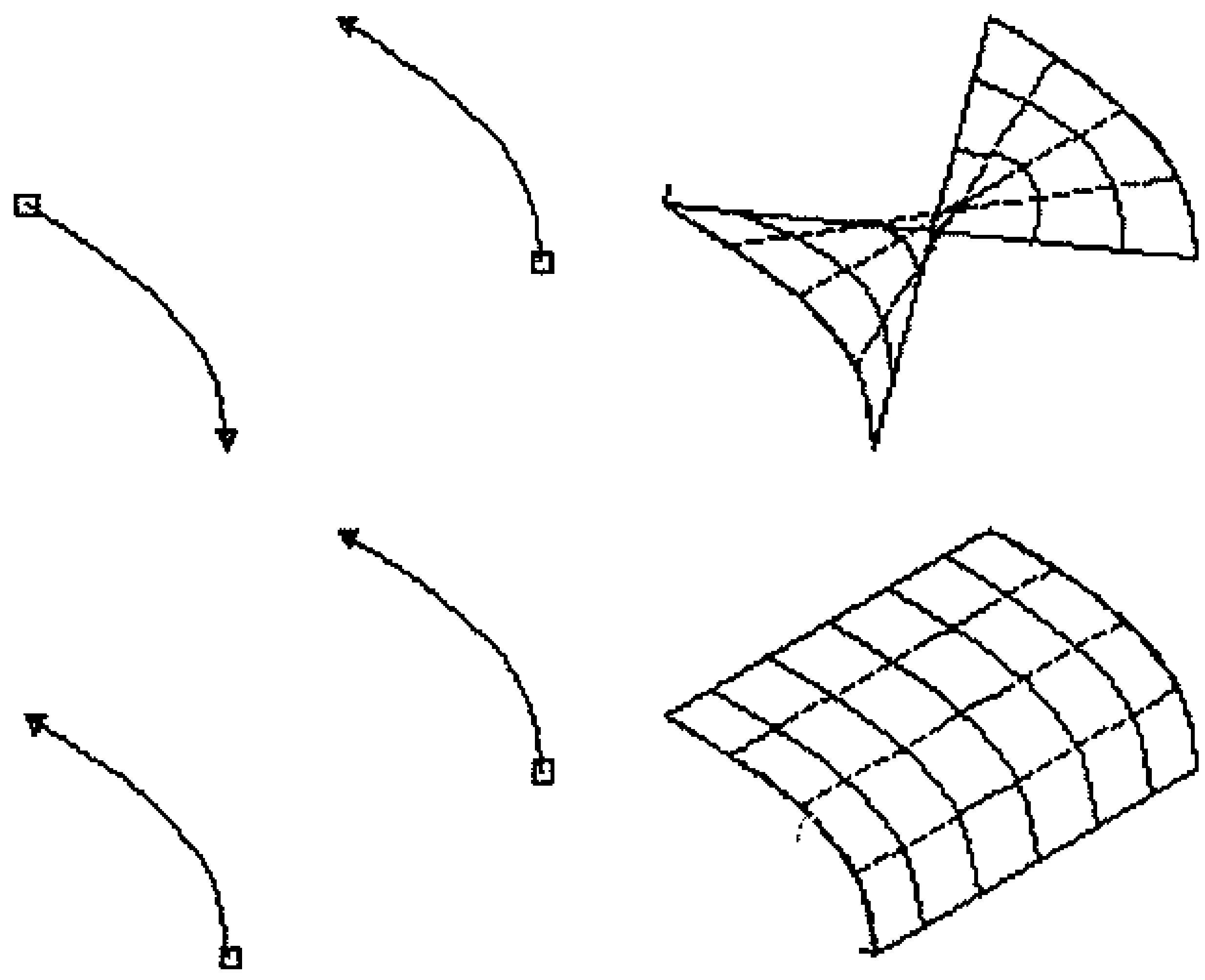


Figure 2.41 Wire directions and the rule surfaces constructed

Refining a Wire

Wires have control points. You can edit a wire by manipulating its control points. To define a more accurate surface, you may have to refine the control points of wires before using them in surface construction.

2.3 Surface Modeling Preferences

By now, you should have a general idea of surface modeling and a general understanding of what you can do by using the surface modeling tools of Mechanical Desktop.

Before you start working on the guided tutorials, spend some time gaining an understanding of the system preferences that affect the outcome of surface modeling. Select the Preferences... item of the Surface pull-down menu to use the AMPREFS command.

Figure 2.42 shows the Surfaces tab of the Desktop Preferences dialog box. It has four major areas. If you are a surface modeling novice, you may find it difficult to set the preferences. To avoid the trouble of setting each of them individually, you can set them collectively by selecting the [Model Size...] button. (See Figure 2.43.) Using this dialog box, you simply select the model size and the unit of measurement. If you have already constructed some objects on your screen, you can select the [Measure Model] button to let the computer measure the model for you.

<Surface> **<Preferences...>**

Command: **AMPREFS**

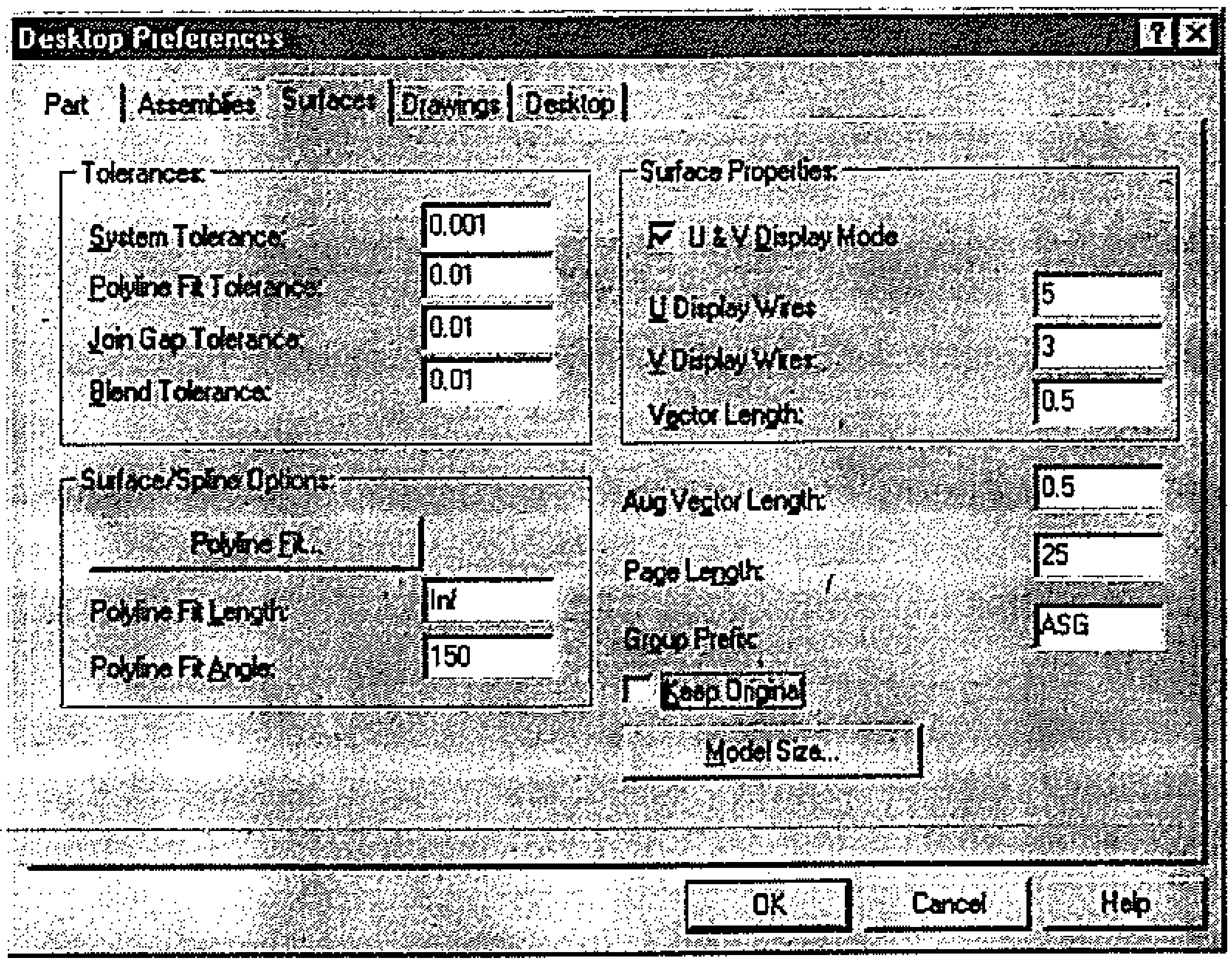


Figure 2.42 Surfaces tab of the Desktop Preferences dialog box

In the Tolerances area, you can set four kinds of tolerances: System Tolerance, Polyline Fit Tolerance, Join Gap Tolerance, and Blend Tolerance.

1. The System Tolerance box controls the system tolerance for constructing flow lines on the surfaces.
2. The Polyline Fit Tolerance box controls the tolerance for fitting a polyline into a spline.
3. The Join Gap Tolerance box controls the tolerance of the gap (if any) between two endpoints.
4. The Blend Tolerance box determines whether to create C0 breaks in the blend joint.

As we have said, Mechanical Desktop employs NURBS mathematics for defining curves and surfaces. Basically, it uses 3D NURBS splines to construct 3D NURBS surfaces. However, it also accepts lines, arcs, 2D polylines, and 3D polylines. While using these objects as wires, Mechanical Desktop fits them into splines according to a set of rules. In the Surface/Spline Options area, you can determine approximately how the spline is fitted. There are two options: Polyline Fit Length and Polyline Fit Angle.

1. Polyline Fit Length sets the polyline length for holding splines and surfaces flat.
2. Polyline Fit Angle sets the polyline angle to break splines and surfaces at corners.

To set these options, you can enter values in the appropriate boxes or select the [Polyline Fit...] button to bring out the Polyline Fit dialog box. (See Figure 2.44.)

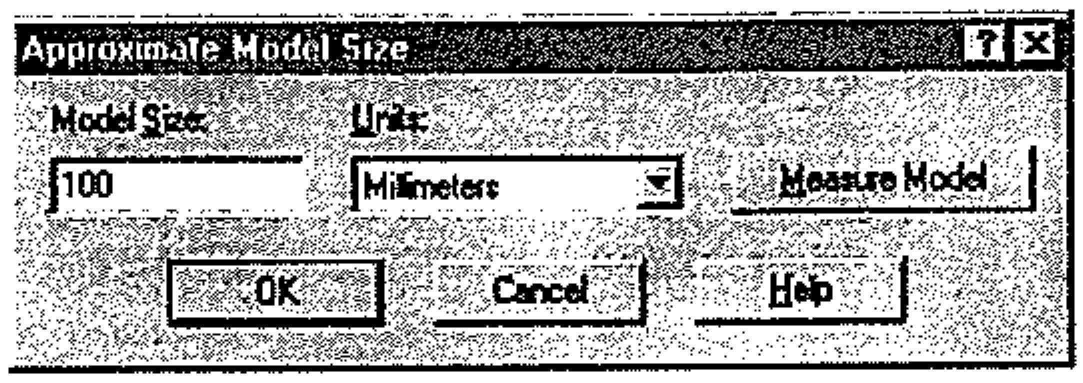


Figure 2.43 Approximate Model Size dialog box

In the Polyline Fit dialog box, you can set the length and angle graphically by selecting the [Length Prompt<] and the [Angle Prompt<] buttons. Now select the [OK] button to return to the Desktop Preferences dialog box. (See Figure 2.42 again.)

At the top right corner (Figure 2.42) is the Surface Properties area. It concerns the display of the surfaces on the screen. On your screen, a surface has three display components: boundary edge, U- and V- wires, and normal vector. The boundary edge is the edge of the surface, the U- and V- wires are wires in two orthogonal directions to depict the profile of the surface, and the normal vector is used to indicate the normal direction of the surface.

- The U & V Display Mode box determines the kind of linetypes to be used to display the U- and V- wires
- The U Display Wires box sets the number of U display wires
- The V Display Wires box sets the number of V display wires
- The Vector Length box sets the length of the vector normal

  Below the Surface Properties area, there are a few boxes.
- The Aug Vector Length box controls the length of the augmented lines. Augmented lines are series of normal vectors along a spline, a line, or a polyline.
- The Page Length box sets the text page length of the text window.
- The Group Prefix box sets the prefix name.
- The Keep Original box determines whether the original object used to make another object is retained.

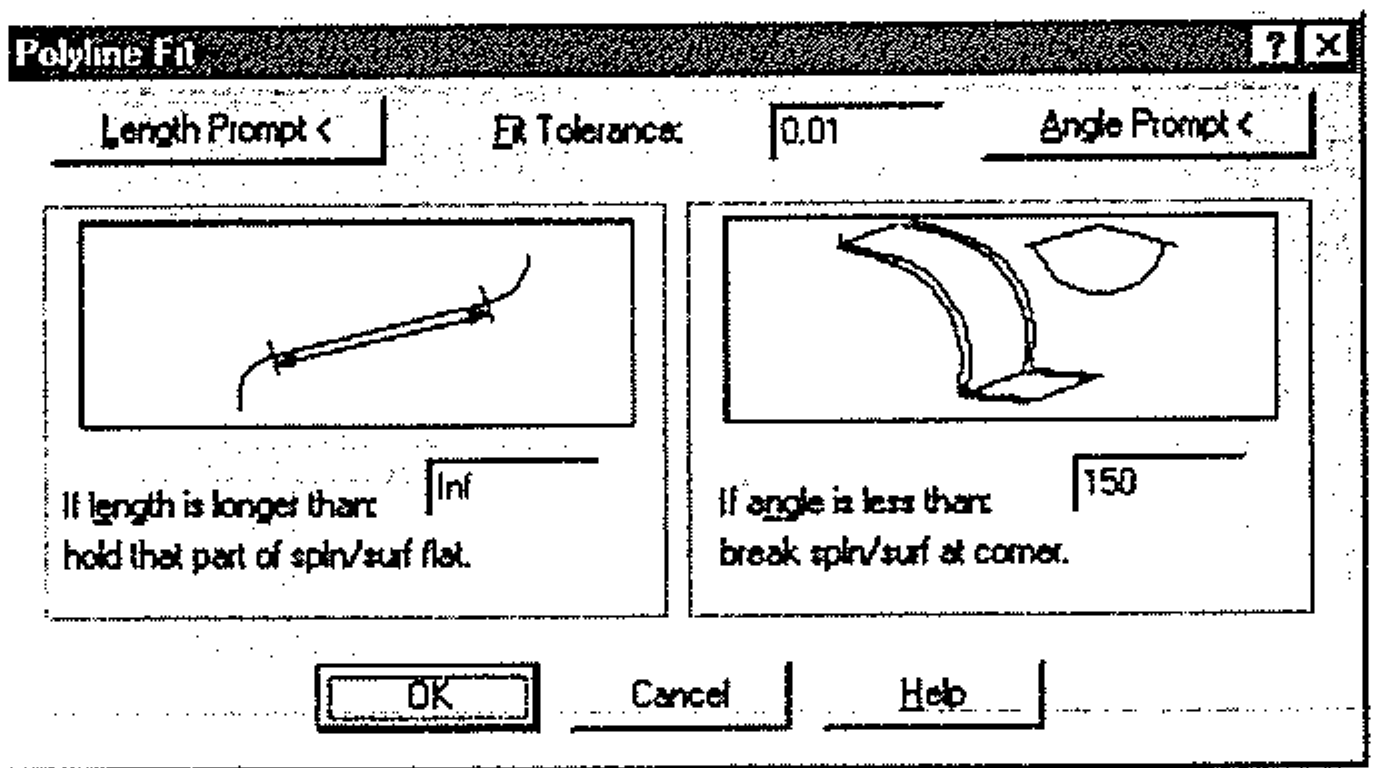


Figure 2.44 Polyline Fit dialog box

When you have finished setting the preferences, select the [OK] button to exit. When you know how to set the preferences for surface model construction, you can work on the projects that follow.

2.4 Infant Toy Project

Figure 2.45 shows the rendered image of an infant toy. It is a 3D free-form object. You will first analyze the model to determine what surfaces are required and what wires are needed to generate the surfaces. Then you will construct the wires and make the surfaces.

Figure 2.45 Rendered image of the infant toy

Analysis

This model is symmetrical about its center. It has two surfaces. Each of them is a loft uv surface. To make the loft uv surface, you will construct a set of U-wires and a set of V-wires. Using the U- and V- wires, you will then construct a loft uv surface. Figure 2.46 shows the surfaces moved apart.

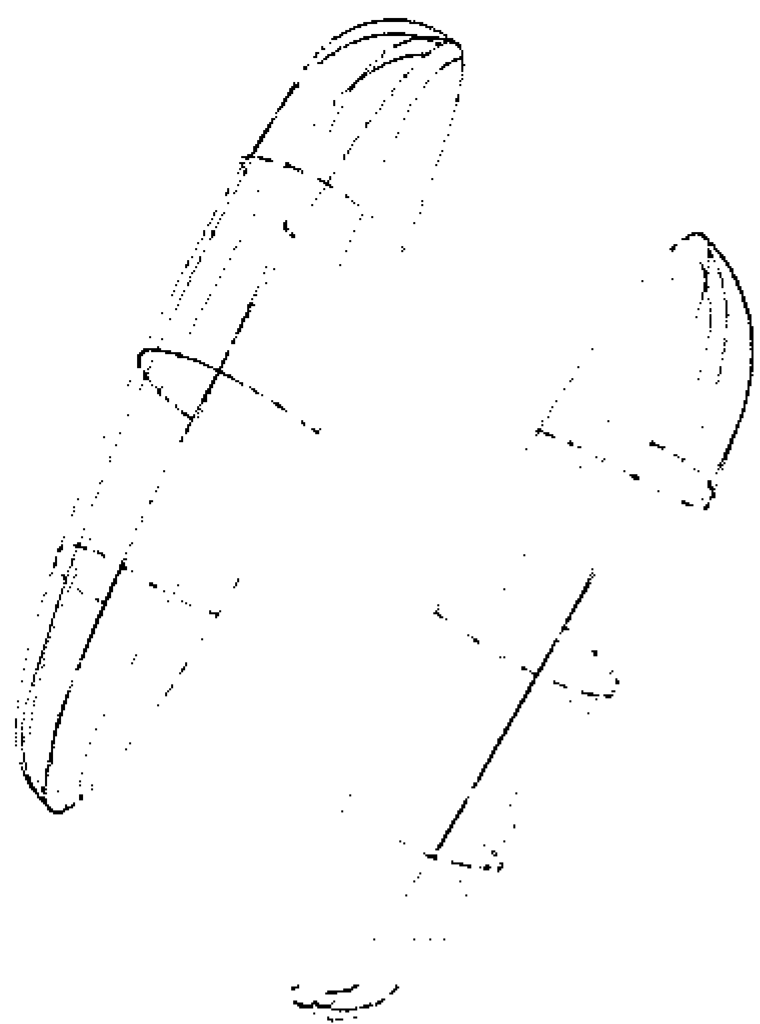


Figure 2.46 Loft uv surfaces moved apart

Drawing Setup

Use the NEW command to start a new drawing from scratch with metric default settings.

<File> **<New...>**

Command: **NEW**

[Start from Scratch
Metric **OK**]

Because it will take you considerable time to construct wires, you should develop the habit of setting up a layer and keeping the wires there. After completing the surface model, you can turn off that layer. If you want to make any change later, you always have some wires to base it on.

Select the Layer... item of the Format cascading menu of the Assist pull-down menu to use the LAYER command to create two additional layers, Surface and Wire, with the colors green and blue, respectively. Then set the current layer to Wire. You will find a number of layers prefixed by the letters AM. Mechanical Desktop uses these layers for specific reasons. Do not make any changes to these layers or manipulate the entities residing on them.

<Assist> **<Format>** **<Layer...>**

Command: **LAYER**

[Layer

Name	Color	Linetype
0	White	Continuous
Surface	Red	Continuous
Wire	Green	Continuous

Current Layer: **Wire**

OK]

For the sake of clarity, make sure the UCS icon is turned on. To keep track of the location of the XY plane of the UCS, set the UCS icon to display at the origin.

<Assist> <v**UCS Icon** >

<Assist> <v **Icon at Origin**>

Because you may need to use these settings in a number of surface modeling projects in this chapter, you should now save the current drawing to a template for other projects. Select the Save As... item of the File pull-down menu to use the SAVEAS command. To have a drawing file for the current project, save the drawing again.

<File> **<Save As...>**

Command: **SAVEAS**

[Save Drawing As
File name: **Surf_mdl.dwt**
Save as type: **Drawing Template File (*.dwt)**
Save]

<File> **<Save As...>**

Command: **SAVEAS**

[Save Drawing As
File name: **Infant.dwg**
Save as type: **AutoCAD R14 Drawing (*.dwg)**
Save]

Now you have a template and a drawing. The template is called Surf_mdl.dwt, and the current working drawing is called Infant.dwg. Select the Preferences... item of the Surface pull-down menu to run the AMPREFS command.

<Surface> **<Preferences...>**

Command: **AMPREFS**

Select [Model Size...] on the Surfaces tab of the Desktop Preferences dialog box. Then set the model size to 200 mm. Select the [OK] button to exit.

Wire Construction

To make this model, you will construct a loft uv surface and then make a mirror copy of it. Figure 2.47 shows the wires required.

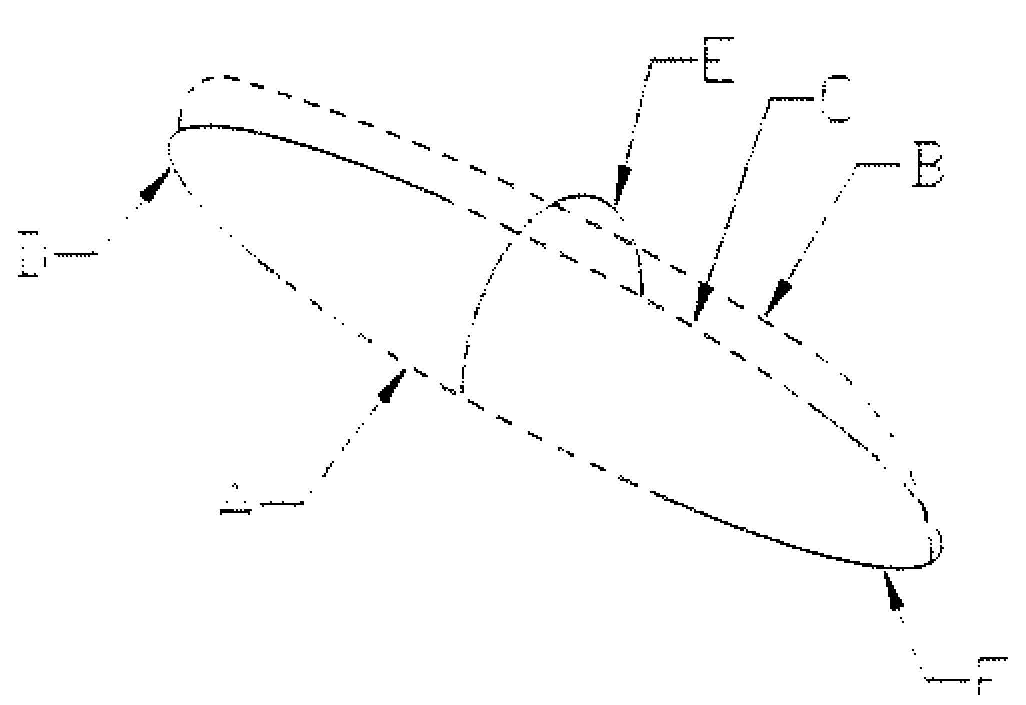


Figure 2.47 U- and V-wires illustrated in different linetypes

In Figure 2.47, wires A, B, and C are U-wires, and wires D, E, and F are V-wires. They are depicted in different linetypes to show you which are the U-wires and which are the V-wires. When you construct these wires, you do not have to change their linetypes.

To start with, use the ELLIPSE command to construct an ellipse, and use the XLINE command to construct two vertical lines. Then use the shortcut key [F] to fit the display to the screen. (See Figure 2.48.)

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **0,0**
Axis endpoint: **@60<0**
<Other axis distance>/Rotation: **14**

Command: **F**

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **V**
Through point: **-30,0**
Through point: **30,0**
Through point: **[Enter]**

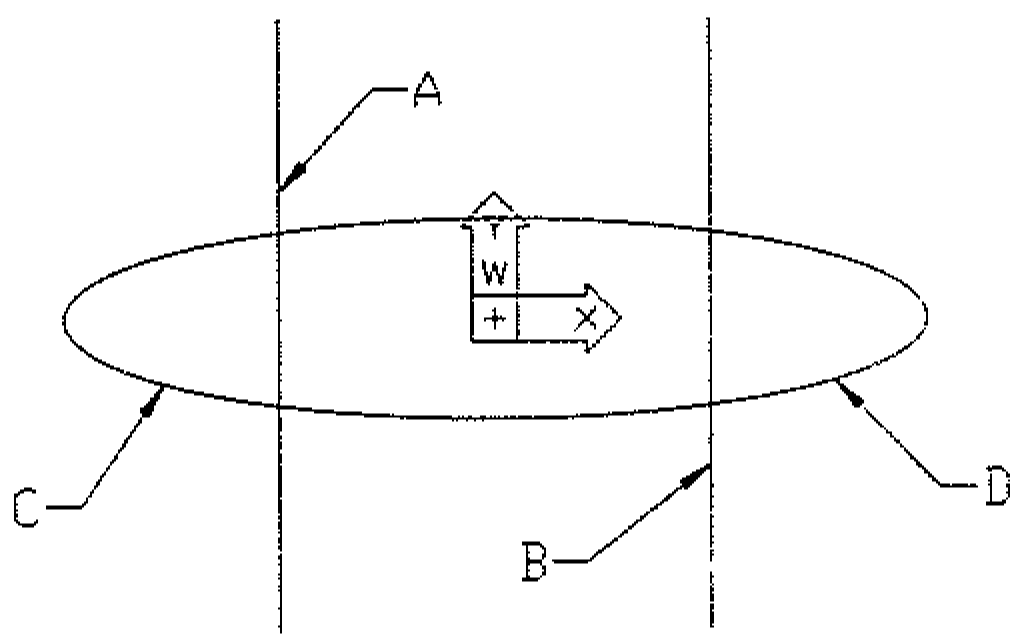


Figure 2.48 Ellipse and vertical lines constructed

Now use the TRIM command to trim the ellipse. (See Figure 2.49.)

<Modify> **<Trim>**

Command: **TRIM**
Select objects: [**Select A and B (Figure 2.48).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select C and D (Figure 2.48).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

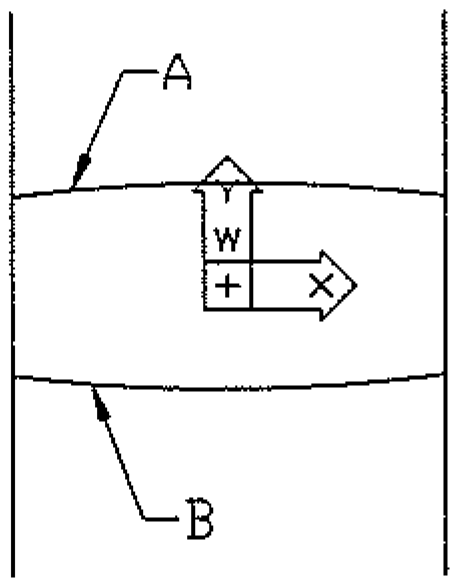


Figure 2.49 Ellipse trimmed

Next, construct another ellipse at a location that overlaps the two elliptical wires. To avoid confusion, hide the elliptical arcs. Select the Visibilities... item of the Surface pull-down menu to use the AMVISIBLE command. (See Figure 2.50, the Desktop Visibility dialog box.) Select the [Select<] button and then select the two elliptical arcs that you want to hide. After that, select the [OK] button to exit the command.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

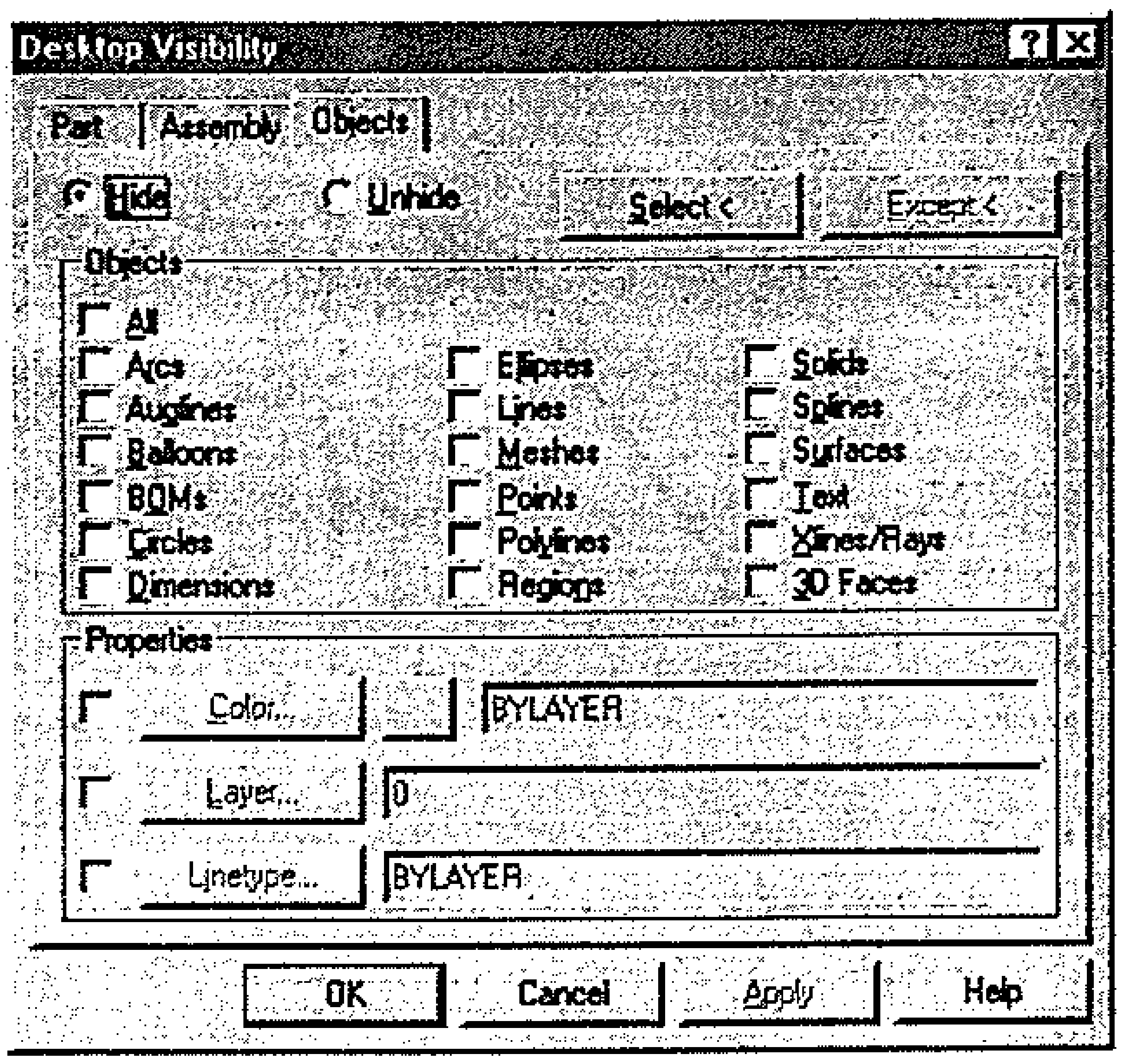


Figure 2.50 Desktop Visibility dialog box

Select objects to hide: [**Select A and B (Figure 2.49).**]
Select objects to hide: [**Enter**]

Now the elliptical arcs are hidden. Construct another ellipse. (See Figure 2.51.)

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **0,0**
Axis endpoint: **@60<0**
<Other axis distance>/Rotation: **14**

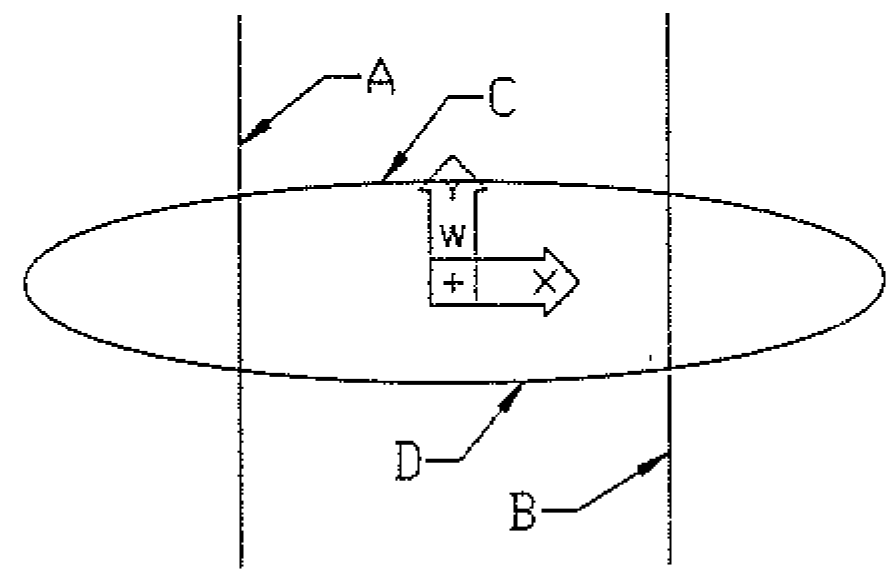


Figure 2.51 Elliptical arcs hidden, second ellipse constructed

To make two more elliptical arcs, use the TRIM command again. After trimming, the vertical construction lines are not required. Use the ERASE command to delete them.

<Modify> **<Trim>**

Command: **TRIM**
Select objects: [**Select A and B (Figure 2.51).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select C and D (Figure 2.51).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

<Modify> **<Erase>**

Command: **ERASE**
Select objects: [**Select A and B (Figure 2.51).**]
Select objects: [**Enter**]

Now use the shortcut key [8] to set the display to an isometric view and use the UCS command to rotate the UCS 90° about the X axis. (See Figure 2.52.)

Command: **8**

<Assist> **<UCS>** **<X Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

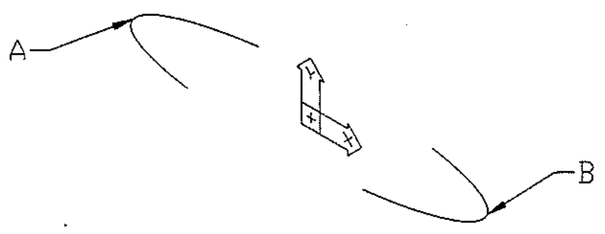


Figure 2.52 Ellipse trimmed, construction lines deleted, and UCS rotated

Construct two circles on the new UCS plane. Then set the display to the plan view of the current UCS by using the shortcut key [9]. (See Figure 2.53.)

<Design> <Circle> <2 Points>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **2P**
First point on diameter: **QUA** of **[Select A (Figure 2.52).]**
Second point on diameter: **@25<0**

<Design> <Circle> <2 Points>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **2P**
First point on diameter: **QUA** of **[Select B (Figure 2.52).]**
Second point on diameter: **@40<180**

Command: **9**

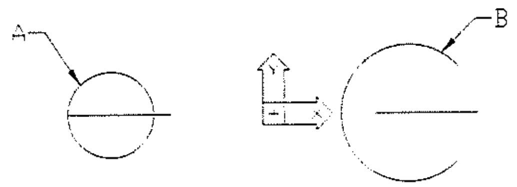


Figure 2.53 Two circles constructed, display set

As shown in Figure 2.54, construct a circle that is tangent to the last two circles and construct a horizontal line.

<Design> <Circle> <Tan, Tan, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **TTR**
Enter Tangent spec: **[Select A (Figure 2.53).]**
Enter second Tangent spec: **[Select B (Figure 2.53).]**
Radius: **300**

<Design> **<Line>**

Command: **LINE**
From point: **60,0**
To point: **-60,0**
To point: **[Enter]**

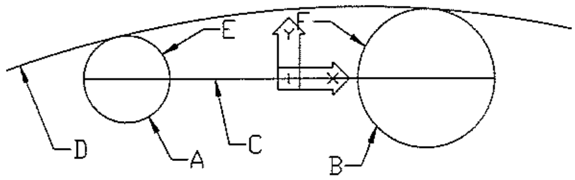


Figure 2.54 Tangent circle and horizontal line constructed

Now use the TRIM command to trim the circles. After trimming, erase the unwanted horizontal line. Then set the display to an isometric view. (See Figure 2.55.)

<Modify> **<Trim>**

Command: **TRIM**
Select objects: **[Select A, B, C, and D (Figure 2.54).]**
Select objects: **[Enter]**
<Select object to trim>/Project/Edge/Undo: **[Select A, B, D, E, and F (Figure 2.54).]**
<Select object to trim>/Project/Edge/Undo: **[Enter]**

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select C (Figure 2.54).]**
Select objects:

Command: **8**

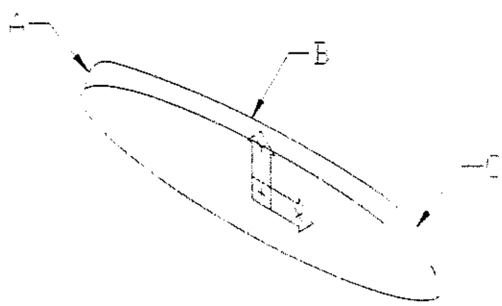


Figure 2.55 Circles trimmed, line erased, and display set

Arcs A, B, and C (Figure 2.55) are used to form a spline. Select the Join.... item of the Edit Wireframe cascading menu of the Surface pull-down menu to use the AMJOIN3D command to join them together into a spline. (See Figure 2.56, the JOIN3D dialog box.)

There are two joining modes, Manual and Automatic. Manual mode joins wires regardless of how far apart they are. Using Automatic mode, the maximum distance between the wires is determined by the Gap Tolerance, which you can set by using the AMPREFS command. Use Automatic mode. There are three kinds of outputs: polyline, spline, and augmented line. Select the spline option. Then select the [OK] button. After that, select the wires to be joined. Now you should see a small triangle at one end of the first selected wire and a small square box at the last selected wire. The triangle depicts the direction of the spline and the box indicates the starting point. Accept the default.

<Surface> **<Edit Wireframe>** **<Join...>**

Command: **AMJOIN3D**

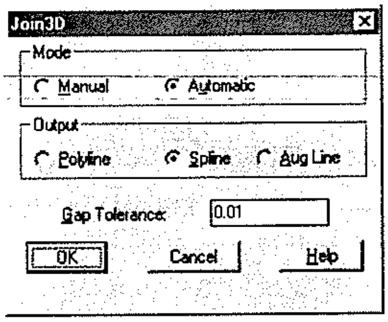


Figure 2.56 JOIN3D dialog box

Select start wire: [**Select A (Figure 2.55).**]
Select wires to join: [**Select B and C (Figure 2.55).**]
Select wires to join: [**Enter**]
Reverse? Yes/<No>: [**Enter**]

After joining, there should be no noticeable change on your screen. However, the circular arcs are removed and a spline is approximated to the three selected arcs. Now use the AMVISIBLE command to unhide the hidden elliptical arcs. Then construct a line. After that, rotate the UCS 90° about the Y axis. (See Figure 2.57.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[**Unhide** **All** **OK**]

<Design> **<Line>**

Command: **LINE**
From point: **0,0**
To point: **@30<90**
To point: [**Enter**]

<Assist> **<UCS>** **<Y Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **Y**
Rotation angle about Y axis: **90**

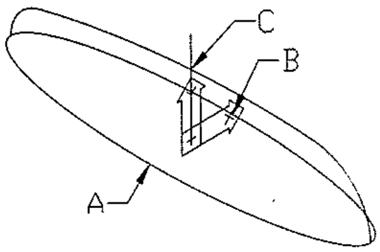


Figure 2.57 Elliptical arcs unhidden, line constructed, and UCS rotated

On the new UCS plane, construct an ellipse and a line. (See Figure 2.58.)

<Design> **<Ellipse>** **<Axis, End>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **QUA** of [**Select A (Figure 2.57).**]
Axis endpoint 2: **QUA** of [**Select B (Figure 2.57).**]
<Other axis distance>/Rotation: **INT** of [**Select C (Figure 2.57).**]

<Design> **<Line>**

Command: **LINE**
From point: >: **QUA** of [**Select A (Figure 2.57).**]
To point: **QUA** of [**Select B (Figure 2.57).**]
To point: [**Enter**]

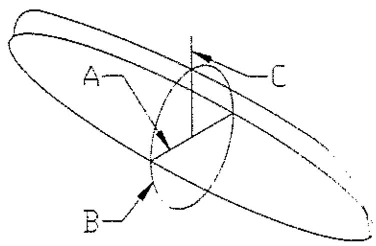


Figure 2.58 Ellipse and line constructed

Use line A to trim ellipse B (Figure 2.58). After trimming, lines A and C (Figure 2.58) are not required. Use the ERASE command to delete them. (See Figure 2.59.)

<Modify> **<Trim>**

Command: **TRIM**
Select objects: [**Select A (Figure 2.58).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select B (Figure 2.58).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

<Modify> **<Erase>**

Command: **ERASE**
Select objects: [**Select A and C (Figure 2.58).**]
Select objects: [**Enter**]

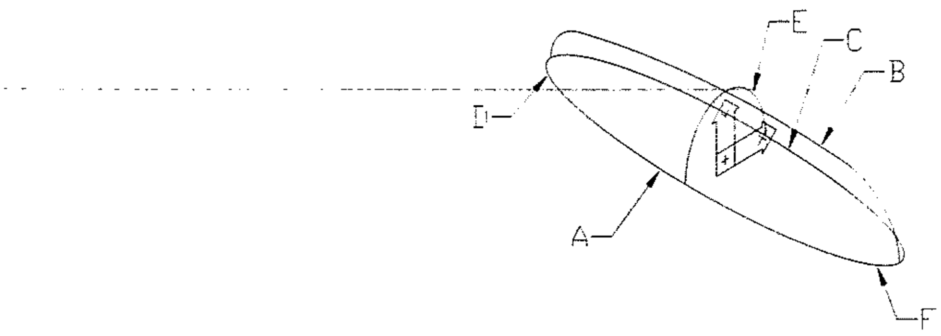


Figure 2.59 Ellipse trimmed and lines deleted

From the Mechanical Main toolbar, select the Surface layer of the Layer Control item to set the current layer to Surface. Then select the LoftUV item of the Create Surface cascading menu of the Surface pull-down menu to run the AMLOFTUV command. (See Figure 2.60.)

[**Mechanical Main**] [**Layer Control**]

Current Layer: **Surface**

<Surface> **<Create Surface>** **<LoftUV>**

Command: **AMLOFTUV**
Select U wires: [**Select A, B, and C (Figure 2.59).**]
Select U wires: [**Enter**]
Select V wires: [**Select D, E, and F (Figure 2.59).**]
Select V wires: [**Enter**]

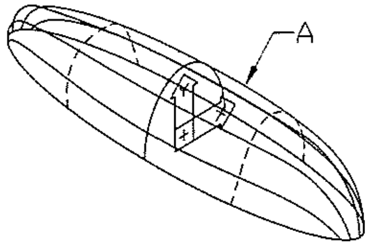


Figure 2.60 Loft uv surface constructed

The surface for one half of the model of the infant toy is complete. Because the wires are not needed, turn off layer Wire. To construct the other half of the surface model, use the MIRROR3D command to make a mirror copy of the loft uv surface. (See Figure 2.61.)

[Mechanical Main] **[Layer Control]**

Off Layer: **Wire**
Current Layer: **Surface**

<Construct> **<3D Translations>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select A (Figure 2.60).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **ZX**
Point on ZX plane <0,0,0>: **[Enter]**
Delete old objects? <N> **[Enter]**

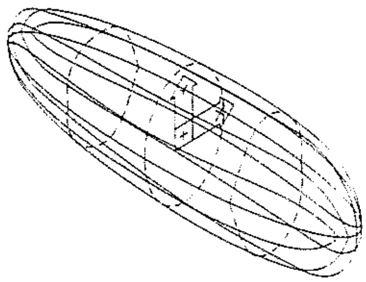


Figure 2.61 Loft uv surface mirrored

Now the surface model is complete. Save your drawing.

<File> **<Save>**

In making this model, you constructed wires before making the surfaces. The process of surface modeling starts from making the wires. However, the first things that come to your mind when you want to make a surface model are the shape, profiles, and silhouettes of the surfaces, but not the wires. Because you need to construct the wires first, you have to think about how they look. Before you do that, you have to identify what kinds of free-form surfaces are to be constructed. Once the kinds of surfaces are identified, you can make the wires. From the wires, you can construct the surfaces.

In this and other projects in this chapter, you can see that the most tedious job in surface modeling is the making of the wires and that the making of the surfaces from the wires is simple. You need only use the appropriate surface construction commands.

2.5 Joy Pad Project

Figure 2.62 shows the rendered image of the surface model of a joy pad. In the last project, you had to construct only one free-form surface, but in this one, you have to construct a number of intersecting surfaces and treat the edges.

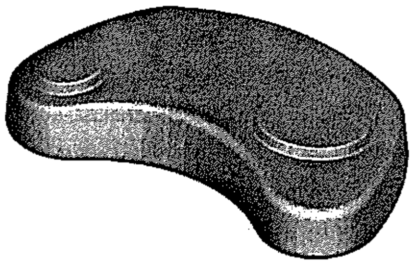


Figure 2.62 Rendered image of the joy pad

Analysis

Examine Figure 2.63 carefully. Take some time to analyze the model to find out what surfaces are required, how the edges are formed, and what wires are needed.

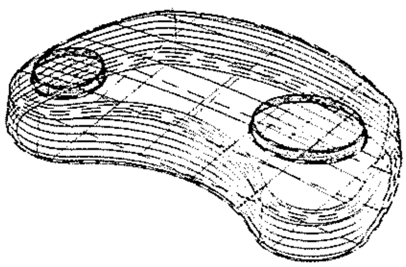


Figure 2.63 Surface model of the joy pad

You can see that the edges of the model are rounded. Naturally, they are fillet surfaces. To illustrate how the model looks without the fillets, the rounded edges of the model are removed. (See Figure 2.64.)

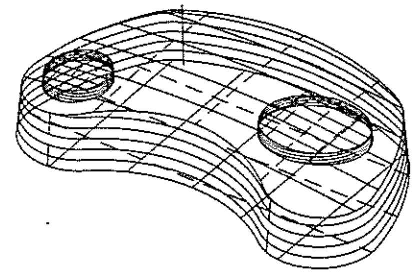


Figure 2.64 Fillet edges removed

You can see in Figure 2.64 that there are three major features: the main body and two raised buttons.

The main body has three main surfaces: bottom, side, and top.

- The bottom surface of the model is a trimmed planar surface.
- The side wall of the model is an extrude surface.
- The top face is a loft u surface.

For each raised button, there are two surfaces.

- The side of the button is an extrude surface.
- The top of the button is a surface offset from the top surface of the main body.

To make the model, you will construct surfaces that are larger than required from smooth defining wires. Then you will trim the surfaces. Figure 2.65 shows the base surfaces before trimming.

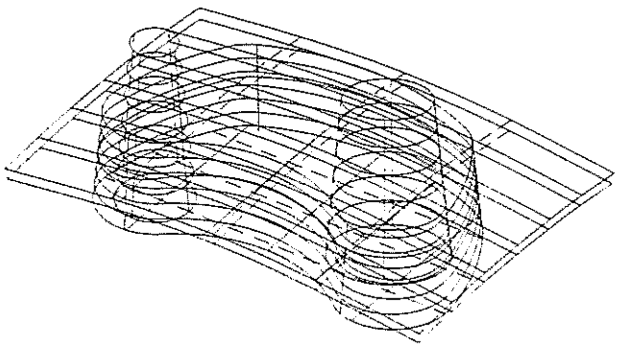


Figure 2.65 Base surfaces of the surface model

Knowing what base surfaces are required, you can think about the wires. For the bottom and side surfaces of the main body, you need a wire that resembles the top view of the model to construct a trimmed planar surface and an extrude surface. For the top surface of the main body, you need three wires to construct a loft u surface. For the side surfaces of the buttons, you need a circle and an ellipse to construct two extrude surfaces. For the top surfaces of the buttons, you offset the top surface of the main body. Figure 2.66 shows the wires needed.

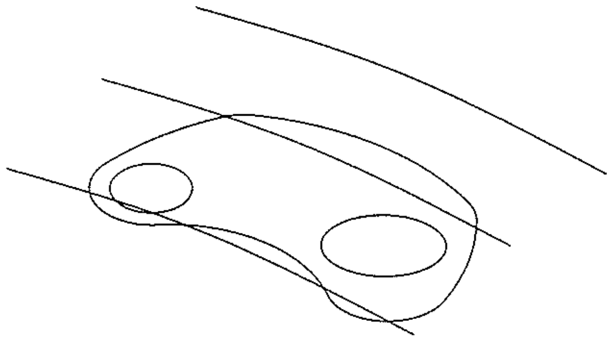


Figure 2.66 Wires needed to construct the surfaces

Drawing Setup

Start a new drawing and use the template drawing that you saved earlier in this chapter. If you saved a different template file name, browse to locate the file.

<File> **<New...>**

Command: **NEW**

[Use a Template
Select a Template: **Surf_mdl.dwt**
OK]

Before you make the wires and surfaces, set the related system variables by using the AMPREFS command.

<Surface> **<Preferences...>**

Command: **AMPREFS**

Select [Model Size...] on the Surfaces tab of the Desktop Preferences dialog box. Then set the model size to 150 mm and select the [OK] button. After that, set the Vector Length to 5 units. Finally, select the [OK] button to exit.

Bottom and Side Surfaces

As we have explained, the bottom and side surfaces of the joy pad are constructed from a wire that resembles the top view of the model. To make this wire, you will construct a wire from a series of trimmed circles. Note that you should work on the layer Wire. To begin, use the CIRCLE command and the MIRROR command to construct four circles as shown in Figure 2.67. The (120,0) and (120,1) points below define a vertical mirror line.

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

<Design> <Circle> <Center, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **50,50**
Diameter/<Radius>: **30**

<Design> <Circle> <Center, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **70,100**
Diameter/<Radius>: **30**

<Construct> <Mirror>

Command: **MIRROR**
Select objects: **ALL**
Select objects: [**Enter**]
First point of mirror line: **120,0**
Second point: **120,1**
Delete old objects? **N**

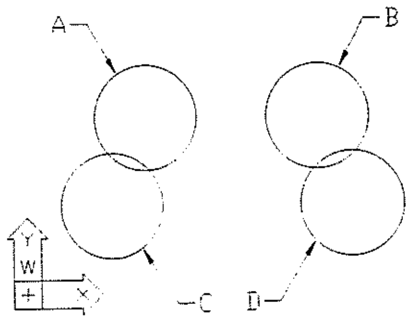


Figure 2.67 Circles constructed

Now construct two tangent circles. (See Figure 2.68.)

<Design> <Circle> <Tan, Tan, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **TTR**
Enter Tangent spec: [**Select A (Figure 2.67).**]
Enter second Tangent spec: [**Select B (Figure 2.67).**]
Radius: **150**

<Design> <Circle> <Tan, Tan, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **TTR**
Enter Tangent spec: [**Select C (Figure 2.67).**]
Enter second Tangent spec: [**Select D (Figure 2.67).**]
Radius: **80**

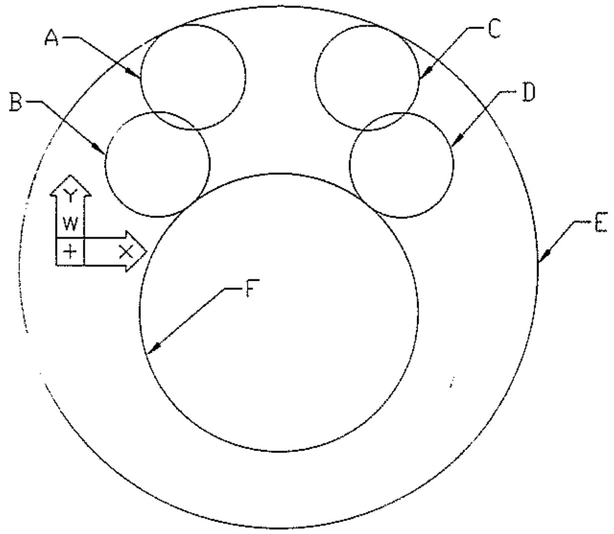


Figure 2.68 Two tangent circles drawn

As in Figure 2.69, use the TRIM command to trim the tangent circles.

<Modify> **<Trim>**

Command: **TRIM**
Select objects: [**Select A, B, C, and D (Figure 2.68).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select E and F (Figure 2.68).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

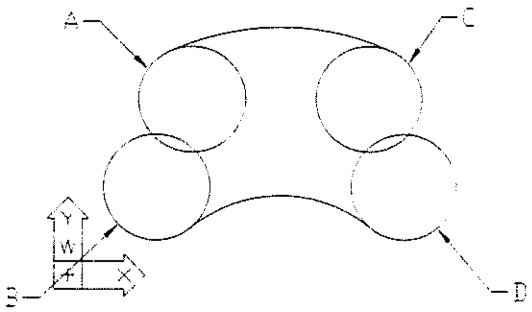


Figure 2.69 Tangent circles trimmed

Now construct two more tangent circles. (See Figure 2.70.)

<Design> **<Circle>** **<Tan, Tan, Radius>**

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **TTR**
Enter Tangent spec: [**Select A (Figure 2.69).**]
Enter second Tangent spec: [**Select B (Figure 2.69).**]

Radius: **150**

<Design> <Circle> <Tan, Tan, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **TTR**
Enter Tangent spec: [**Select C (Figure 2.69).**]
Enter second Tangent spec: [**Select D (Figure 2.69).**]
Radius: **150**

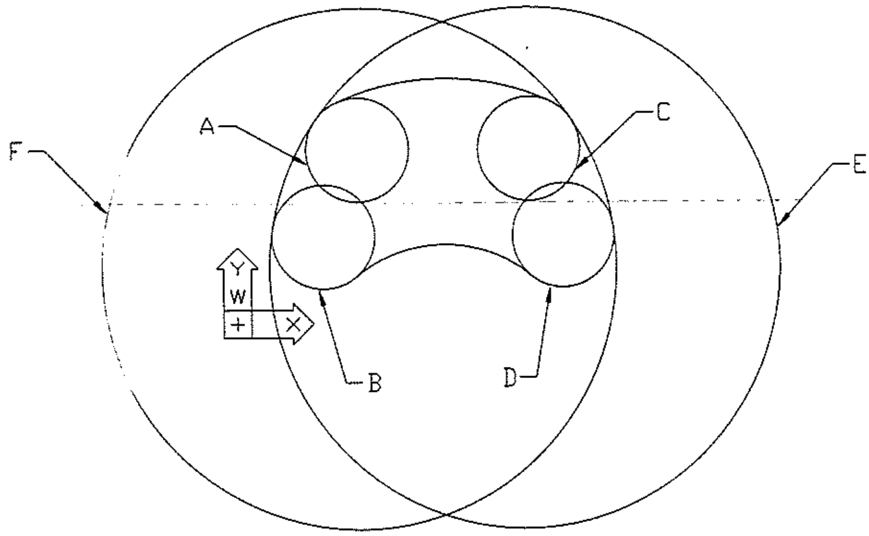


Figure 2.70 Two more tangent circles constructed

Trim the two tangent circles as in Figure 2.71.

<Modify> <Trim>

Command: **TRIM**
Select objects: [**Select A, B, C, and D (Figure 2.70).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select E and F (Figure 2.70).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

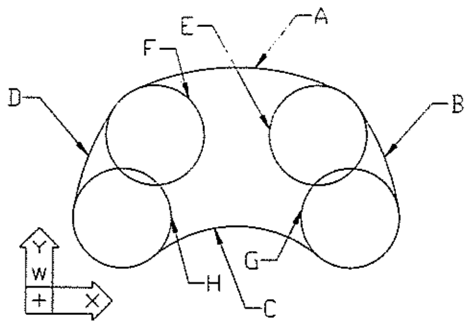


Figure 2.71 Tangent circles trimmed

Repeat the TRIM command to trim the four circles. (See Figure 2.72.)

<Modify> **<Trim>**

Command: **TRIM**
Select objects: [**Select A, B, C, and D (Figure 2.71).**]
Select objects: [**Enter**]
<Select object to trim>/Project/Edge/Undo: [**Select E, F, G, and H (Figure 2.71).**]
<Select object to trim>/Project/Edge/Undo: [**Enter**]

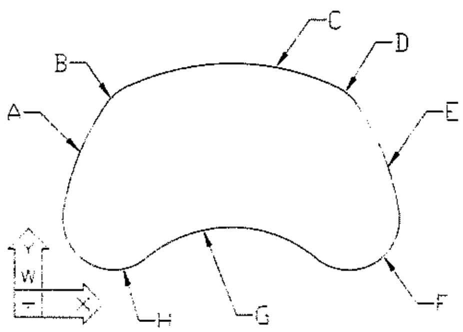


Figure 2.72 Circles trimmed

Now you have eight circular arcs. Together, they represent the top view of the model. To use these wires for making the bottom surface and side surface, you will join them into a spline. Select the Join... item of the Edit Wire cascading menu of the Surface pull-down menu to use the AMJOIN3D command.

<Surface> **<Edit Wire>** **<Join...>**

Command: **AMJOIN3D**

[Join3D
Automatic **Spline**
OK]

Select start wire: [**Select A (Figure 2.72).**]

Select wires to join: **[Select B, C, D, E, F, G, and H (Figure 2.72).]**
Select wires to join: **[Enter]**
Reverse? Yes/<No>: **[Enter]**

Now you have a spline. You will use it to construct the bottom and side surfaces of the model. Set the current layer to Surface.

[Mechanical Main] [Layer Control]

Current Layer: **Surface**

The bottom face of the model is a planar surface with a trimmed edge. To make this surface, select the Planar Trim item of the Create Surface cascading menu of the Surface pull-down menu to use the AMPLANE command. After making the surface, set the display to an isometric view. (See Figure 2.73.)

<Surface> <Create Surface> <Planar Trim>

Command: **AMPLANE**
Plane/Wires/<First corner>: **WIRE**
Select wires: **[Select A (Figure 2.72).]**
Select wires: **[Enter]**

Command: **8**

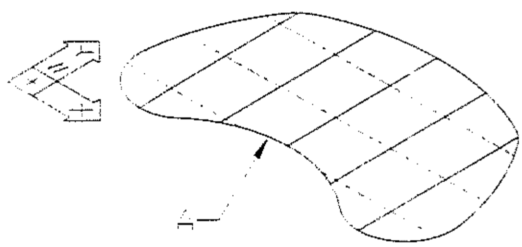


Figure 2.73 Planar surface constructed, display set

The bottom surface of the model is complete. To make the side surface, use the AMEXTRUDESF command to make an extrude surface. The extrusion height is 50 units and the taper angle is –5°. The position of the UCS icon tells you that the wire for extrusion lies on the XY plane. Naturally, the direction of extrusion is Z. (See Figure 2.74.)

<Surface> <Create Surface> <Extrude>

Command: **AMEXTRUDESF**
Select wires: **[Select A (Figure 2.73).]**
Select wires: **[Enter]**
Direction: Viewdir/Wire/X/Y/Z/<Start point>: **Z**
Distance: **50**
Flip/<Accept>: **[Accept if the direction arrow is pointing upward.]**
Taper angle: **-5**

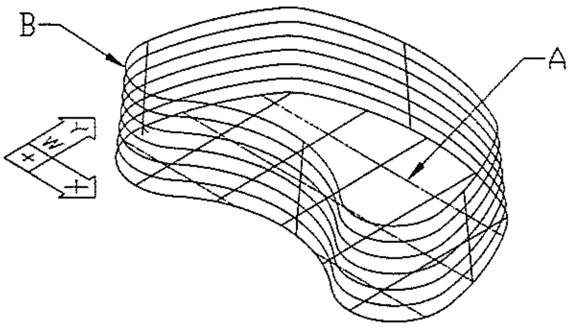


Figure 2.74 Extrude surface constructed

An extrude surface is constructed. To round off the edge between the extrude surface and the planar surface, use the AMFILLETSF command. After you select two surfaces, a dialog box appears. (See Figure 2.75.)

The Fillet Surface dialog box has three areas: Trim, Fillet Type, and Radii. The Trim area enables you to decide whether to trim the first selected surface, the second selected surface, or both surfaces. Here, trim both surfaces.

A fillet surface has an arc-shaped cross section. The Fillet Type area enables you to decide which type of fillet surface to construct. If you select the Variable box, you can select either Linear or Cubic. The fillet radii of a linear variable fillet change linearly, and the fillet radii of a cubic variable fillet change cubically from one end to the other. Because the surfaces selected for making a fillet may not have the same edge length, you can choose to extend the fillet surface to the longer edge by selecting the Extended box. In some situations, surfaces selected for filleting may have been trimmed. To construct the fillet surface in accordance with the original untrimmed surface, use the Base Surface box. Here, specify a constant fillet radius of 5 units in the Radii area. (See Figure 2.75.)

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select A (Figure 2.74).**]
Select second surface: [**Select B (Figure 2.74).**]

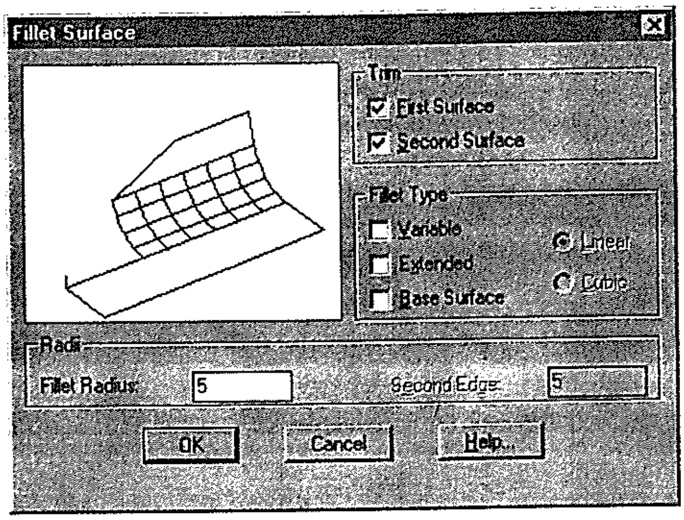


Figure 2.75 Fillet Surface dialog box

[Trim **First Surface**
Second Surface
Fillet Radius **5**
OK]

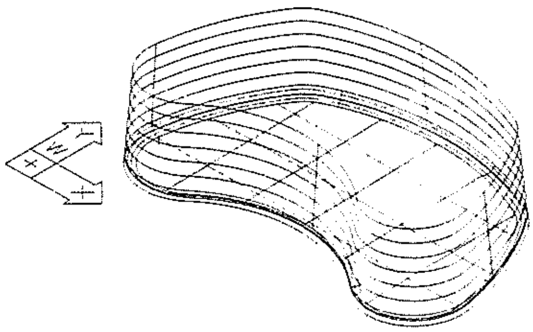


Figure 2.76 Fillet surface created

Top Surface

Now you will work on the top surface. As we have said, it is a loft u surface that builds on three splines. Set the current layer to Wire. Then construct three splines by using the SPLINE command and the COPY command. (See Figure 2.77.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
0,0,20	**120,0,30**	**240,0,20**	**[Enter]**	**[Enter]**	**[Enter]**

<Construct> **<Copy>**

Command: **COPY**
Select objects: **L**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,80,5**
Second point of displacement: **[Enter]**

<Construct> **<Copy>**

Command: **COPY**
Select objects: **L**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,80,-5**
Second point of displacement: **[Enter]**

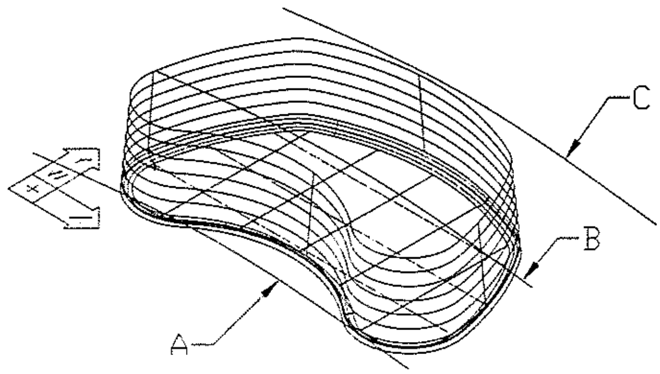


Figure 2.77 Spline constructed, splines copied

The wires for the top surface are complete. Set the current layer to Surface. Then use the AMLOFTU command. This command enables you to construct a surface from a set of curves at regular intervals. It brings up a dialog box. (See Figure 2.78.)

Select the Align box to adjust the direction of selected wires to make them unidirectional. Select the Smooth box to fit the input wires to reduce the complexity of the surface created. Select the Respace box to adjust any poorly proportioned curve ends. Finally, select the [OK] button. A loft u surface is constructed. (See Figure 2.79.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Surface**

<Surface> **<Create Surface>** **<LoftU...>**

Command: **AMLOFTU**
Select U wires: **[Select A, B, and C (Figure 2.77).]**
Select U wires: **[Enter]**

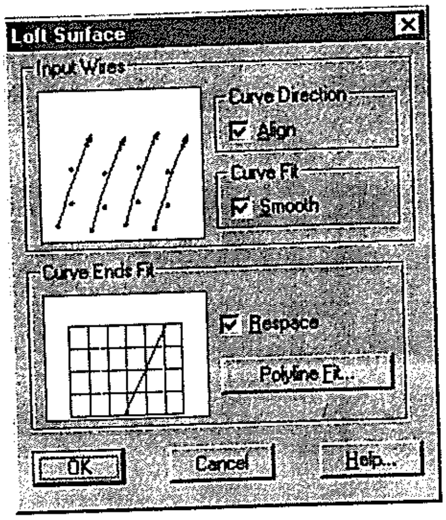


Figure 2.78 Loft Surface dialog box

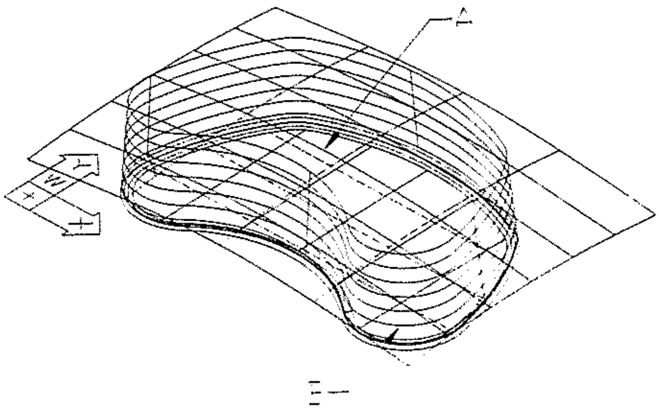


Figure 2.79 Loft u surface constructed

Now use the AMFILLETSF command to form a fillet surface between the top and side surfaces. Because the two surfaces intersect, there are four possible fillet surfaces, depending on where you select the surfaces. Here, select the central part of the loft surface and the lower part of the extrude surface. (See Figure 2.80.)

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select A (Figure 2.79).**]
Select second surface: [**Select B (Figure 2.79).**]

[Trim **First Surface**
Second Surface
Fillet Radius **5**
OK]

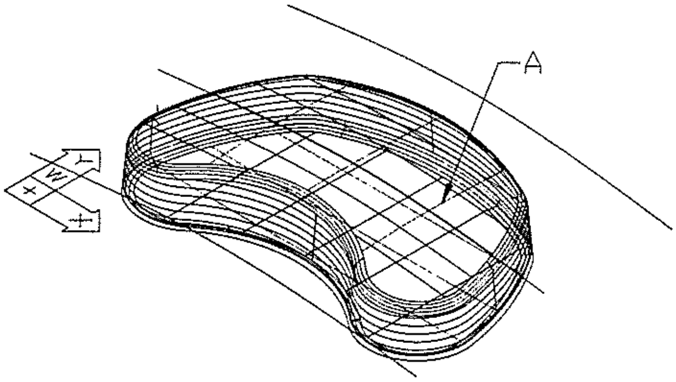


Figure 2.80 Fillet surface between the top and side surfaces constructed

Compare your drawing with Figure 2.80. If the fillet surfaces are not the same, you have probably selected the wrong places on the surfaces while filleting. If so, undo the operation and try again.

Now the surfaces for the main body are complete. Use the AMVISIBLE command to hide all objects except the top surface. (See Figure 2.81.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Objects
Hide **Select<** **]**

Select objects to hide: **ALL**
Select objects to hide: **R**
Remove objects: **[Select A (Figure 2.80).]**
Remove objects: **[Enter]**

[OK **]**

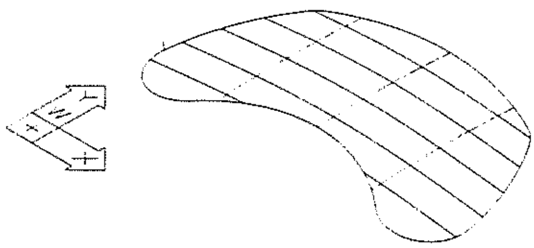


Figure 2.81 All objects except the top surface hidden

Raised Buttons

As we have said, each raised button consists of an extrude surface and a surface offset from the top surface. Set the current layer to Wire. Then construct a circle and an ellipse. (See Figure 2.82.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

<Design> **<Circle>** **<Center, Radius>**

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **50,50**
Diameter/<Radius>: **20**

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **170,75**
Axis endpoint: **@30<30**
<Other axis distance>/Rotation: **25**

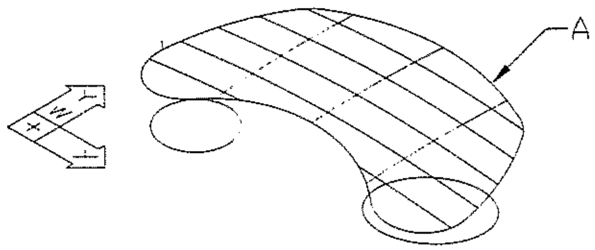


Figure 2.82 Circle and ellipse constructed

The wires for the raised buttons are complete. Set the current layer to Surface.

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

To retain the top surface after an offset surface is constructed from it, set the system variable DELOBJ to 0.

Command: **DELOBJ**
New value for DELOBJ: **0**

Now use the AMOFFSETSF command.

<Surface> <Create Surface> <Offset>

Command: **AMOFFSETSF**
Select surfaces: **[Select A (Figure 2.82).]**
Select surfaces: **[Enter]**

As shown in Figure 2.82, the normal vector of the top surface is pointing upward. Therefore, the offset surface at 6 units' distance is constructed above the original surface.

Offset distance: **6**

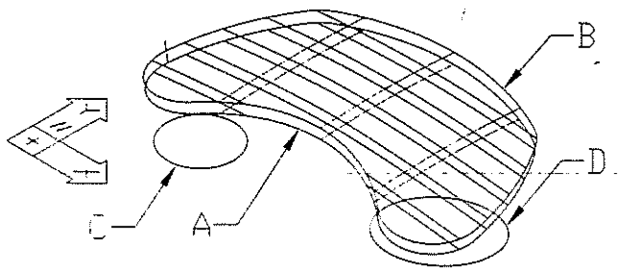


Figure 2.83 Offset surface constructed

While you are making the raised buttons, the top surface of the main body is not required. Use the AMVISIBLE command to hide it.

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Objects
Hide Select<]

Select objects to hide: **[Select A (Figure 2.83).]**
Select objects to hide: **[Enter]**

[OK]

You need two offset surfaces, one for each raised button. To make a second offset surface, use the COPY command.

<Construct> <Copy>

Command: **COPY**
Select objects: **[Select B (Figure 2.83).]**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0**
Second point of displacement: **[Enter]**

After copying, you should not see any visual difference on your screen, because the original and the copied offset surfaces reside at the same location. Now use the

AMEXTRUDESF command to construct two extrude surfaces from the circle and ellipse. After making the extrude surfaces, use the shortcut key [2] to obtain a two-viewport display. (See Figure 2.84.)

<Surface> <Create Surface> <Extrude>

Command: **AMEXTRUDESF**
Select wires: **[Select C and D (Figure 2.83).]**
Select wires: **[Enter]**
Direction: Viewdir/Wire/X/Y/Z/<Start point>: **Z**
Distance: **50**
Flip/<Accept>: **[Accept if the direction arrow is pointing upward.]**
Taper angle: **-5**

Command: **2**

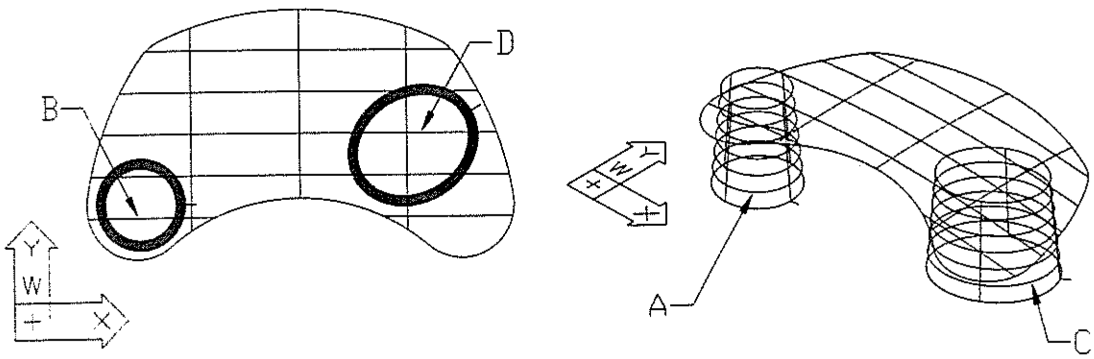


Figure 2.84 Top surface hidden, offset surface copied, extrude surfaces constructed, and display set

Now use the AMFILLETSF command to fillet and trim a copy of the offset surface and the extrude cylindrical surface. Then repeat the command to fillet and trim another copy of the offset surface and the extrude elliptical surface.

<Surface> <Create Surface> <Fillet...>

Command: **AMFILLETSF**
Select first surface: **[Select A (Figure 2.84).]**
Select second surface: **[Select B (Figure 2.84).]**

[Trim **First Surface**
Second Surface
Fillet Radius **1**
OK]

Command: **[Enter]**
AMFILLETSF
Select first surface: **[Select C (Figure 2.84).]**
Select second surface: **[Select D (Figure 2.84).]**

[Trim **First Surface**

Second Surface
Fillet Radius 1
OK]

Return to a single isometric display by using the shortcut keys [1] and [8]. (See Figure 2.85.)

Command: **1**

Command: **8**

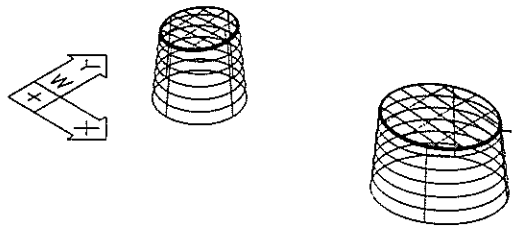


Figure 2.85 Fillet surfaces constructed on the raised buttons, display set

Turn off layer Wire and unhide the hidden top surface. (See Figure 2.87.)

[Mechanical Main] [Layer Control]

Off layer: **Wire**
Current layer: **Surface**

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Object
Unhide Select<]

After you select the [Select<] button, all the hidden objects appear. (See Figure 2.86.) Select the top surface.

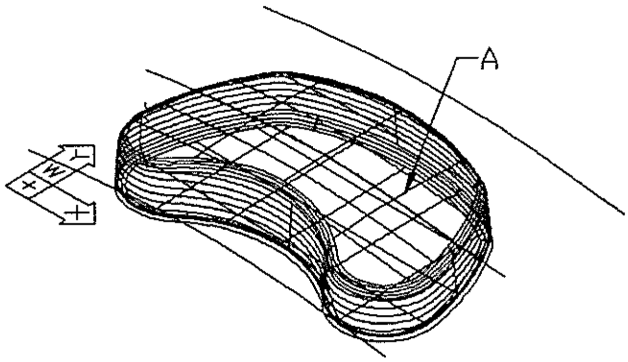


Figure 2.86 Hidden surfaces displayed for selection

Select objects to unhide: **[Select A (Figure 2.86).]**
Select objects to unhide: **[Enter]**

[OK]

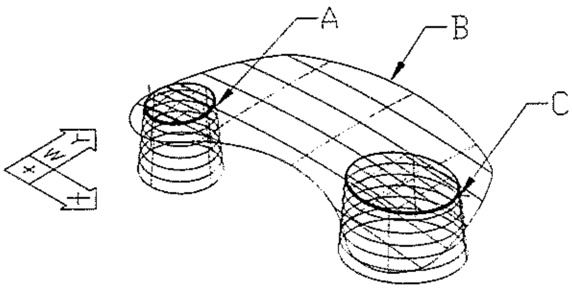


Figure 2.87 Top surface unhidden

Now the top surface of the main body is complete. To construct fillet surfaces between the side surfaces of the raised buttons and the top surface of the main body, apply the AMFILLETSF command. (See Figure 2.88.)

<Surface> <Create Surface> <Fillet...>

Command: **AMFILLETSF**
Select first surface: **[Select A, the upper end of the extrude surface (Figure 2.87).]**
Select second surface: **[Select B (Figure 2.87).]**

[Trim **First Surface**
Second Surface
Fillet Radius **1**
OK]

Command: **[Enter]**
AMFILLETSF
Select first surface: **[Select C, the upper end of the extrude surface (Figure 2.87).]**
Select second surface: **[Select B (Figure 2.87).]**

[Trim **First Surface**
Second Surface
Fillet Radius **1**
OK]

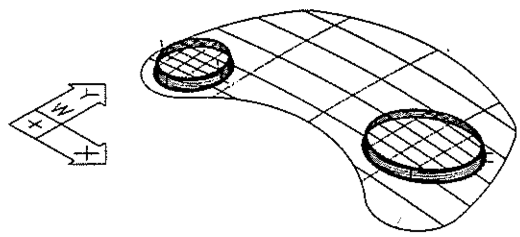


Figure 2.88 Fillet surfaces between the raised buttons and top surface constructed

Complete Model

To display all the hidden surfaces, use the AMVISIBLE command. (See Figure 2.89.) The surface model for the joy pad is complete. Save your drawing.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Object **Unhide** **All**
OK]

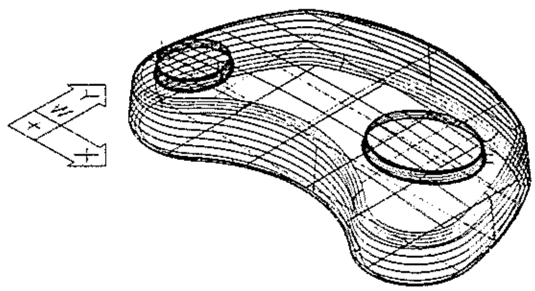


Figure 2.89 All surfaces unhidden

<File> **<Save>**

File name: **Joypad.dwg**

In constructing the model, you have learned how to construct smooth wires for making smooth surfaces, treat the intersection of surfaces by filleting, construct a planar surface, and generate an offset surface.

2.6 Desktop Visualization Tools

Mechanical Desktop has a set of visualization tools so you can visualize your 3D model. Now you will use them to visualize the joy pad model. To see how they work, select the Toolbars... item of the View pull-down menu to use the TOOLBAR command to bring out the appropriate toolbars. (See Figure 2.90.)

<View> **<Toolbars...>**

Command: **TOOLBAR**

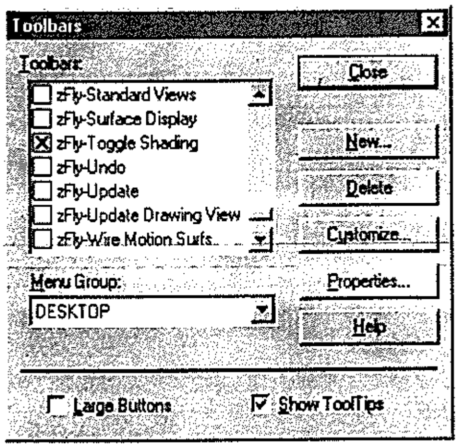


Figure 2.90 Toolbars dialog box

In the Toolbars dialog box, select the DESKTOP item of the Menu Group pull-down box. Then select the Desktop View, zFly-Rotations, and zFly-Toggle Shading items of the Toolbars scroll box. After that, select the [Close] button. (See Figure 2.91.)

Figure 2.91 Desktop View, zFly-Rotations, and zFly-Toggle Shading toolbars

The Desktop View toolbar is a comprehensive toolbar that incorporates six zFly toolbars: zFly-Rotations, zFly-Toggle Shading, zFly-Pan, zFly-Zoom Dynamic, zFly-Standard Views, and zFly-Restore View#1. It performs all the tasks that the six toolbars do. Here, you will use the zFly-Rotations and zFly-Toggle Shading toolbars.

Dynamic Rotation

Select the first icon from the left of the zFly-Rotations toolbar (Dynamic Rotation) to use the AVROTATE command. This command enables you to rotate the display in real time.

[zFly-Rotations] **[Dynamic Rotation]**

Command: **AVROTATE**
Press pick button and move cursor to rotate [Enter to exit]

Now hold down your left mouse button and move the mouse around. On your screen, you can see that the object rotates. After rotating the view to a selected position, press the [Enter] key. Now return to an isometric view by using the shortcut key [8].

Command: **8**

Toggle Shading

To select objects for toggle shading, select the Select Objects item of the Z-Fly Toggle Shading toolbar.

[Z-Fly Toggle Shading] **[Select Objects]**

Command: **AVSELECT**
Select objects: **[Select all the surfaces.]**
Select objects: **[Enter]**

Select the second icon from the left of the zFly-Toggle Shading toolbar (Display Shaded) to shade the display. This command enables you to get a rendered view quickly. (See Figure 2.92.)

[zFly-Toggle Shading] **[Display Shaded]**

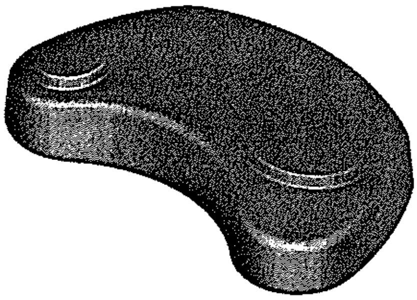


Figure 2.92 Display shaded

While the model is shaded, you can select the Dynamic Rotation icon of the zFly-Rotations toolbar to get a real-time rotated rendering. To return to a wireframe display mode, select the fourth icon from the left of the zFly-Toggle Shading toolbar (Display Wireframe).

[zFly-Toggle Shading] **[Display Wireframe]**

To quickly toggle between shaded display and wireframe display, you can select the first icon from the left of the zFly-Toggle Shading toolbar (Toggle Shading/Wireframe).

[zFly-Toggle Shading] **[Toggle Shading/Wireframe]**

To obtain a wireframe display with hidden lines removed, select the fifth icon from the left of the zFly-Toggle Shading toolbar (Display Hidden Lines). (See Figure 2.93.)

[zFly-Toggle Shading] **[Display Hidden Lines]**

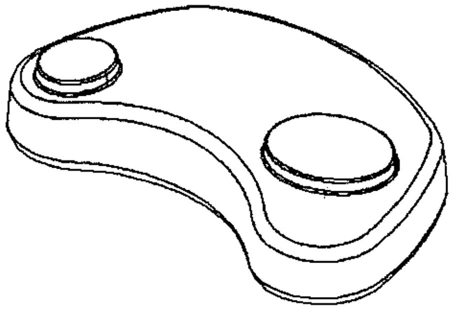


Figure 2.93 Hidden lines removed

By using the visualization tools of Mechanical Desktop, you can inspect your 3D object dynamically while constructing the model.

2.7 Mobile Phone Project

Now you will work on the surface model of the upper casing of a mobile phone. Figure 2.94 shows the rendered image; Figure 2.95 shows the surface model.

Figure 2.94 Rendered image of the mobile phone casing

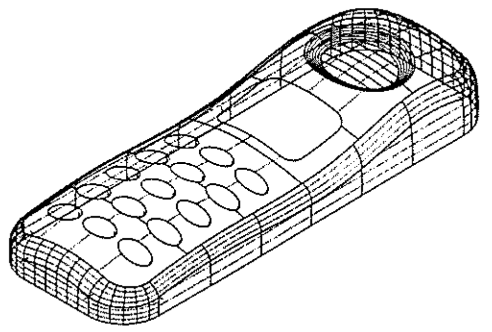


Figure 2.95 Surface model of the mobile phone

Analysis

As in Figure 2.95, this mobile phone model has three main features: main body, earpiece, and button and display panel openings.

The main body of the model consists of a number of surfaces: side, top, and fillet surfaces. Because the fillet surfaces have variable fillet radii, your approach will be different from that of the last project. Instead of constructing the sides of the model as a single surface, you will construct a number of individual surfaces. With individual side surfaces, you can construct a number of variable radii fillet surfaces. Figure 2.96 shows the top and side surfaces before filleting.

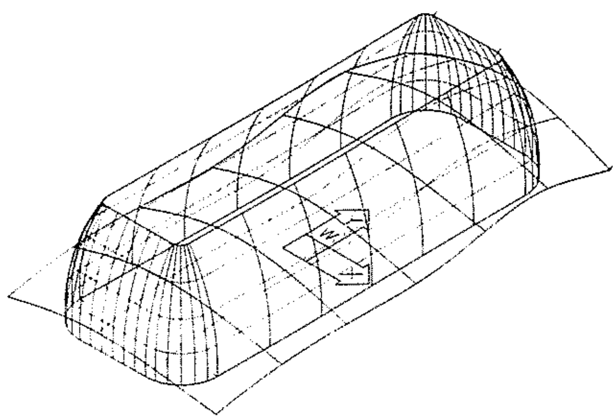


Figure 2.96 Top and side surfaces

To make the earpiece, you will construct an offset surface from the top surface and use two wires to trim the top surface and the offset surface. After trimming, you will add a rule surface at the trimmed edges. Finally, you will cut the button openings by using a number of wires. Figure 2.97 shows the wires required to make the main body, the earpiece, and the button openings.

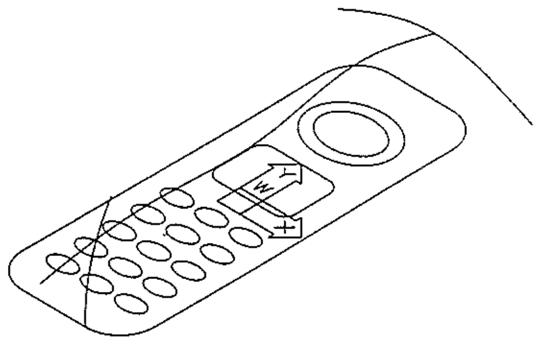


Figure 2.97 Wires required to construct the mobile phone

Drawing Setup

Start a new drawing. Use Surf_mdl.dwt as the template. Then use the AMPREFS command to set the model size to 150 millimeters and the vector length to 5 units.

<File> **<New...>**

<Surface> **<Preferences...>**

Side Surfaces

The sides of the model are a number of sweep surfaces. To construct these surfaces, you will construct a number of wires on layer Wire.

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

Select the Rectangle item of the Design pull-down menu to use the RECTANG command to construct a rectangle with rounded corners. Then use the shortcut key [F] to fit the rectangle to the display view. (See Figure 2.98.)

<Design> **<Rectangle>**

Command: **RECTANG**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **F**
Fillet radius for rectangles: **12**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **-23,-65**
Other corner: **@46,130**

Command: **F**

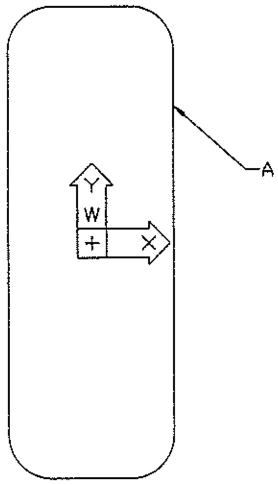


Figure 2.98 Rectangle with rounded corners constructed

Use the EXPLODE command to decompose the rectangle, which is a closed polyline, into eight lines and arc segments. You will use these segments as rails for making sweep surfaces. After exploding, set the display to an isometric view by using the shortcut key [8]. Then set the Z axis of the UCS at (1,0) of the current UCS. (See Figure 2.99.)

<Modify> **<Explode>**

Command: **EXPLODE**
Select objects: **[Select A (Figure 2.98).]**
Select objects: **[Enter]**

Command: **8**

<Assist> **<UCS>** **<Z Axis Vector>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **ZAXIS**
Origin point <0,0,0>: **[Enter]**
Point on positive portion of Z-axis: **1,0**

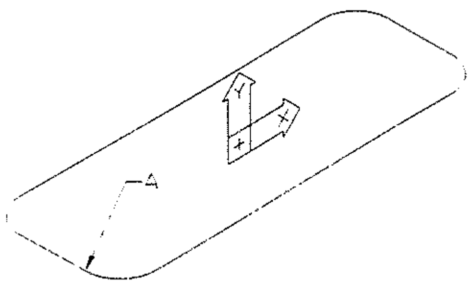


Figure 2.99 Rectangle exploded, UCS rotated

Use the ARC command to construct an arc. You will use it as the cross section for the sweep surfaces. (See Figure 2.100.)

<Design> <Arc> <Start, Center, Angle>

Command: **ARC**
Center/<Start point>: **END** of **[Select A (Figure 2.99).]**
Center/End/<Second point>: **C**
Center: **@60,3**
Angle/Length of chord/<End point>: **A**
Included angle: **-35**

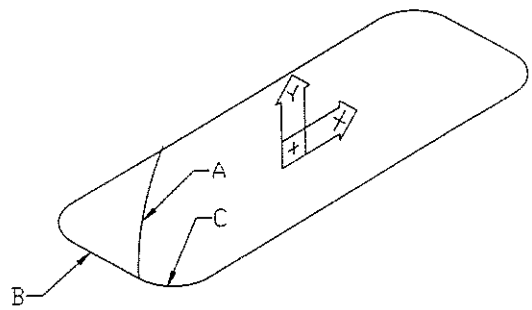


Figure 2.100 Arc constructed

The wires for the side surfaces are complete. Set the current layer to Surface.

[Mechanical Main] [Layer Control]

Current Layer: **Surface**

The sides of the main body of the mobile phone are sweep surfaces. A sweep surface is constructed by sweeping one or more cross sections along one or two rails.

Select the Sweep item of the Create Surface cascading menu of the Surface pull-down menu to use the AMSWEEPSF command. Use the arc as the cross section and a segment of the rectangle as the rail. (See Figure 2.101, the Sweep Surface dialog box.) There are three ways to define the cross section in relation to the rail: Normal, Parallel, or Direction. Select Normal orientation and then the [OK] button.

<Surface> <Create Surface> <Sweep>

Command: **AMSWEEPSF**
Select cross sections: **[Select A (Figure 2.100).]**
Select cross sections: **[Enter]**
Select rails: **[Select B (Figure 2.100).]**
Select rails: **[Enter]**

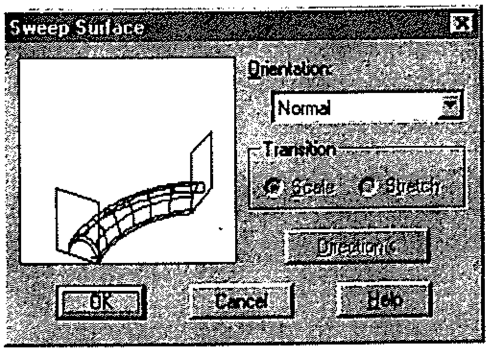


Figure 2.101 Sweep Surface dialog box

Now repeat the AMSWEEPSF command to construct another sweep surface. (See Figure 2.102.)

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: **[Select A (Figure 2.100).]**
Select cross sections: **[Enter]**
Select rails: **[Select C (Figure 2.100).]**
Select rails: **[Enter]**

[Normal OK]

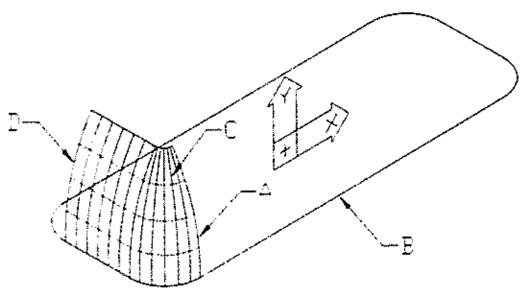


Figure 2.102 Sweep surfaces constructed

Repeat the AMSWEEPSF command. Then use the MIRROR3D command to make a mirror copy of the two sweep surfaces. (See Figure 2.103.)

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: **[Select A (Figure 2.102).]**
Select cross sections: **[Enter]**
Select rails: **[Select B (Figure 2.102).]**
Select rails: **[Enter]**

[Normal OK]

<Construct> **<3D Transitions>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select C and D (Figure 2.102).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **YZ**
Point on YZ plane <0,0,0>:**[Enter]**
Delete old objects? <N> **[Enter]**

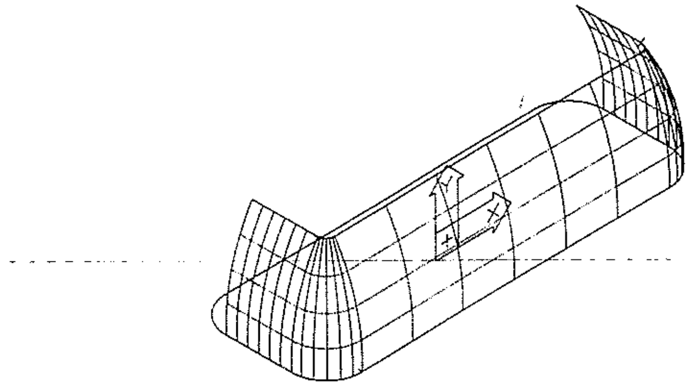


Figure 2.103 A sweep surface constructed and two sweep surfaces mirrored

Now you have five sweep surfaces. Because the mobile phone is symmetrical about its own axis, you will construct the remaining sweep surfaces at a later stage by mirroring.

Top Surface

Now you will work on the top surface. It is also a sweep surface. You will construct two wires. Set the current layer to Wire. Then construct two splines. (See Figure 2.104.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
70,15	**40,20**	**10,15**	**-30,15**	**-60,12**	**-70,10**	**[Enter]**	**[Enter]**	**[Enter]**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
70,5,35	**70,15,0**	**70,5,-35**	**[Enter]**	**[Enter]**	**[Enter]**

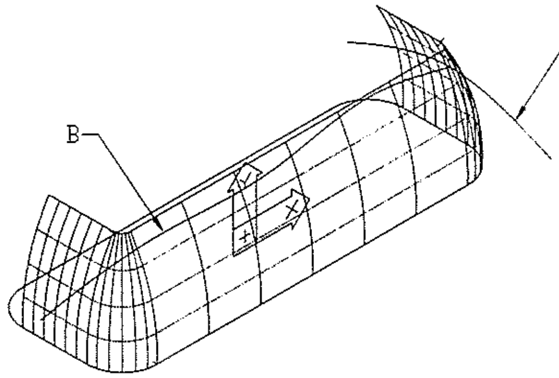


Figure 2.104 Splines constructed

The wires for the top surface are complete. Set the current layer to Surface. Then use the AMSWEEPSF command to construct a sweep surface. (See Figure 2.105.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Surface**

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: [**Select A (Figure 2.104).**]
Select cross sections: [**Enter**]
Select rails: [**Select B (Figure 2.104).**]
Select rails: [**Enter**]

[Normal OK]

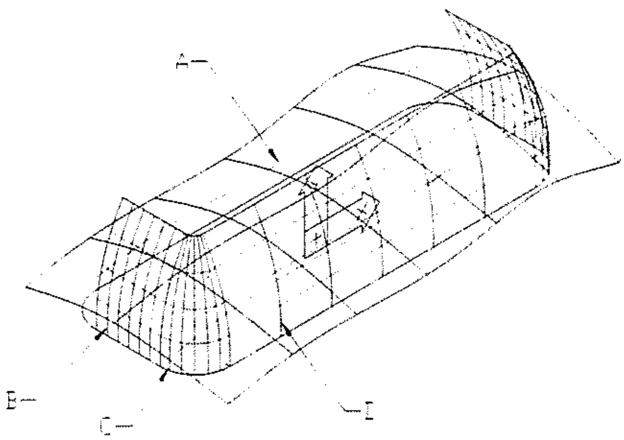


Figure 2.105 Sweep surface constructed

Fillet Surfaces

As shown in Figure 2.105, the top surface intersects with the side surfaces. To treat the intersecting surfaces, you will construct five fillet surfaces. While filleting, you will trim only the side surfaces and not the top surface. To trim the top surface, you will copy the edges of the fillet surfaces and use them as trimming wires.

While filleting, you should select the surfaces in sequence according to which surface you want to trim. If you follow the delineation below to select the side surface first and then select the top surface, you should trim the first surface, but not the second.

Now run the AMFILLETSF command to create a constant fillet surface.

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select B (Figure 2.105).**]
Select second surface: [**Select A (Figure 2.105).**]

[Trim **First Surface**
Fillet Radius **5**
OK]

The second fillet surface is a variable fillet. It varies linearly from 5 units to 4 units. In selecting the surfaces, take great care to ensure that you select a point near the edge of the surface where you put the start fillet radius.

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select C (Figure 2.105).**]
Select second surface: [**Select A (Figure 2.105).**]

[Trim **First Surface**
Fillet Type: **Variable** **Linear**
First edge: **5**
Second edge: **4**
OK]

Now construct the third fillet. It is a cubical variable fillet that changes from 4 units to 6 units.

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select D (Figure 2.105).**]
Select second surface: [**Select A (Figure 2.105).**]

[Trim **First Surface**
Fillet Type: **Variable** **Cubic**
First edge: **4**
Second edge: **6**
OK]

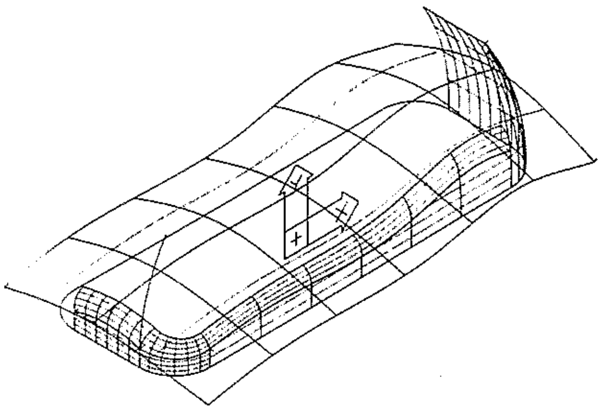


Figure 2.106 Three fillet surfaces constructed

Check your drawing against Figure 2.106. If the fillet surfaces are not the same, you probably selected the wrong places on the surfaces while filleting. If so, undo the operation and try again. Before you fillet the remaining edges, set the display to a back right isometric view. (See Figure 2.107.)

<View> **<Model Views>** **<Back Right Iso>**

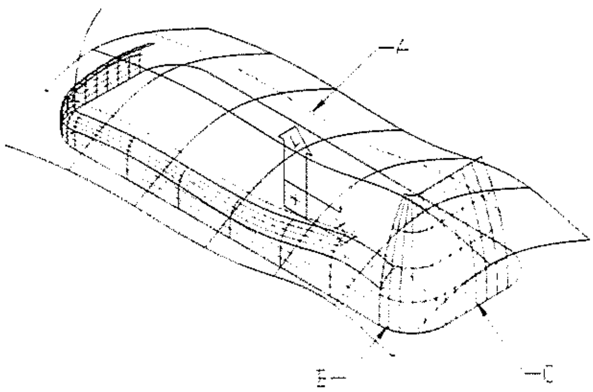


Figure 2.107 Display set to back right isometric

Use the AMFILLETSF command to construct a cubic variable fillet.

<Surface> **<Create Surface>** **<Fillet...>**

Command: **AMFILLETSF**
Select first surface: [**Select B (Figure 2.107).**]
Select second surface: [**Select A (Figure 2.107).**]

[Trim **First Surface**
Fillet Type: **Variable** **Cubic**
First edge: **6**
Second edge: **7**

OK]

Use the AMFILLETSF command to construct a constant-radius fillet.

<Surface> <Create Surface> <Fillet...>

Command: **AMFILLETSF**
Select first surface: [**Select C (Figure 2.107).**]
Select second surface: [**Select A (Figure 2.107).**]

[Trim **First Surface**
Fillet Radius **7**
OK]

Now check your drawing against Figure 2.108.

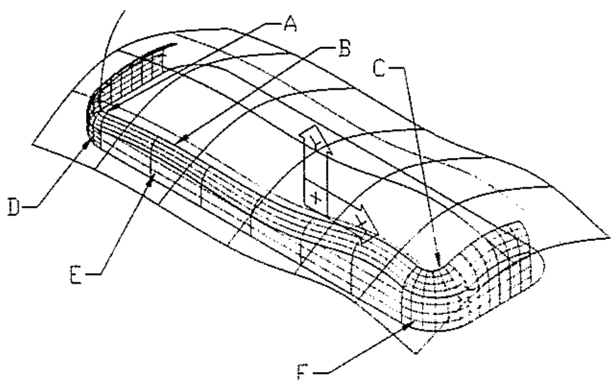


Figure 2.108 Two fillet surfaces constructed

Main Body

Because the phone is symmetrical about its Y axis, use the MIRROR3D command to mirror three fillet surfaces and three side surfaces. (See Figure 2.109.)

<Construct> <3D Transitions> <3D Mirror>

Command: **MIRROR3D**
Select objects: [**Select A, B, C, D, E, and F (Figure 2.108).**]
Select objects: [**Enter**]
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **XY**
Point on XY plane <0,0,0>: [**Enter**]
Delete old objects? <N> [**Enter**]

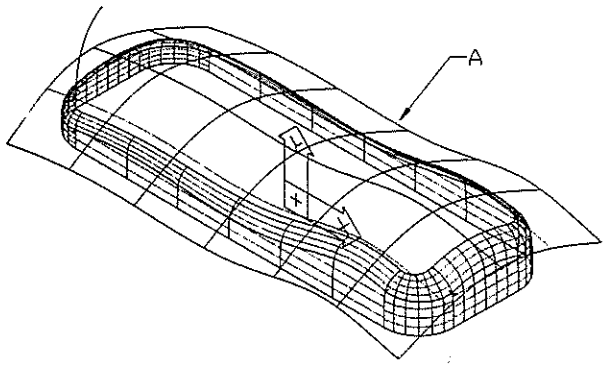


Figure 2.109 Side and fillet surfaces mirrored

The side and fillet surfaces are complete. Now use the GROUP command to put all the entities except the top surface into an entity group called S1 so that you can select them all at once.

<Construct> **<Group...>**

Command: **GROUP**

[Group name: **S1**
Selectable: **Yes**
New<]

Select objects for grouping:
Select objects: **ALL**
Select objects: **R**
Remove objects: [**Select A (Figure 2.109).**]
Remove objects: [**Enter**]

[**OK**]

Use the shortcut key [5] to set the display to top view. (See Figure 2.110.)

Command: **5**

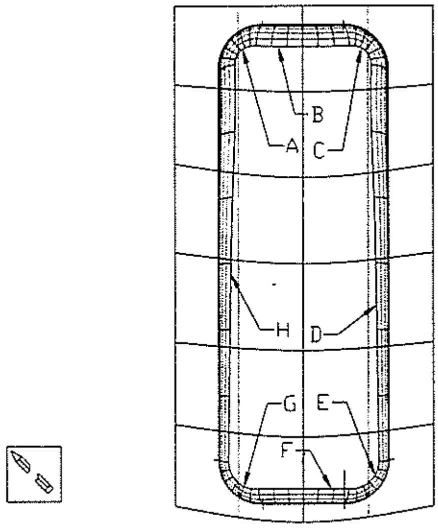


Figure 2.110 Top view

To trim the top surface, you need to copy the edges from the fillet surfaces. Set the current layer to Wire.

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

Use the AMEDGE command to copy the upper edges of the fillet surfaces. As you copy, some green wires will appear at the selected edges. They are the copied wires residing on layer Wire.

<Surface> **<Create Wireframe>** **<Copy Edge>**

Command: **AMEDGE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **OUTPUT**
Polyline/<Spline>: [**Enter**]
(Output = Spline)
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **COPY**
Select surface edge: [**Select A, B, C, D, E, F, G, and H (Figure 2.110).**]
Select surface edge: [**Enter**]

To see the copied edges clearly, run the AMVISIBLE command to hide the entity group S1. (See Figure 2.111.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Objects
Hide **Select<** **]**

Select objects to hide: **G**

Enter group name: **S1**
Select objects to hide: **[Enter]**

[OK]

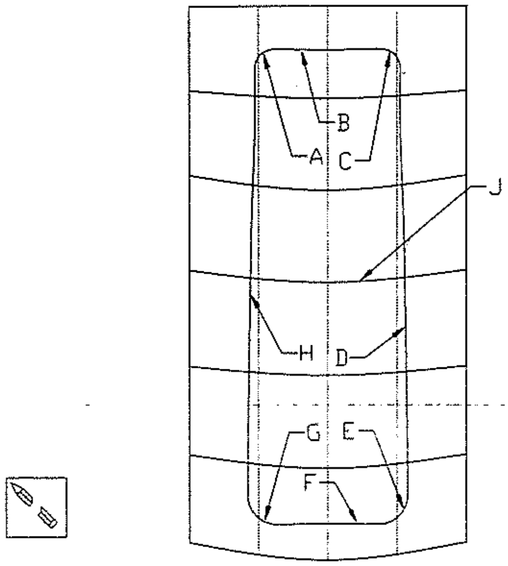


Figure 2.111 Edges copied, entity group S1 hidden

Using the copied edges as wires to project, run the AMPROJECT command to trim the top surface. Select the central part of the surface, because you have to retain this portion of the surface after trimming. (See Figure 2.112, the Project To Surface dialog box.) Direction is Normal to surface and Output Type is Trim surface. (See Figure 2.113.)

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select A, B, C, D, E, F, G, and H (Figure 2.111).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select J (Figure 2.111).]**
Select target surfaces: **[Enter]**

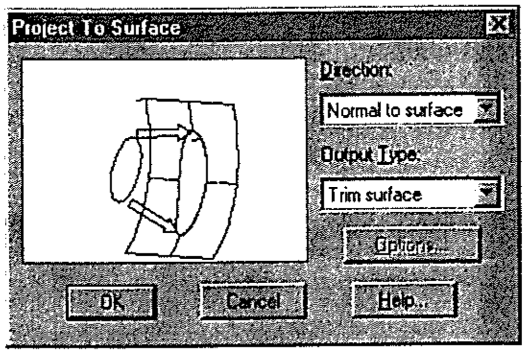


Figure 2.112 Project To Surface dialog box

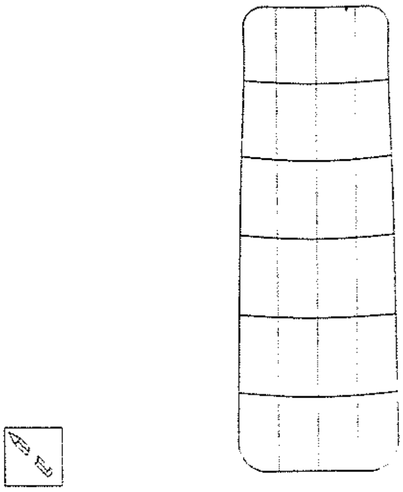


Figure 2.113 Top surface trimmed

The main body of the mobile phone surface model is complete.

Earpiece

The earpiece consists of an opening in the top surface and two additional surfaces: an offset surface and a rule surface. To cut a hole and make these surfaces, you need to construct two wires. Set the UCS to World. Then use the ELLIPSE command to construct two ellipses. (See Figure 2.114.)

<Assist> <UCS> <World>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Design> <Ellipse> <Center>

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **0,40**
Axis endpoint: **@15<0**

<Other axis distance>/Rotation: **12**

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **0,40**
Axis endpoint: **@11<0**
<Other axis distance>/Rotation: **8**

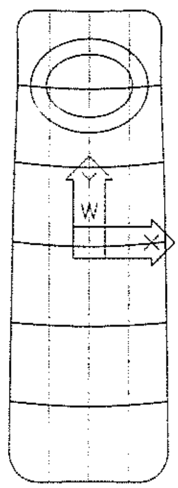


Figure 2.114 UCS set, ellipses constructed

Use the shortcut key [8] to set the display to an isometric view. (See Figure 2.115.)

Command: **8**

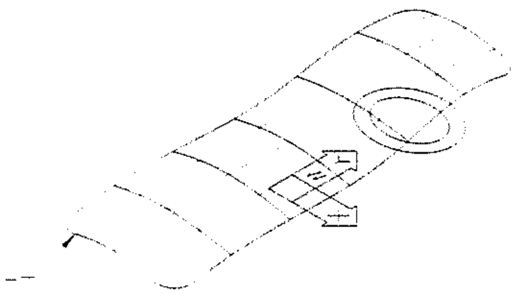


Figure 2.115 Display set to an isometric view

To make the earpiece. you have to derive an offset surface that resides in the minus Z direction of the top surface. Before you use the AMOFFSETSF command, you have to do two things: check the normal direction of the top surface and set the system variable DELOBJ.

Examine the normal direction of the top surface. (See A in Figure 2.115.) It should point in the positive Z direction. If not, use the AMEDITSF command to reverse the normal direction.

The system variable DELOBJ determines whether the original surface is deleted after an offset surface is created from it. To keep the original top surface, set DELOBJ to 0.

Command: **DELOBJ**
New value for DELOBJ: **0**

Now set the current layer to Surface. Then use the AMOFFSETSF command to make an offset copy of the top surface. (See Figure 2.116.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Surface**

<Surface> **<Create Surface>** **<Offset>**

Command: **AMOFFSETSF**
Select surfaces: **[Select A (Figure 2.115).]**
Select surfaces: **[Enter]**
Offset distance: **-3**

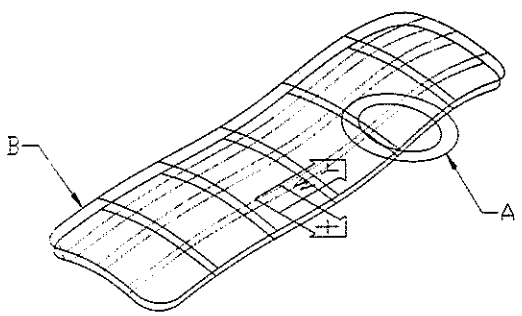


Figure 2.116 Offset surface constructed

Now use the larger ellipse to project and trim the upper, original surface. Select the edge of the surface to cut an elliptical hole. (See Figure 2.117.)

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: **[Select A (Figure 2.116).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select B (Figure 2.116).]**
Select target surfaces: **[Enter]**

[Direction: **Normal to UCS**
Output type: **Trim surface**
OK]

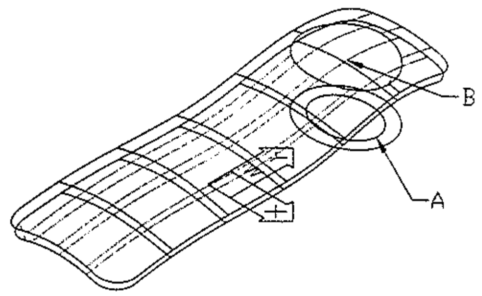


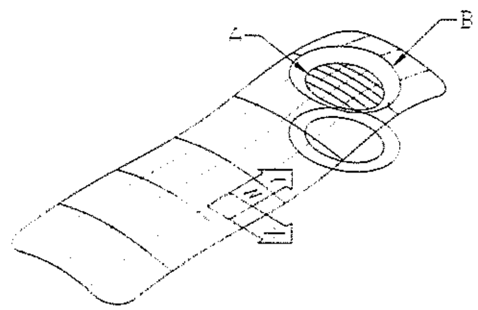
Figure 2.117 Elliptical hole cut in the top surface

Repeat the AMPROJECT command to trim the offset surface. Select a point directly above the smaller ellipse so that an elliptical surface is retained after trimming. (See Figure 2.118.)

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: **[Select A (Figure 2.117).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select B (Figure 2.117).]**
Select target surfaces: **[Enter]**

[Direction: **Normal to UCS**
Output type: **Trim surface**
OK]



2.118 Elliptical hole cut, elliptical surface constructed

To complete the earpiece, use the AMRULE command to produce a rule surface that runs between the trimmed edges of the top surface and the offset surface. (See Figure 2.119.)

<Surface> **<Create Surface>** **<Rule>**

Command: **AMRULE**
Select first wire: **[Select A (Figure 2.118).]**
Select second wire: **[Select B (Figure 2.118).]**

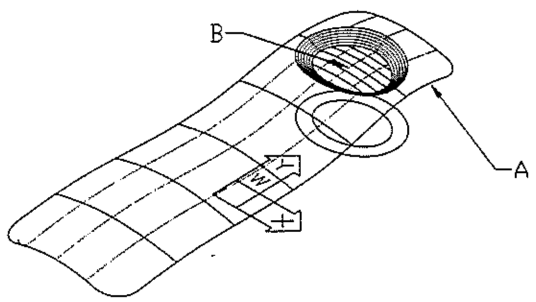


Figure 2.119 Rule surface constructed

The earpiece is complete. As noted above, a trimmed surface consists of a base surface and a trimmed boundary. To display the base surfaces of the top surface and the offset surface, select the Surface Display... item of the Surface pull-down menu. (See Figure 2.120, the Individual Surface Display dialog box.) Select the Show Base Surface box and then the [OK] button. (See Figure 2.121.)

<Surface> **<Surface Display...>**

Command: **AMDISPSF**
Select surfaces: **[Select A and B (Figure 2.119).]**
Select surfaces: **[Enter]**

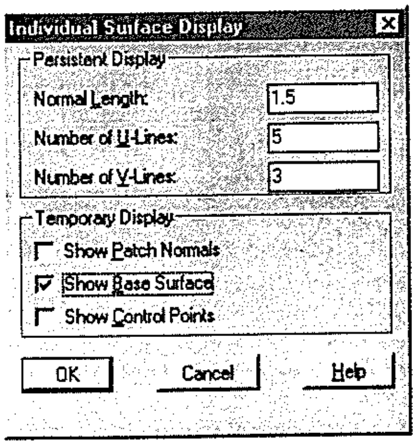


Figure 2.120 Individual Surface Display dialog box

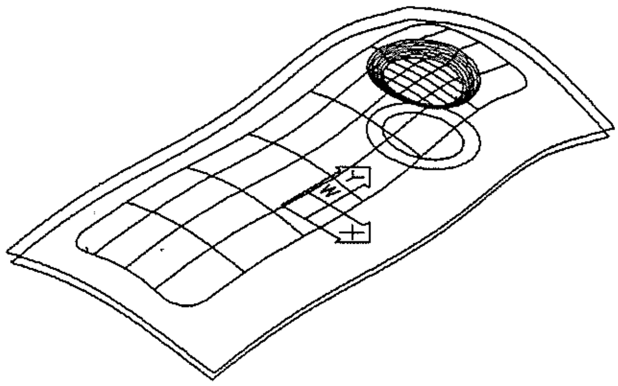


Figure 2.121 Base surfaces of the top and offset surfaces

As in Figure 2.121, the base surfaces are much larger than those required to define the profile and silhouettes of the surfaces. To reduce the memory size of the drawing file, you can reduce the base surfaces by truncation. Select the Truncate item of the Edit Surface cascading menu of the Surface pull-down menu.

<Surface> <Edit Surface> <Truncate>

Command: **AMEDITSF**
Select surfaces: [**Select A and B (Figure 2.119).**]
Select surfaces: [**Enter**]

After truncating, use the REDRAW command to refresh the screen. Then use the AMDISPSF command to display the base surfaces again. (See Figure 2.122.)

Command: **REDRAW**

<Surface> <Surface Display...>

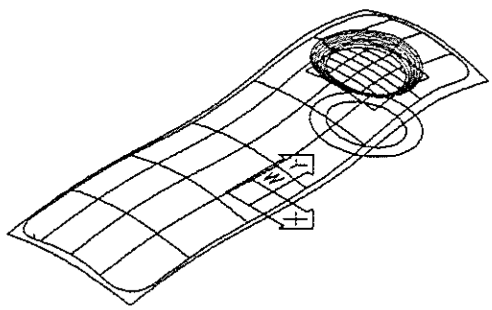


Figure 2.122 Base surfaces truncated

Panel and Button Openings

To complete the model, construct a series of wires to cut a number of openings. Set the current layer to Wire. Then construct a rectangle with rounded corners and construct 15 ellipses in 5 rows and 3 columns. (See Figure 2.123.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Wire**

<Design> **<Rectangle>**

Command: **RECTANG**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **F**
Fillet radius for rectangles: **4**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **-14,2**
Other corner: **@28,20**

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **-12,-10**
Axis endpoint: **@5<0**
<Other axis distance>/Rotation: **3**

<Construct> **<Array>** **<Rectangular>**

Command: **ARRAY**
Select objects: **L**
Select objects: **[Enter]**
Rectangular or Polar array (<R>/P): **R**
Number of rows (---): **5**
Number of columns (|||): **3**
Unit cell or distance between rows (---): **-10**
Distance between columns (|||): **12**

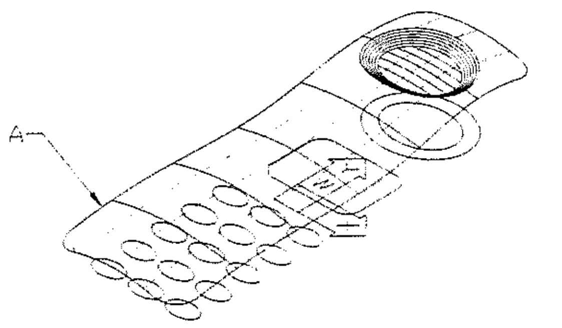


Figure 2.123 Wires for the panel and button openings

Project the wires to trim the upper surface. (See Figure 2.124.)

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: **[Select the rectangle and the ellipses.]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select A (Figure 2.123).]**
Select target surfaces: **[Enter]**

[Direction: **Normal to UCS**
Output type: **Trim surface**
OK]

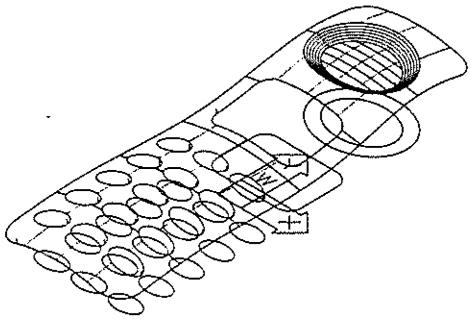


Figure 2.124 Panel and button openings cut

Unhide all the hidden surfaces. Then set the current layer to Surface and turn off layer Wire. (See Figure 2.125.)

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Unhide All OK]

[Mechanical Main] [Layer Control]

Off Layer: **Wire**
Current Layer: **Surface**

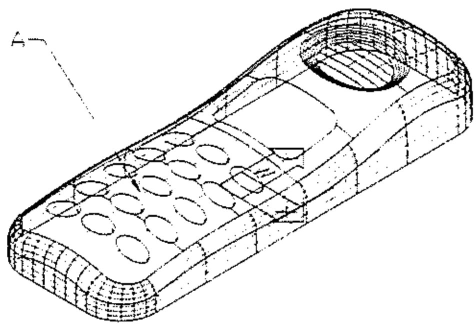


Figure 2.125 Completed model

The mobile phone model is complete. Save your drawing.

<File> <Save>

File name: **Phone.dwg**

In making this model, you constructed a number of intersecting surfaces. Because the top surface of the mobile phone intersects with more than one side surface, you did not trim it while filleting. You trimmed only the side surfaces. Then you copied the trimmed edges of the fillets. After that, you used the wires to project and trim the top surface. To construct the earpiece, you constructed offset surfaces rather than offsetting wires and generating the surfaces from the offset wires.

2.8 Surface Modeling Utilities

NURBS surface models are ideal 3D free-form objects stored in a computer for use in manufacturing processes. With a NURBS surface model, you can output flow lines, parting lines, cross sections, and augmented lines.

Add an additional layer called Uty and set it as the current layer. You will construct flow lines, parting lines, cross sections, and augmented lines on this layer.

<Assist> <Format> <Layer...>

Command: **LAYER**

[New Layer: **UTY**
Current Layer: **UTY**
OK]

Flow Lines

Flow lines are U- and V-wire meshes representing a surface in two orthogonal directions. In making the mesh, you can specify any number of U- and V-wires, regardless of the current UV display line number or UV patch line number.

Run the AMFLOW command to see how the flow lines look. (See Figure 2.126, the Surface Flow Lines dialog box.) Set the numbers of U-Wires and V-Wires to 10. Then select the Save box and the [OK] button. (See Figure 2.127.)

<Surface> <Create Wireframe> <Flow...>

Command: **AMFLOW**
Select surfaces: [**Select the top surface.**]
Select surfaces: [**Enter**]

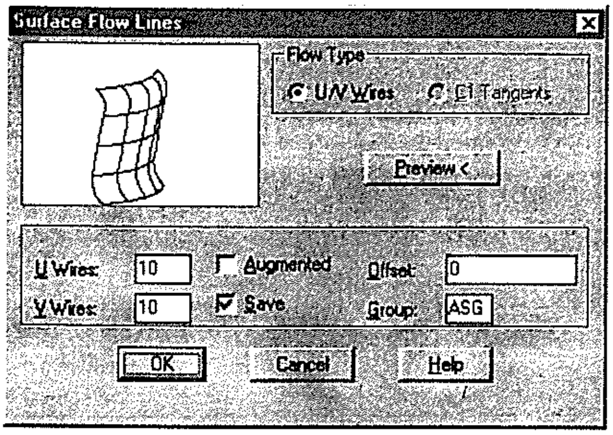


Figure 2.126 Surface Flow Lines dialog box

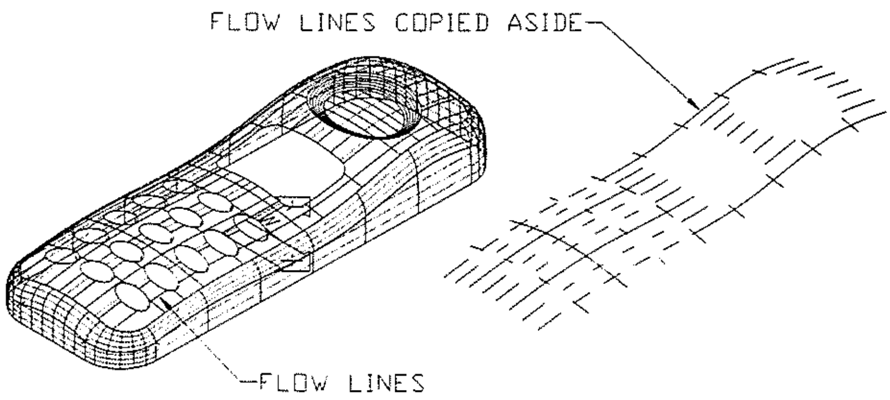


Figure 2.127 Flow lines created and copied aside

As you can see, the flow lines do not include the edges of the surface. If you need the edges as well, use the AMEDGE command.

Like the edges of a surface, the resulting flow lines can be treated as ordinary wires for defining a surface, trimming a surface, or projecting onto a surface to obtain other wires. If you need the wires, you should save them. In addition to setting the wire mesh density, you can output offset flow lines. Offset flow lines are much the same as flow lines generated from an offset surface. If you want a set of wires offset at a distance from a given surface, you do not have to make an offset surface for outputting flow lines. Instead, you can specify an offset distance while making the flow lines.

Parting Lines

When you use a NURBS surface model in mold making, you need a parting line to split the surface model into two halves. To obtain a parting line, you simply use the AMPARTLINE command. (See Figure 2.128.)

<Surface> **<Create Wireframe>** **<Parting Line>**

Command: **AMPARTLINE**
Select surfaces: **[Select all the surfaces.]**
Select surfaces: **[Enter]**
Direction: Viewdir/Wire/X/Y/Z/<Start point>: **Z**

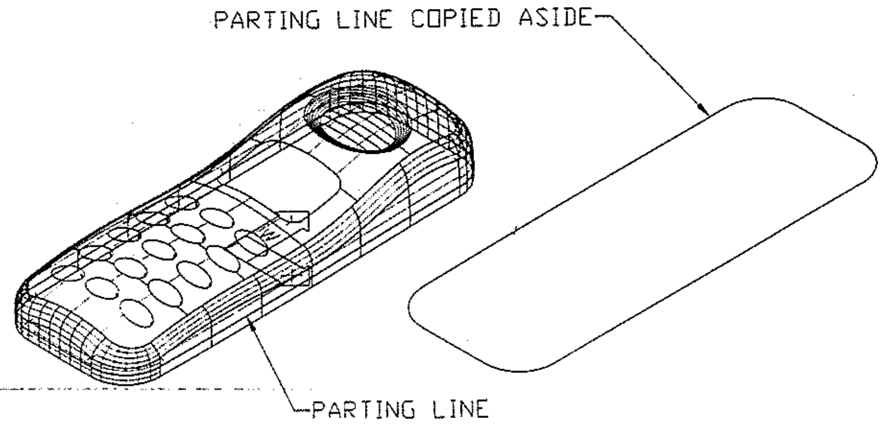


Figure 2.128 Parting lines created and copied aside

Section Lines

Flow lines are wires on a single surface. If you want to generate a cross section across a set of surfaces, use the AMSECTION command. This command outputs a series of wires at regular intervals. In addition to using the sections as wires, you can use them to verify and check your design, and as tool paths for machining the surface.

Set the UCS to a new orientation.

<Assist> **<UCS>** **<Z Axis Vector>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **ZAXIS**
Origin point <0,0,0>: **[Enter]**
Point on positive portion of Z-axis: **1,0**

Use the AMSECTION command to generate a section line across the surface model. (See Figure 2.129, the Surface Cross Sections dialog box.) Enter a value of 100 in the Stop box and select the [Define<] box to set the initial plane. After selecting the surfaces and returning to the Surface Cross Sections dialog box, select the [OK] button. (See Figure 2.130.)

<Surface> **<Create Wireframe>** **<Section Cuts...>**

Command: **AMSECTION**
Select surfaces: **ALL**
Select surfaces: **[Enter]**

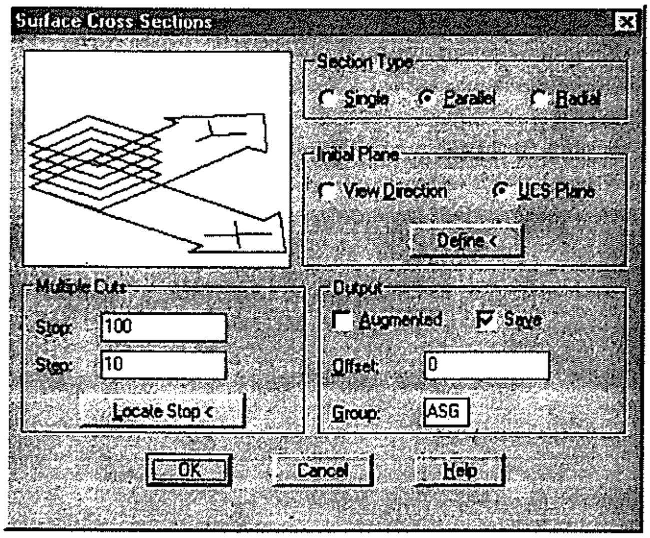


Figure 2.129 Surface Cross Sections dialog box

Locate UCS origin: **0,0,-50**

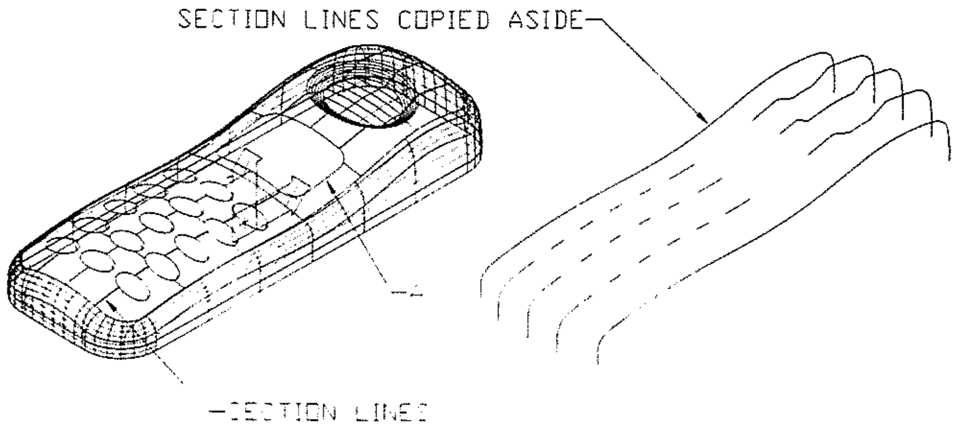


Figure 2.130 Section lines created and copied aside

Augmented Lines

Advanced 5-axis machines have two more cutting-tool motions available in addition to X, Y, and Z movements. These are rotational motions about the X axis and the Y axis. Naturally, simple 3D tool paths from the U and V flow lines or the section lines do not provide adequate information to these machines.

To meet such a need, you can generate augmented lines from the surface model. Augmented lines are polylines with normal vectors along them. Use the AMAUGMENT command. (See Figure 2.131.)

<Surface> <Create Wireframe> <Augmented Lines>

Command: **AMAUGMENT**

(Angle = 0, Distance = 0, Spacing = Optimal)
Angle/Distance/Spacing/<sElect surface wire>: **S**
Step/Vertices/<Optimal>: **S**
Step: **3**
(Angle = 0, Distance = 0, Spacing = Step)
Angle/Distance/Spacing/<sElect surface wire>: **[Select A (Figure 2.130).]**
Angle/Distance/Spacing/<sElect surface wire>: **[Enter]**

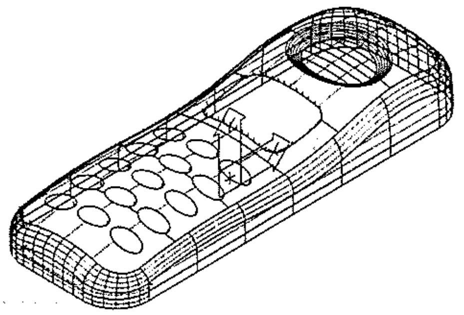


Figure 2.131 Augmented lines created

If you compare your drawing with Figure 2.131, the lengths of the augmented lines may not be the same. To change their lengths, you can use the AMSURFVARS command before making the augmented lines, or you can edit the augmented lines later.

The default directions of the vectors on the augmented lines are 0°, that is, normal to the surface. To meet practical needs, you may edit these vectors.

Set the system variable DELOBJ to 1 so that the original augmented line is deleted after editing.

Command: **DELOBJ**
New value for DELOBJ: **1**

Now run the AMEDITAUG command to change the vector length and to rotate the vectors. (See Figure 2.132.)

<Surface> <Edit Wireframe> <Vector Length>

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **N**
Select augmented lines: **[Select the augmented line.]**
Select augmented lines: **[Enter]**
Normal length: **10**

<Surface> <Edit Wireframe> <Rotate Vectors>

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **R**
Plane/<Angle>: **A**
Angle: **45**
All/Range/<Select vector>: **ALL**
Select augmented line: **[Select the augmented line.]**
Select augmented line: **[Enter]**

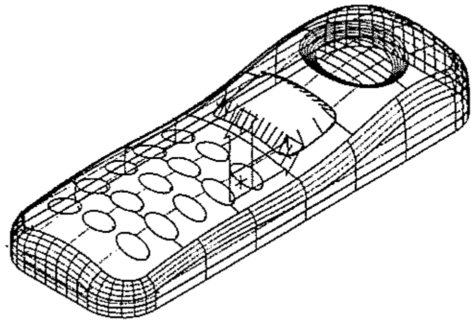


Figure 2.132 Augmented lines edited

Mass Properties

A surface has no thickness. Therefore, you cannot ask the computer to calculate the mass property of a surface unless you assign a thickness to it.

Suppose the thickness of this mobile phone casing is 1.5 units and the density of the material used to make it is 2. Use the AMSURFPROP command to find out the properties of this model shell.

<Surface> <Utilities> <Mass Properties>

Command: **AMSURFPROP**
Select surfaces: [**Select all the surfaces.**]
Select surfaces: [**Enter**]
Density/Thickness/tYpe/<Calculate properties>: **D**
Density: **2**
Density/Thickness/tYpe/<Calculate properties>: **T**
Thickness: **1.5**
Density/Thickness/tYpe/<Calculate properties>: **Y**
Shell/<Enclosed model>: **S**
Density/Thickness/tYpe/<Calculate properties>: [**Enter**]

Depending on the setting of the system variable CMDDIA, the result is displayed at the prompt line or in the dialog box. The following messages appear at the command line if the CMDDIA setting is 0:

```
---------------- SURFACES ----------------

Type:        Shell
Thickness:   1.5
Density:     2
Area:        8227.3122
Mass:        24681.9367
```

If the CMDDIA setting is 1, a dialog box similar to the one shown in Figure 2.133 appears.

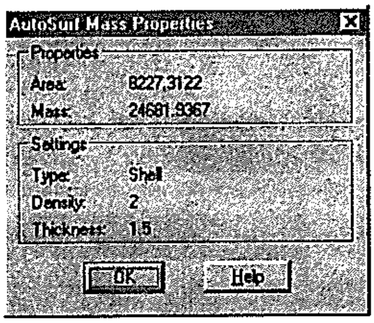


Figure 2.133 AutoSurf Mass Properties dialog box

Import and Export

In pursuing a project that involves making a surface model, you may have to input data from or output data to other applications. Depending on the compatibility of other applications with Mechanical Desktop, you may be able to use the IGESIN command for inputting and use the IGESOUT command for outputting.

Command: **IGESIN**

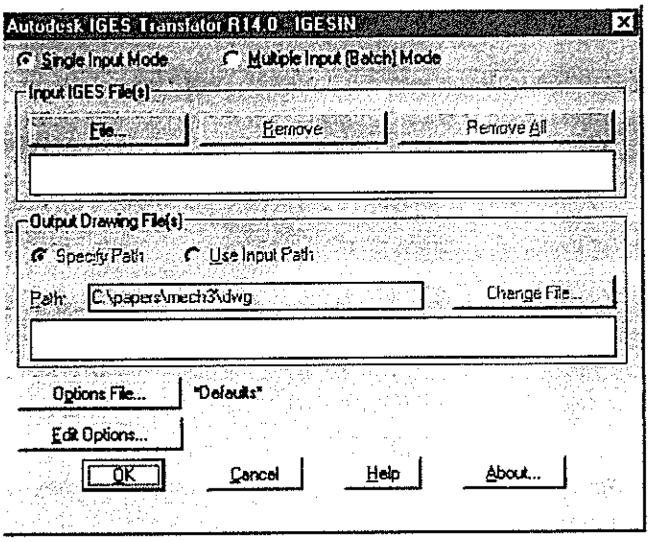


Figure 2.134 Autodesk IGES Translator R14.0 - IGESIN dialog box

Command: **IGESOUT**

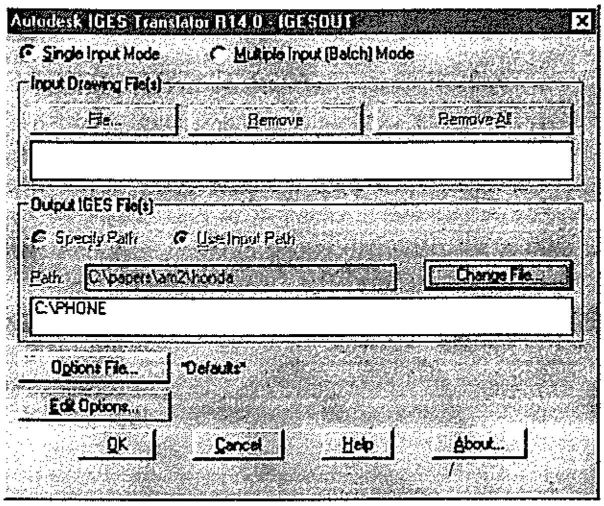


Figure 2.135 Autodesk IGES Translator R14.0 - IGESOUT dialog box

Export to 3D Studio Max and STL Format

To get a photo-realistic image from a surface model, you can output in 3DS format by using the EXPORT command. Before exporting, set the FACETRES variable to adjust the facet resolution. A value of 10 gives the highest resolution.

Command: **FACETRES**
New value for FACETRES: **10**

<File> **<Export...>**

Command: **EXPORT**

In the Export Data dialog box, select 3D Studio as the output file type. Then you can import the file to 3D Studio Max by selecting the Import... item of the File pull-down menu. After a NURBS surface is imported to the 3D Studio Max environment, you can output the model in STL format by selecting the Export... item of the File pull-down menu. In the Select File to Export dialog box, set the output file type to STL. (STL will be explained in Chapter 3.)

2.9 Operation on Solids

There are two kinds of solids: native solids constructed by using AutoCAD commands and parametric solids constructed by using Mechanical Desktop commands. Solids are usually regular in shape. To add free-form features to a solid model, you can use a surface to cut a solid. A surface that is used to cut a solid must have all its boundary edges lying outside the boundary of the solid. In this chapter, you will use surfaces to cut native solids. In the next chapter, you will use surfaces to cut parametric solids.

To explore the concept of using surfaces to cut native solids, you will work on three projects: a remote control casing, a helical spring, and a bevel gear.

Remote Control Casing

Figure 2.136 shows the rendered image of the solid model of the casing of a remote control. This model consists of three major features: a native solid constructed by extruding a 2D region, a top face that is a free-form surface, and a fillet edge. You will construct a native solid by extruding a 2D region. Then you will construct a free-form surface and use the surface to cut the solid. After that, you will fillet an edge of the solid. Figure 2.137 shows the extrude solid and the free-form surface.

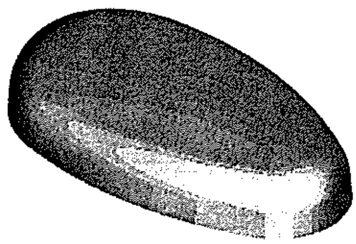


Figure 2.136 Rendered image of the casing of a remote control

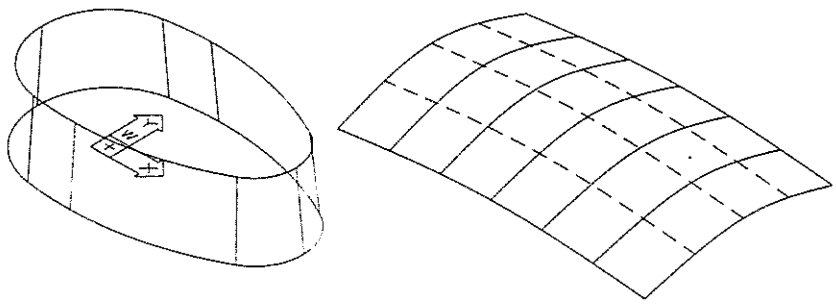


Figure 2.137 Extrude solid and free-form surface

Start a new drawing. Use the template Surf_mdl.dwt. Then use the LAYER command to add a new layer Solid.

<File> **<New...>**

<Assist> **<Format>** **<Layer...>**

Command: **LAYER**

[Layer

Name	Color	Linetype
0	White	Continuous
Solid	Cyan	Continuous
Surface	Red	Continuous
Wire	Green	Continuous

Current Layer: **Wire**

OK]

Now save the current drawing as a template drawing file for use in this project and in the other two solid modeling projects. Then save it as a drawing.

<File> **<Save As...>**

Command: **SAVEAS**

[File name: **Surfsol.dwt**
Save as type: **Drawing Template File (*.dwt)**
Save]

<File> **<Save As...>**

Command: **SAVEAS**

[File name: **Rmote1.dwg**
Save as type: **AutoCAD R14 Drawing (*.dwg)**
Save]

The current working layer is Wire. Construct a set of wires as shown in Figure 2.138. Do not include dimensions and center lines in your drawing. After constructing the wires, use the REGION command to convert the four circular arcs into a region.

Command: **REGION**
Select objects: [**Select A, B, C, and D (Figure 2.138).**]
Select objects: [**Enter]**

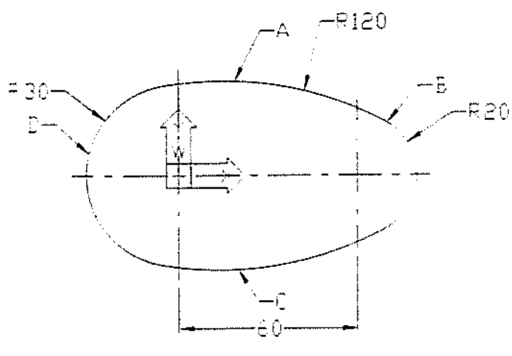


Figure 2.138 Dimensions of the wires

Set the display to an isometric view and set the current layer to Solid. Then select the Extrude item from the Solids cascading menu of the Design pull-down menu to use the EXTRUDE command to extrude the 2D region into an extrude solid. (See Figure 2.139.)

Command: **8**

[Mechanical Main] **[Layer Control]**

Current layer: **Solid**

<Design> **<Solids>** **<Extrude>**

Command: **EXTRUDE**
Select objects: **LAST**
Select objects: **[Enter]**
Path/<Height of Extrusion>: **30**
Extrusion taper angle: **5**

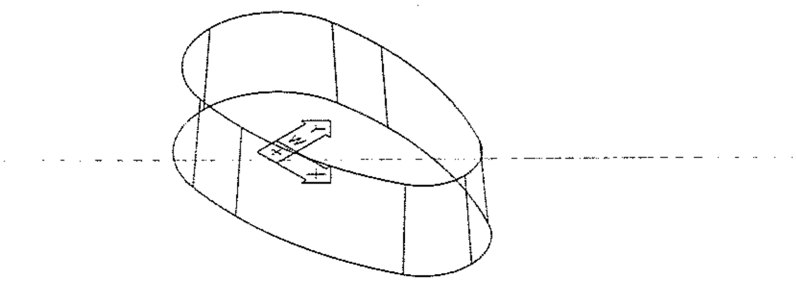


Figure 2.139 Extrude solid constructed

Now you have the solid model of the main body of the remote control. Set the current layer to Wire and construct two splines. They define the cross section and rail of a sweep surface. (See Figure 2.140.)

[Mechanical Main] **[Layer Control]**

Current layer: **Wire**

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-40,-40,5	**-40,0,15**	**-40,40,5**	**[Enter]**	**[Enter]**	**[Enter]**
-40,0,15	**25,0,20**	**90,0,15**	**[Enter]**	**[Enter]**	**[Enter]**

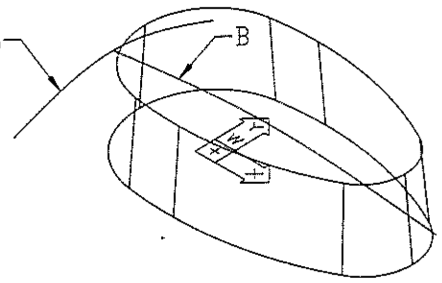


Figure 2.140 Splines constructed

Set the current layer to Surface. Then use the AMSWEEPSF command to construct a sweep surface. (See Figure 2.141.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: **[Select A (Figure 2.140).]**
Select cross sections: **[Enter]**
Select rails: **[Select B (Figure 2.140).]**
Select rails: **[Enter]**

[Normal OK]

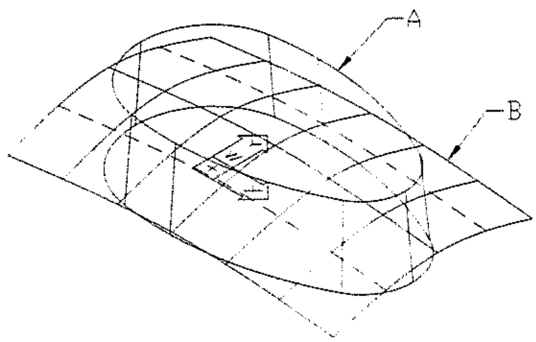


Figure 2.141 Sweep surface constructed

The sweep surface for cutting the solid is complete. Note that all its boundary edges must lie outside the boundary of the solid that is to be cut. Now use the AMSOLCUT command. (See Figure 2.142.) To cut a solid with a surface, you must decide which side of the solid is to be removed. In Figure 2.142, the arrow indicates the portion of the solid that is to be cut away.

<Surface> **<Edit Solid>**

Command: **AMSOLCUT**
Select solid to cut: **[Select A (Figure 2.141).]**
Select surface: **[Select B (Figure 2.141).]**

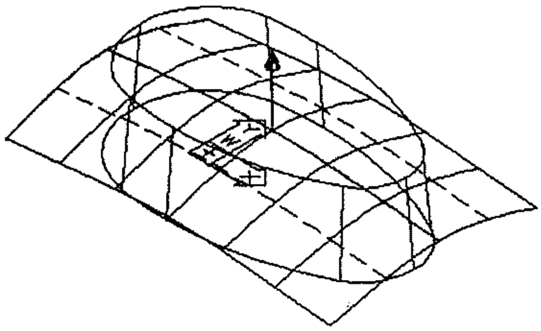


Figure 2.142 Arrow indicating the direction of solid removal

Portion to remove: Flip/<Accept>: **[F** (if the direction arrow is pointing downward) or **A** (if the direction arrow is pointing upward).**]**

After cutting, the wires and the surface are not needed. Set the current layer to Solid and turn off layers Wire and Surface. (See Figure 2.143.)

[Mechanical Main] **[Layer Control]**

Off layers: **Surface, Wire**
Current layer: **Solid**

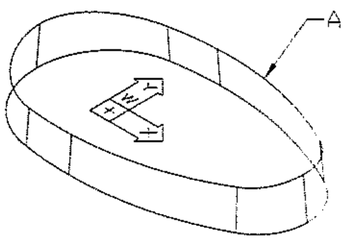


Figure 2.143 Solid cut by the surface

After cutting a solid with a surface, you can add other solid features. Use the FILLET command to round off the upper edge. (See Figure 2.144.)

<Modify> **<Fillet>**

Command: **FILLET**
Polyline/Radius/Trim/<Select first object>: **[Select A (Figure 2.143).]**
Enter radius: **10**
Chain/Radius/<Select edge>: **C**
Edge/Radius/<Select edge chain>: **[Select A (Figure 2.143).]**
Edge/Radius/<Select edge chain>: **[Enter]**

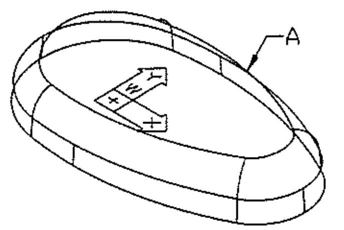


Figure 2.144 Edges filleted

The solid model is complete. Save your drawing.

<File> **<Save>**

As we said earlier, a solid can be converted into a number of surfaces. To see how this is done, use the AM2SF command. (See Figure 2.145.)

<Surface> **<Create Surface>** **<From ACAD>**

Command: **AM2SF**
Face/<Objects>: **O**
Select objects: **[Select A (Figure 2.144).]**
Select objects: **[Enter]**

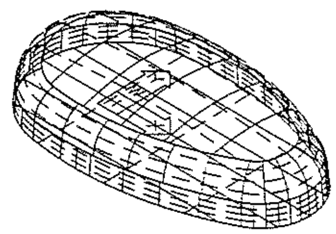


Figure 2.145 Solid converted into a set of surfaces

Save your drawing with another file name.

<File> **<Save As...>**

File name: **Rmote2.dwg**

Now you have two drawing files: a solid model and a surface model. In constructing the model, you learned that a surface can be used to cut a solid, a cut solid can be further modified, and a solid can be converted into a set of surfaces.

Coil Spring

Figure 2.146 shows the rendered image of the solid model of a helical coil spring. To construct this solid model, you will construct a helical surface by sweeping. The cross section of sweeping is circular and the rail of sweeping is generated from an augmented line. Then you will construct a solid box. After that, you will use the surface to cut the solid. Figure 2.147 shows the helical surface and the solid box.

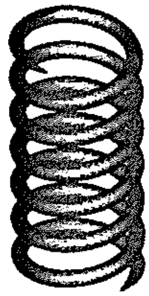


Figure 2.146 Rendered image of the model of a helical coil spring

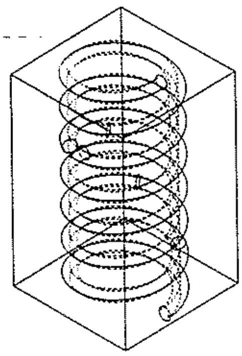


Figure 2.147 Solid and helical sweep surface

As in Figure 2.147, the solid is a simple box. Its base area is slightly larger than the top view of the spring, and its height is equal to the height of the spring. The helical surface is a sweep surface constructed by sweeping a circular cross section along a rail defined by an augmented line. (See Figure 2.148.)

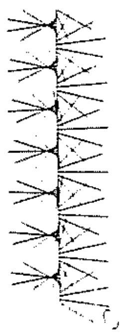


Figure 2.148 Circular cross section and augmented line

Start a new drawing from the template drawing that you saved in the preceding project.

<File> **<New...>**

Command: **NEW**

[Use a Template
Select a Template: **Surfsol.dwt**
OK]

You will construct an augmented line for use as a rail of the sweep surface. To set the default augmented line vector length, use the AMPREFS command.

<Surface> **<Preferences...>**

Command: **AMPREFS**
[Surfaces
Aug Vector Length: **5**
OK]

The free length of the coil spring is 32 units, its pitch is 7 units, its wire diameter is 2 units, and its mean diameter is 14 units. For the boundary edges of the surface to lie outside the solid, which should be 32 units in height, you will construct a helical sweep surface with a total free length of 35 units instead of 32 units.

The current layer is Wire. Use the LINE command to construct a line. Then set the display to an isometric view. (See Figure 2.149.)

<Design> **<Line>**

Command: **LINE**
From point: **0,0,-1.5**
To point: **@0,0,35**
To point: **[Enter]**

Command: **8**

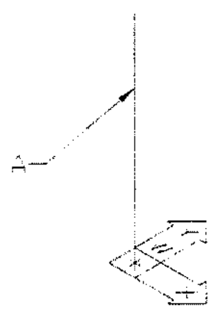


Figure 2.149 Line constructed

For a free length of 35 units, there will be seven coils. Suppose you want to place 12 control points for each coil; you need 84 control points. Use the AMREFINE3D command to refine the line into 84 control points.

<Surface> **<Edit Wireframe>** **<Refine>**

Command: **AMREFINE3D**
Select lines or polylines: **[Select A (Figure 2.149).]**
Select lines or polylines: **[Enter]**
Points/<Tolerance>: **P**
Points: **84**

To convert the line to an augmented line, use the Add option of the AMEDITAUG command. (See Figure 2.150.)

<Surface> **<Edit Wireframe>** **<Augment Polyline>**

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **ADD**
Select lines or polylines: **[Select A (Figure 2.149).]**
Select lines or polylines: **[Enter]**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **[Enter]**

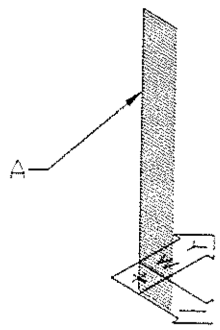


Figure 2.150 Vectors added

Because there are seven coils and each coil rotates 360°, twist the vectors of the augmented line 2520°. (See Figure 2.151.)

<Surface> **<Edit Wireframe>** **< Twist Vectors >**

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **TWIST**
Total angle: **2520**
Range/<All>: **ALL**
Select augmented line: **[Select A (Figure 2.150).]**
Select augmented line: **[Enter]**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **[Enter]**

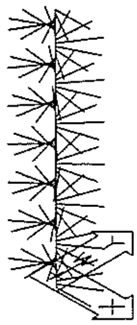


Figure 2.151 Vectors twisted

The rail for the sweep surface is complete. Now rotate the UCS 90° about the X axis and construct a circle. (See Figure 2.152.)

<Assist> **<UCS>** **<X Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

<Design> **<Circle>** **<Center, Radius>**

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **7,0**
Diameter/<Radius>: **1**

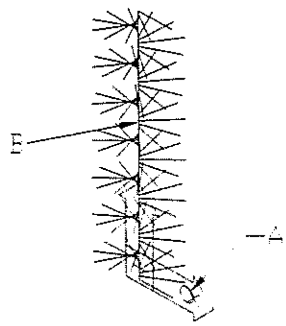


Figure 2.152 UCS rotated, circle constructed

Set the current layer to Surface. Using the circle as a cross section and the augmented line as a rail, use the AMSWEEPSF command to construct a sweep surface. (See Figure 2.153.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Surface**

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: **[Select A (Figure 2.152).]**
Select cross sections: **[Enter]**
Select rails: **[Select B (Figure 2.152).]**
Select rails: **[Enter]**

[Normal OK]

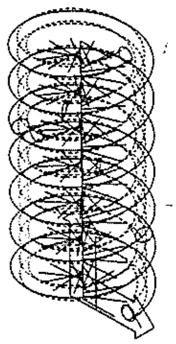


Figure 2.153 Sweep surface constructed

Set the current layer to Solid and turn off layer Wire. Then set the UCS to World and construct a solid box.

[Mechanical Main] **[Layer Control]**

Off layer: **Wire**
Current Layer: **Solid**

<Assist> **<UCS>** **<World>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Design> **<Solids>** **<Box>** **<Corner>**

Command: **BOX**
Center/<Corner of box>: **-10,-10,1**
Cube/Length/<other corner>: **10,10,1**
Height: **32**

Use the shortcut key [6] to set the display to a front view. (See Figure 2.154.)

Command: **6**

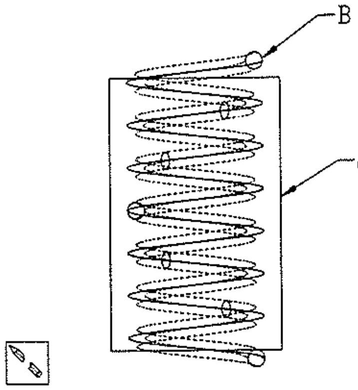


Figure 2.154 Solid box constructed

To cut the solid with the surface, use the AMSOLCUT command. Note the direction of material removal, as shown in Figure 2.155.

<Surface> **<Edit Solid>**

Command: **AMSOLCUT**
Select solid to cut: [**Select A (Figure 2.154).**]
Select surface: [**Select B (Figure 2.154).**]

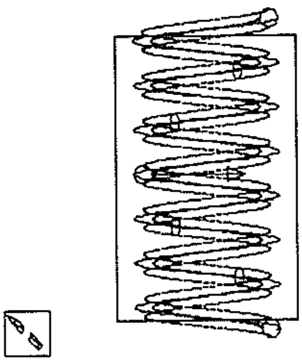


Figure 2.155 Direction of solid removal

Portion to remove: Flip/<Accept>: [**A** (if the arrow is the same as that shown in Figure 2.155.)]

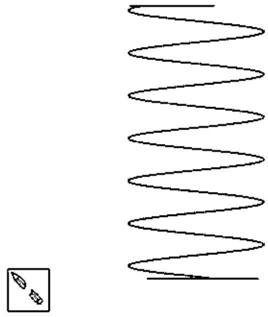


Figure 2.156 Solid cut by surface

To display the silhouette of the solid coil spring, set DISPSILH to 1. Then regenerate the display. (See Figure 2.157.)

Command: **DISPSILH**
New value for DISPSILH: **1**

Command: **REGEN**

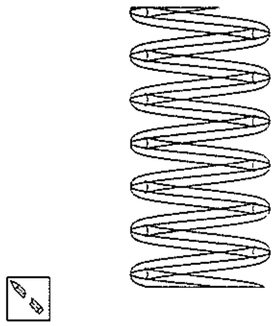


Figure 2.157 Silhouette displayed

The solid model of a coil spring is complete. Save your drawing.

<File> **<Save>**

File name: **Coil.dwg**

In making this model, you learned how to use an augmented line as a rail for sweeping and how to use a surface to cut a solid.

Bevel Gear

Figure 2.158 shows the rendered image of the solid model of a bevel gear. The main body of this solid model is a solid of revolution, and the teeth are constructed by cutting with free-form surfaces. Figure 2.159 shows the solid model.

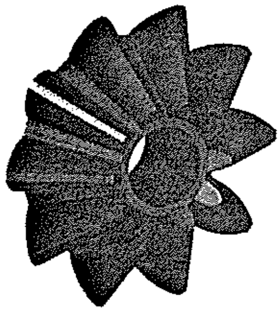


Figure 2.158 Rendered image of the bevel gear

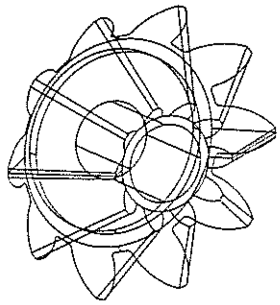


Figure 2.159 Solid model of the bevel gear

To make the gear teeth, you first make a rule surface to define the profile of a tooth. Then you array the rule surface about the axis of the solid. Finally, you cut the solid with the surfaces. Figure 2.160 shows the surfaces and the solid of revolution.

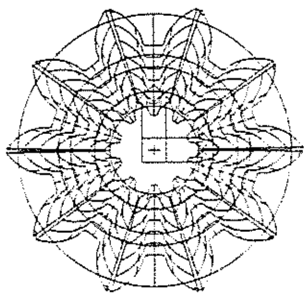


Figure 2.160 Rule surfaces and solid of revolution

Start a new drawing. Use the template Surf_mdl.dwt.

<File> <New...>

Using the dimensions shown in Figure 2.161, construct a set of wires on layer Wire. Do not include the dimensions in your drawing. While making the wires, note the position of the origin. Your drawing should resemble Figure 2.162.

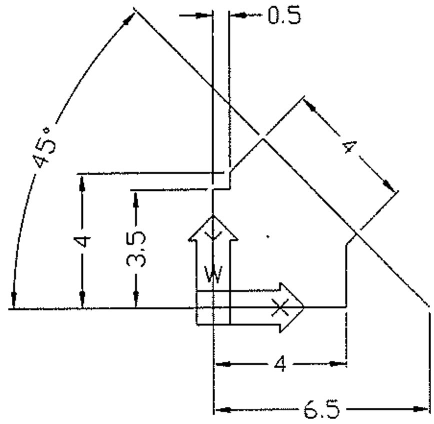


Figure 2.161 Dimensions for a set of wires

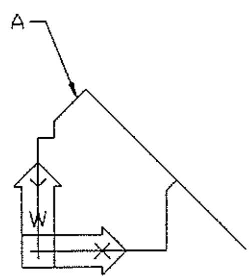


Figure 2.162 Wires constructed

Lengthen line A (Figure 2.162) by 1 unit. (See Figure 2.163.)

<Modify> **<Lengthen>**

Command: **LENGTHEN**
DElta/Percent/Total/DYnamic/<Select object>: **DE**
Angle/<Enter delta length>: **1**
<Select object to change>/Undo: **[Select A (Figure 2.162).]**
<Select object to change>/Undo: **[Enter]**

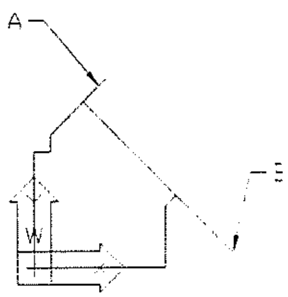


Figure 2.163 Line A lengthened

Now construct a line. (See Figure 2.164.)

<Design> **<Line>**

Command: **LINE**
From point: **END** of **[Select A (Figure 2.163).]**
To point: **END** of **[Select B (Figure 2.163).]**
To point: **[Enter]**

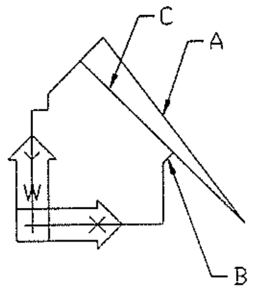


Figure 2.164 Line drawn

To complete the set of wires, fillet a corner with zero radius and erase a line. (See Figure 2.165.)

<Modify> **<Fillet>**

Command: **FILLET**
Polyline/Radius/Trim/<Select first object>: **R**
Enter fillet radius: **0**

Command: **[Enter]**
FILLET
Polyline/Radius/Trim/<Select first object>: **[Select A (Figure 2.164).]**
Select second object: **[Select B (Figure 2.164).]**

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select C (Figure 2.164).]**
Select objects: **[Enter]**

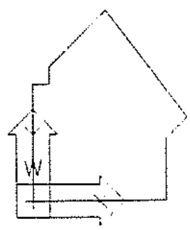


Figure 2.165 Corner edited, line erased

The set of wires is complete. Use the REGION command to convert them into a region. Then set layer Solid as the current layer. After that, use the REVOLVE command to revolve the region about its lower edge to a solid. (See Figure 2.166.)

Command: **REGION**
Select objects: **[Select all the wires (Figure 2.165).]**
Select objects: **[Enter]**

[Mechanical Main] **[Layer Control]**

Current layer: **Solid**

<Design> **<Solids>** **<Revolve>**

Command: **REVOLVE**
Select objects: **LAST**
Select objects: **[Enter]**
Axis of revolution - Object/X/Y/<Start point of axis>: **X**
Angle of revolution <full circle>: **[Enter]**

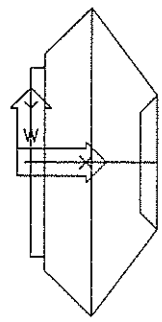


Figure 2.166 Region formed, solid revolved

Change the UCS origin to a new position. (See Figure 2.167.)

<Assist> **<UCS>** **<Origin>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **O**
Origin point: **-4,0**

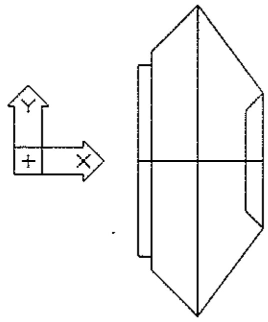


Figure 2.167 UCS origin changed

Change the display to a left isometric view. Then change the UCS orientation again. (See Figure 2.168.)

<View> **<Model Views>** **<Left Isometric>**

<Assist> **<UCS>** **<Z Axis Vector>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **ZAXIS**
Origin point: **0,0,0**
Point on positive portion of Z-axis: **-1,0,1**

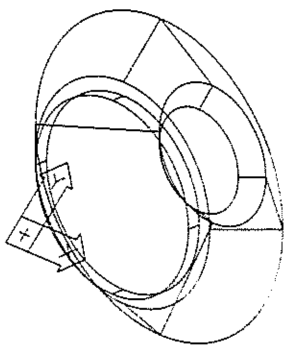


Figure 2.168 Display changed to left isometric, UCS changed

Set the current layer to Wire and turn off layer Solid. Then use the shortcut key [9] to set the display to the plan view of the current UCS. After that, construct a set of wires. (See Figure 2.169.)

[Mechanical Main] **[Layer Control]**

Off layer: **Solid**
Current layer: **Wire**

Command: **9**

<Design> <Line>

Command: **LINE**
From point: **0,0**
To point: **@6.2<86**
To point: **[Enter]**

<Design> <Arc> <3 Points>

Command: **ARC**
Center/<Start point>: **[Enter]**
End point: **@1.4,2.2**

<Design> <Line>

Command: **LINE**
From point: **0,0**
To point: **@6.2<94**
To point: **[Enter]**

<Design> <Arc> <3 Points>

Command: **ARC**
Center/<Start point>: **[Enter]**
End point: **@-1.4,2.2**

<Design> <Circle> <Center, Radius>

Command: **CIRCLE**
3P/2P/TTR/<Center point>: **0,0**
Diameter/<Radius>: **6.2**

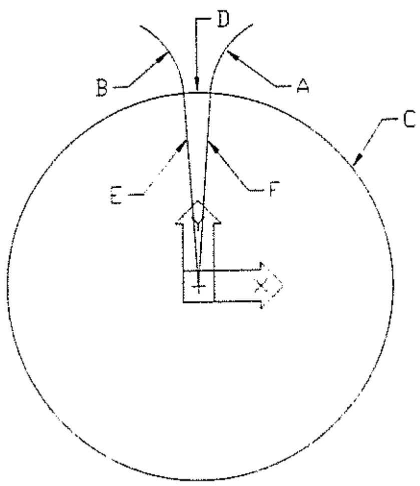


Figure 2.169 Layer Solid turned off, wires constructed on layer Wire

Edit the wires. (See Figure 2.170.)

<Modify> <Trim>

Command: **TRIM**
Select objects: **[Select A and B (Figure 2.169).]**
Select objects: **[Enter]**
<Select object to trim>/Project/Edge/Undo: **[Select C (Figure 2.169).]**
<Select object to trim>/Project/Edge/Undo: **[Enter]**

<Surface> <Edit Wireframe> <Fillet>

Command: **AMFILLET3D**
Radius/Trim/<Select first object>: **R**
Radius: **0.2**
Radius/Trim/<Select first object>: **[Select A (Figure 2.169).]**
Radius/Trim/<Select second object>: **[Select D (Figure 2.169).]**

Command: **[Enter]**
AMFILLET3D
Radius/Trim/<Select first object>: **[Select B (Figure 2.169).]**
Radius/Trim/<Select second object>: **[Select D (Figure 2.169).]**

<Modify> <Erase>

Command: **ERASE**
Select objects: **[Select E and F (Figure 2.169).]**
Select objects: **[Enter]**

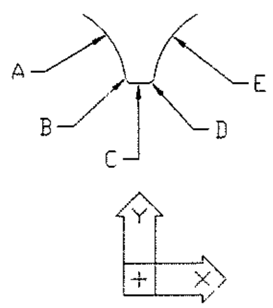


Figure 2.170 Wires edited

Join the wires so that they become a polyline. Then make a copy and scale down the copied polyline. (See Figure 2.171.)

<Surface> <Edit Wireframe> <Join...>

Command: **AMJOIN3D**

[Join3D
Mode: **Automatic**
Output: **Polyline**
OK]

Select start wire: **[Select A (Figure 2.170).]**

Select wires to join: **[Select B, C, D, and E (Figure 2.170).]**
Select wires to join: **[Enter]**
Reverse? Yes/<No>: **[Enter]**

<Construct> <Copy>

Command: **COPY**
Select objects: **[Select A (Figure 2.170).]**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0**
Second point of displacement: **[Enter]**

<Modify> <Scale>

Command: **SCALE**
Select objects: **[Select A (Figure 2.170).]**
Select objects: **[Enter]**
Base point: **0,7.5**
<Scale factor>/Reference: **1/3**

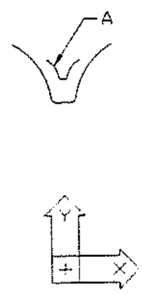


Figure 2.171 Wires joined and copied, copied wire scaled

Move the scaled wire. Then set the display to a left isometric view. (See Figure 2.172.)

<Construct> <Move>

Command: **MOVE**
Select objects: **[Select A (Figure 2.171).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,-5**
Second point of displacement: **[Enter]**

<View> <Model Views> <Left Isometric>

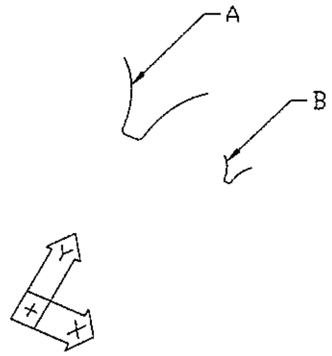


Figure 2.172 Scaled wire moved, display set

Set the current layer to Surface. Then construct a rule surface. (See Figure 2.173.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Rule>**

Command: **AMRULE**
Select first wire: **[Select A (Figure 2.172).]**
Select second wire: **[Select B (Figure 2.172).]**

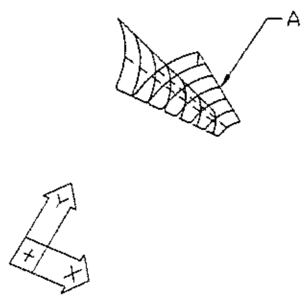


Figure 2.173 Rule surface constructed

Set the UCS to a new position. Then array the rule surface. (See Figure 2.174.)

<Assist> **<UCS>** **<World>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Assist> **<UCS>** **<Z Axis Vector>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **ZAXIS**
Origin point <0,0,0>: **[Enter]**

Point on positive portion of Z-axis: **-1,0**

<Construct> <Array> <Polar>

Command: **ARRAY**
Select objects: **[Select A (Figure 2.173).]**
Select objects: **[Enter]**
Rectangular or Polar array (<R>/P): **P**
Base/<Specify center point of array>: **0,0**
Number of items: **10**
Angle to fill (+=ccw, -=cw) <360>: **[Enter]**
Rotate objects as they are copied? <Y> **[Enter]**

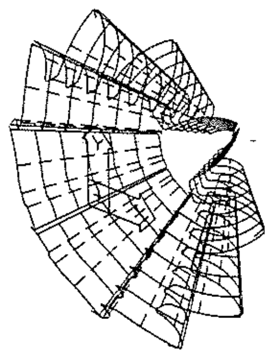


Figure 2.174 UCS changed, rule surface arrayed

Turn on layer Solid, make it the current layer, and turn off layer Wire. Then use the shortcut key [9] to set the display to a plan view. (See Figure 2.175.)

[Mechanical Main] [Layer Control]

Off layer: **Wire**
Current layer: **Solid**

Command: **9**

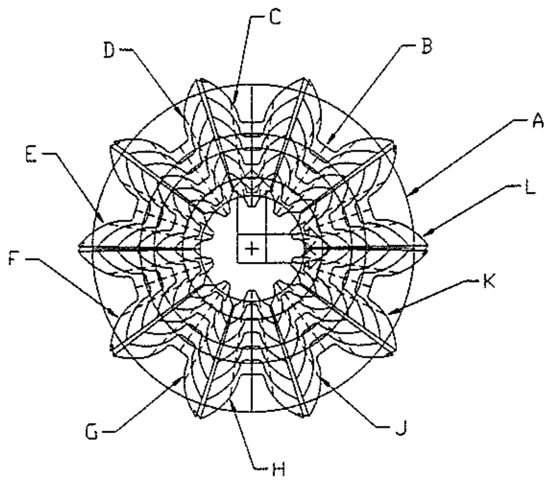


Figure 2.175 Layer Wire turned off, layer Solid turned on, and display set

Now use the AMSOLCUT command to cut the gear teeth. (See Figure 2.176.)

<Surface> **<Edit Solid>**

Command: **AMSOLCUT**
Select solid to cut: **[Select A (Figure 2.175).]**
Select surface: **[Select B (Figure 2.175).]**

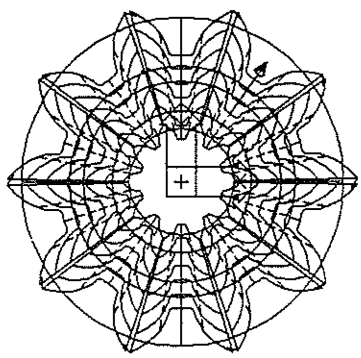


Figure 2.176 Solid removal direction

Portion to remove: Flip/<Accept>: [**A** (if the arrow direction is the same as that shown in Figure 2.176)]

Repeat the AMSOLCUT command nine more times to use surfaces C, D, E, F, G, H, J, K, and L (Figure 2.175) to cut solid A (Figure 2.175). (See Figure 2.177.)

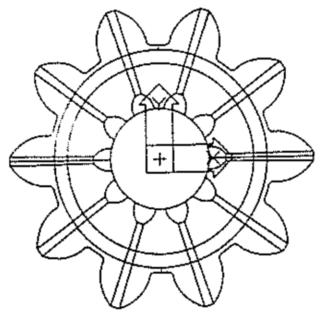


Figure 2.177 Gear teeth cut

Now set the display to a left isometric view. (See Figure 2.178.)

<View> **<Model Views>** **<Left Isometric>**

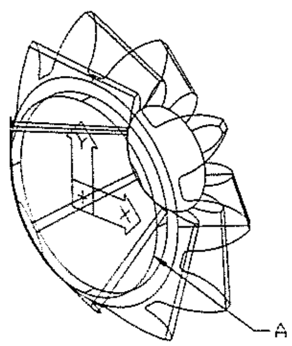


Figure 2.178 Display set to left isometric view

To complete the model, you will cut a hole in the bevel gear. Construct a cylinder. (See Figure 2.179.)

<Design> **<Solids>** **<Cylinder>** **<Center>**

Command: **CYLINDER**
Elliptical/<center point>: **CEN** of **[Select A (Figure 2.178).]**
Diameter/<Radius>: **1.5**
Center of other end/<Height>: **-6**

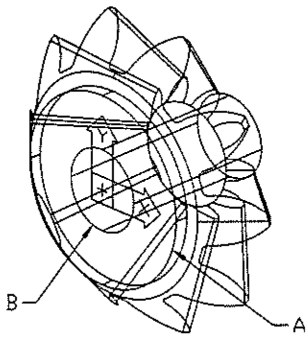


Figure 2.179 Cylinder constructed

To use the cylinder to cut a hole, run the SUBTRACT command. (See Figure 2.180.)

<Modify> <Boolean> <Subtract>

Command: **SUBTRACT**
Select solids and regions to subtract from...
Select objects: **[Select A (Figure 2.179).]**
Select objects: **[Enter]**
Select solids and regions to subtract...
Select objects: **[Select B (Figure 2.179).]**
Select objects: **[Enter]**

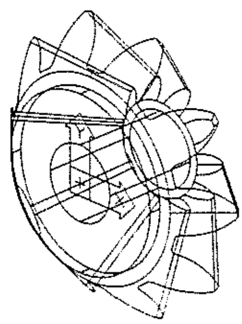


Figure 2.180 Cylinder subtracted from the solid model

The bevel gear is complete. Save your drawing.

<File> <Save>

Command: **Bevel.dwg**

In making this model, you have learned how to use a set of surfaces to cut a solid. Thus, you should realize that you can use a number of surfaces to cut a solid. To reiterate, a surface must have all its boundary edges outside the solid that is to be cut.

2.10 Scale Model Car Project 1

Now that you have learned various surface modeling tools and utilities, you will work on a surface modeling project, a scale model car. The main focus of this project is on real-world engineering applications. A car body has been chosen here because it has many free-form surfaces. To construct them in the computer, you must be able to perceive and identify the kinds of surfaces and to visualize the defining wires of the surfaces. By working on this project, you will further enhance your understanding of surface modeling.

Figure 2.181 shows the rendered image of the body of a 1/10 scale radio-controlled model car that you are going to construct. Take some time to study this rendered image and think about what 3D surfaces must be constructed, what 3D wires are required to generate the 3D surfaces, and how the 3D surfaces are put together.

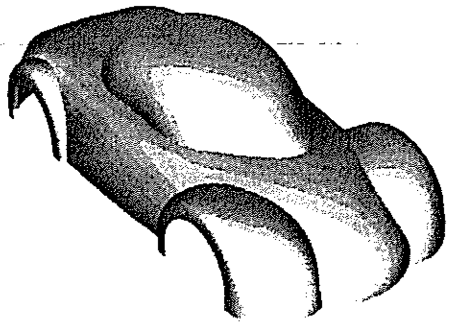


Figure 2.181 Rendered image of the body of a 1/10 scale radio-controlled model car

Analysis

The car body has six surfaces: main body, left and right front fenders, left and right rear fenders, and greenhouse. Because the model is symmetrical about its longitudinal axis, you need to make only four surfaces — main body, left front fender, left rear fender, and greenhouse — and to mirror the left front and rear fenders to produce the right front and rear fenders. (See Figure 2.182.)

Figure 2.182 Surface model of the body of a 1/10 scale radio-controlled model car

For you to see the surfaces of the car more clearly, they are exploded apart and shown in Figure 2.183.

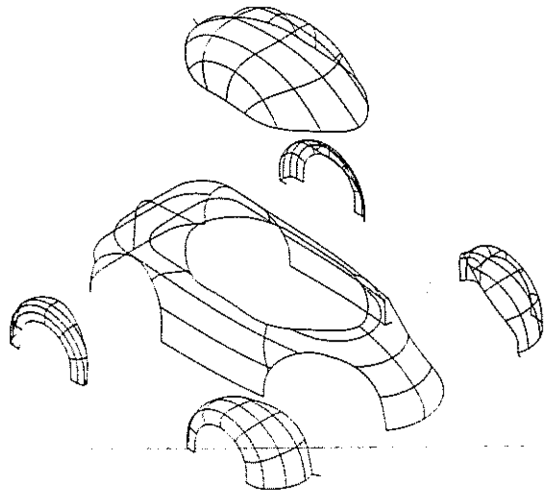


Figure 2.183 Exploded view of the 3D surfaces

As shown in Figure 2.183, the surfaces of the model are trimmed surfaces with irregular edges. To make them, you have to construct smooth base surfaces with smooth defining wires. The base surfaces are shown in Figure 2.184.

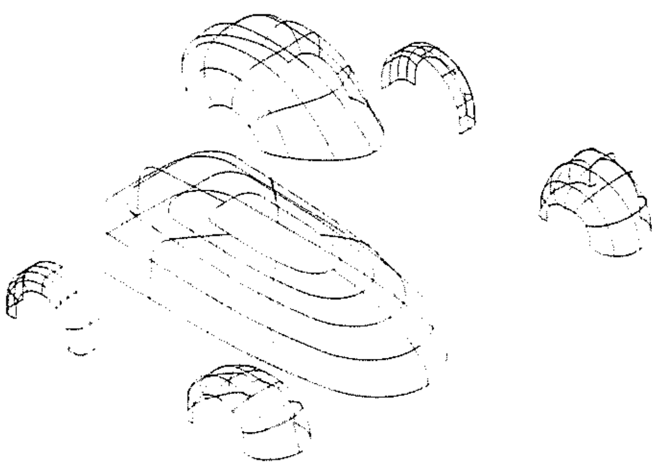


Figure 2.184 Exploded view of the untrimmed base surfaces

Figures 2.185 through 2.188 show the trimmed surfaces, base surfaces, and wires of the four major parts of the model.

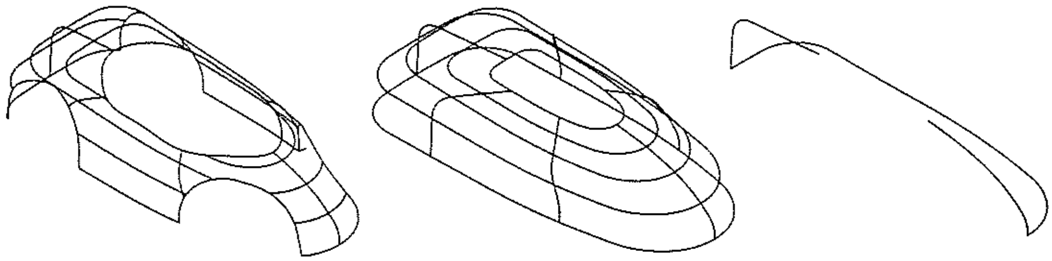


Figure 2.185 Trimmed surface, base surface, and wires of the main body

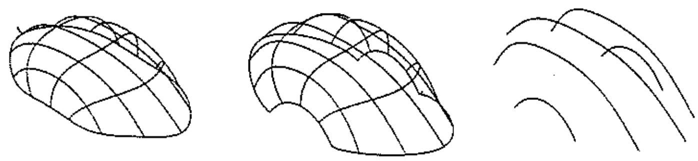


Figure 2.186 Trimmed surface, base surface, and wires of the greenhouse

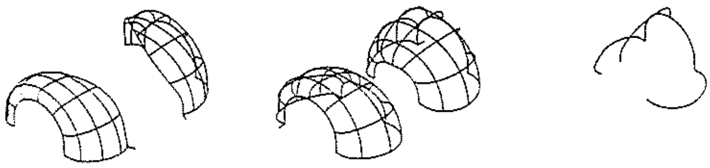


Figure 2.187 Trimmed surfaces, base surfaces, and wires of the front fenders

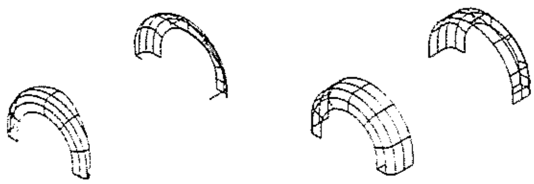


Figure 2.188 Trimmed surfaces, base surfaces, and wires of the rear fenders

Drawing Setup

Start a new drawing. Use the template Surf_mdl.dwt. Then use the AMPREFS command to set preferences. From the Surfaces tab of the Desktop Preferences dialog box, select the [Model Size...] button to bring up the Approximate Model Size dialog box. Then set the model size to 3000 millimeters and select the [OK] button. On returning to the Surfaces tab of the Desktop Preferences dialog box, set Vector length to 10 units and then select the [OK] button.

<File> **<New...>**

<Surface> **<Preferences...>**

Use the LAYER command to construct four additional layers: Body, Front, Rear, and Top. Then set layer Body as the current layer.

<Assist> **<Format>** **<Layer...>**

Drawing setup is complete.

Main Body

Figure 2.189 shows the top view of the main body of the model car. Basically, you can use the silhouette of this view as a cross section and sweep it along two rails, one representing the longitudinal cross section at the front and the other representing the longitudinal cross section at the rear. However, a closed loop is not allowed in multiple-rails sweeping. Because the model is symmetrical about its central plane, this silhouette is broken into two and a sweep surface will be constructed to represent half of the surface of the body. To complete the main body, you will mirror the sweep surface and join it to the new one.

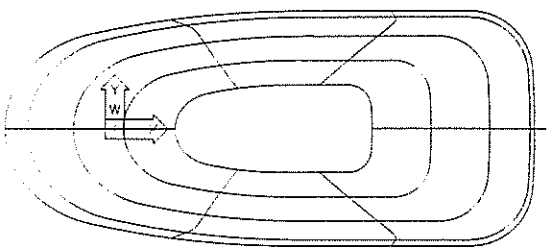


Figure 2.189 Top view of the main body

As we have said, to produce smooth surfaces, you need to use smooth splines. There are three ways to construct smooth splines: use the SPLINE command and input a number of points to define the spline, use the AMJOIN3D command to join a number of line and arc segments to form a spline, and use the AMFITSPLINE command to fit a wire into a spline. Using use the SPLINE command is the most direct way to make a spline. However, the input points are difficult to define and the profile of the output spline is hard to control. Using the AMJOIN3D command or the AMFITSPLINE command, you can better control the form and shape of the spline by constructing lines, arcs, polylines, or 3D polylines.

Now construct a number of line and arc segments, then join them into a spline to define the cross section of a sweep surface.

Using the dimensions shown in Figure 2.190, construct a series of arc and line segments. Do not include the dimensions in your drawing.

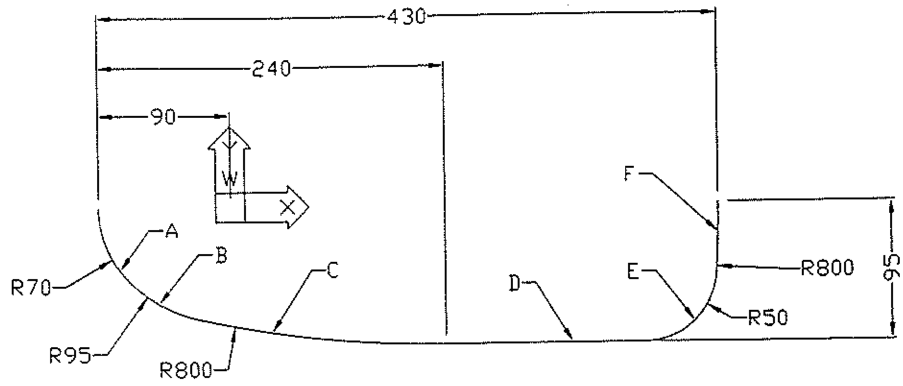


Figure 2.190 Cross section wires to be constructed on the XY plane

To construct a spline from the line and arc segments, use the AMJOIN3D command. After you select the wires, a small square and a triangle appear in your display. They indicate the start point and the direction of the spline, respectively. Accept the default.

<Surface> **<Edit Wireframe>** **<Join...>**

Command: **AMJOIN3D**

[Join3D
Mode: **Automatic**
Output: **Spline**
OK]

Select start wire: **[Select A (Figure 2.190).]**
Select wires to join: **[Select B, C, D, E, and F (Figure 2.190).]**
Select wires to join: **[Enter]**
Reverse? Yes/<No>: **[Enter]**

Because the default setting of the DELOBJ variable is 1, the line and arc segments are erased. Now you have a spline for defining the cross section of the sweep surface. Use the shortcut key [8] to set the display to an isometric view. Then use the SPLINE command to construct a spline to be used as one of the sweeping rails. (See Figure 2.191.)

Command: **8**

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-90,0,0	**-90,0,12**	**-76,0,32**	**50,0,77**	**[Enter]**	**[Enter]**	**[Enter]**

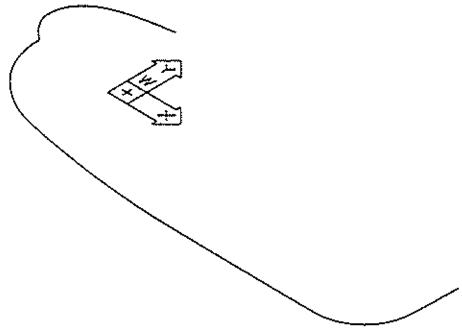


Figure 2.191 Display set to isometric view, spline constructed

Use the LINE command to construct two line segments. (See Figure 2.192.)

<Design> **<Line>**

Command: **LINE**
From point: **210,0,82**
To point: **@200,0,-15**
To point: **[Enter]**

Command: **[Enter]**
LINE
From point: **340,0,0**
To point: **@-15,0,200**
To point: **[Enter]**

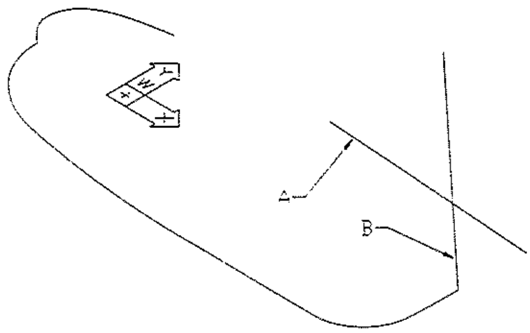


Figure 2.192 Two lines constructed

Form a fillet corner between lines A and B (Figure 2.192). To construct the fillet, you can use either the FILLET or the AMFILLET3D command. AMFILLET3D is a Mechanical Desktop command that enables you to construct a fillet regardless of the current UCS. Use the AMFILLET3D command. (See Figure 2.193.)

<Surface> **<Edit Wireframe>** **<Fillet>**

Command: **AMFILLET3D**
Radius/Trim/<Select first object>: **R**

Radius: **30**
Radius/Trim/<Select first object>: **[Select A (Figure 2.192).]**
Radius/Trim/<Select second object>: **[Select B (Figure 2.192).]**

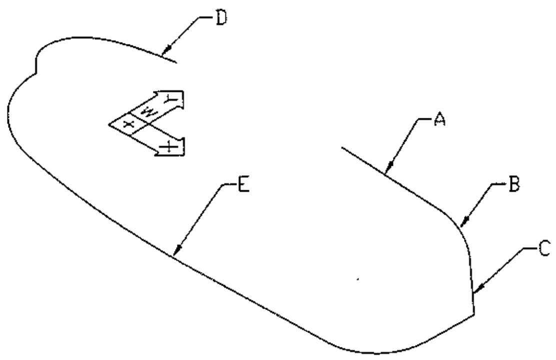


Figure 2.193 3D fillet constructed

Now use the AMJOIN3D command to join wires A, B, and C (Figure 2.193) to form a spline to define the second rail for the sweep surface. There will be no noticeable change on your screen. After joining, set the current layer to Surface. Then construct a sweep surface. (See Figure 2.194.)

<Surface> <Edit Wireframe> <Join...>

Command: **AMJOIN3D**

[Join3D
Automatic **Spline**
OK]

Select start wire: **[Select A (Figure 2.193).]**
Select wires to join: **[Select B and C (Figure 2.193).]**
Select wires to join: **[Enter]**
Reverse? Yes/<No>: **[Enter]**

[Mechanical Main] [Layer Control]

Current layer: **Surface**

<Surface> <Create Surface> <Sweep>

Command: **AMSWEEPSF**
Select cross sections: **[Select E (Figure 2.193).]**
Select cross sections: **[Enter]**
Select rails: **[Select D (Figure 2.193).]**
Select rails: **[Select A (Figure 2.193).]**

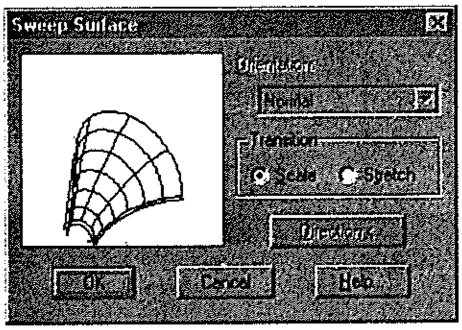


Figure 2.194 Sweep Surface dialog box

In the Sweep Surface dialog box (Figure 2.194), there are two transition methods for a two-rail sweep surface. As the cross section develops along the rails, you can choose to scale the cross section uniformly in all directions or to stretch it in a direction across the rails.

To see the difference between scale sweeping and stretch sweeping, try stretch sweeping first (see Figure 2.195), then undo the surface, and finally perform scale sweeping.

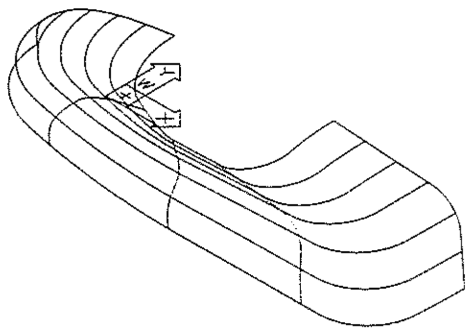


Figure 2.195 Stretch sweeping

Undo the AMSWEEPSF command with the stretch option and use the AMSWEEPSF command with the scale option. (See Figure 2.196.)

<Edit> **<Undo>**

<Surface> **<Create Surface>** **<Sweep>**

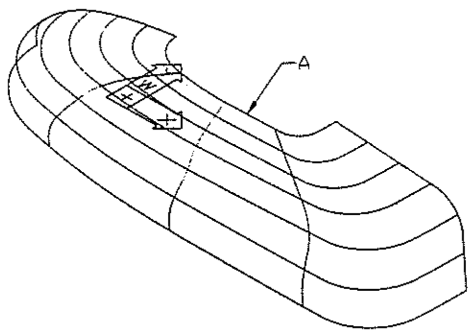


Figure 2.196 Scale sweeping

Compare Figure 2.195 and Figure 2.196 to see the difference between stretch sweeping and scale sweeping. The scale option scales the wire uniformly as a function of the distance between the two rails. The stretch option stretches the wire in a direction defined by the two rails. Here you will use the scale option (Figure 2.196) for the required surface.

Half of the surface for the main body is complete. Turn off layer Body. Then use the MIRROR3D command to construct the other half of the surface. (See Figure 2.197.)

[Mechanical Main] **[Layer Control]**

Off layer: **Body**
Current layer: **Surface**

<Construct> **<3D Translations>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select A (Figure 2.196).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **ZX**
Point on ZX plane: **0,0,0**
Delete old objects? <N> **[Enter]**

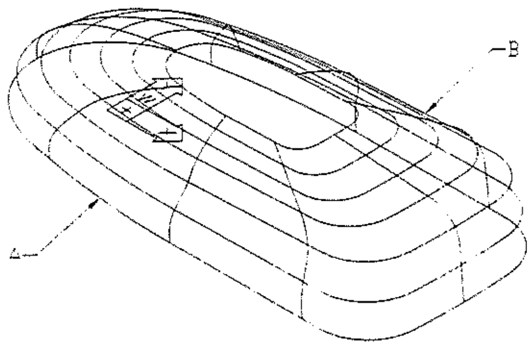


Figure 2.197 Sweep surface mirrored

To complete the main body of the model car, use the AMJOINSF command to join the sweep surfaces together. (See Figure 2.198.) Note that surfaces can be joined only at their untrimmed edges.

<Surface> <Edit Surface> <Join>

Command: **AMJOINSF**
Select surfaces to join: **[Select A and B (Figure 2.197).]**
Select surfaces to join: **[Enter]**

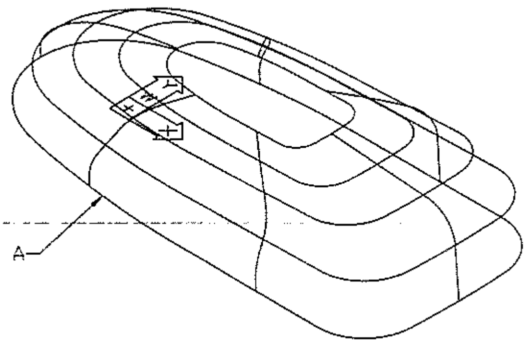


Figure 2.198 Sweep surfaces joined together

To smoothen the surface, you can refine it by using the AMREFINESF command. Before using this command, use the AMDISPSF command to display the control points of the surface. (See Figure 2.199, the Individual Surface Display dialog box.) Select the Show Control Points box and then the [OK] button to see the control points. (See Figure 2.200.)

<Surface> <Surface Display...>

Command: **AMDISPSF**
Select surfaces: **[Select A (Figure 2.198).]**
Select surfaces: **[Enter]**

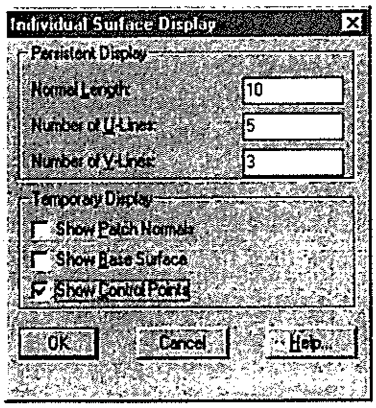


Figure 2.199 Individual Surface Display dialog box

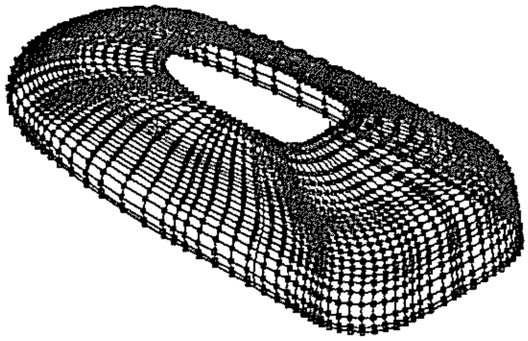


Figure 2.200 Control points displayed

Refresh the display by using the shortcut key [R]. Then use the AMREFINESF command to refine the surface. There are two ways to refine a surface: You can adjust the number of u and v patches or adjust the system tolerance. Here, set the tolerance to 0.1. To display the control points again, use the AMDISPSF command. (See Figure 2.201.)

Command: **R**

<Surface> **<Edit Surface>** **<Refine>**

Command: **AMREFINESF**
Select surfaces: [**Select the surface.**]
Select surfaces: [**Enter**]
Uv patches/<Tolerance>: **T**
Tolerance: **0.1**

<Surface> **<Surface Display...>**

Command: **AMDISPSF**
Select surfaces: [**Select the surface.**]
Select surfaces: [**Enter**]

[**Show Control Points**
OK]

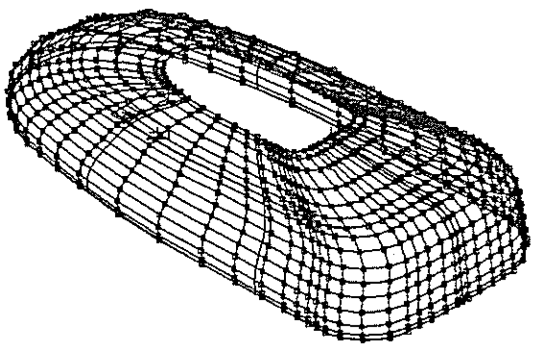


Figure 2.201 Control points refined

Compare Figure 2.200 with Figure 2.201. Note the difference between the sets of control points. The base surface of the main body of the model car is complete. Now you will work on the front fenders.

Front Fenders

There are two front fenders. They are sweep surfaces. They are alike and symmetrical about the central plane of the model car. Set layer Front as the current layer. Then use the UCS command to rotate the UCS 90° about the X axis. After that, construct a polyline. (See Figure 2.202.)

[Mechanical Main] **[Layer Control]**

Current layer: **Front**

<Assist> **<UCS>** **<X Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

<Design> **<Polyline>**

Command: **PLINE**
From point: **38,0,100**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<90**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **A**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **@76<180**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **L**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<270**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **[Enter]**

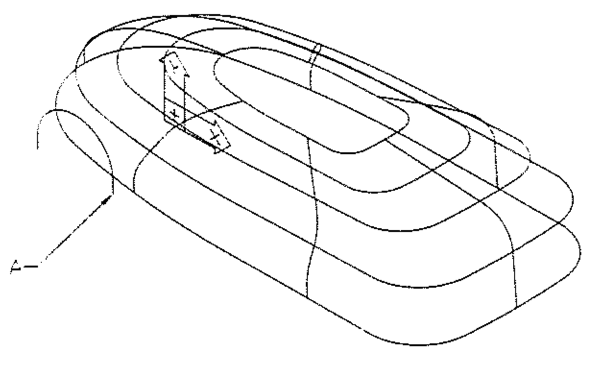


Figure 2.202 UCS rotated, polyline constructed

As we have explained, there are three ways to make a spline. Here, you will use the AMFITSPLINE command to fit the polyline to a spline. (See Figure 2.203, the Fit Spline dialog box.) Accept the default setting and select the [OK] button to exit. Because the system variable DELOBJ is set to 1, the original polyline that is used to fit the spline is deleted. Now you have a spline instead of a polyline.

<Surface> **<Edit Wireframe>** **<Spline Fit...>**

Command: **AMFITSPLINE**
Select wires: **[Select A (Figure 2.202).]**
Select wires: **[Enter]**

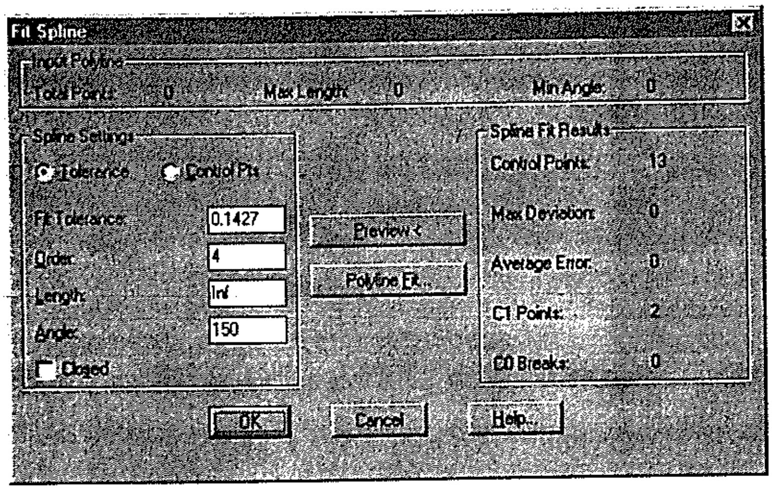


Figure 2.203 Fit Spline dialog box

Now use the SPLINE command to construct three splines. (See Figure 2.204.)

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-38,0,100	**-48,0,100**	**-75,0,75**	**-75,0,40**	**-55,0,25**	**-38,0,25**	**[Enter]**	**[Enter]**	**[Enter]**
0,50,100	**0,60,100**	**0,65,75**	**0,55,40**	**0,40,25**	**0,30,25**	**[Enter]**	**[Enter]**	**[Enter]**
38,0,100	**48,0,100**	**60,0,75**	**50,0,40**	**38,0,25**	**28,0,25**	**[Enter]**	**[Enter]**	**[Enter]**

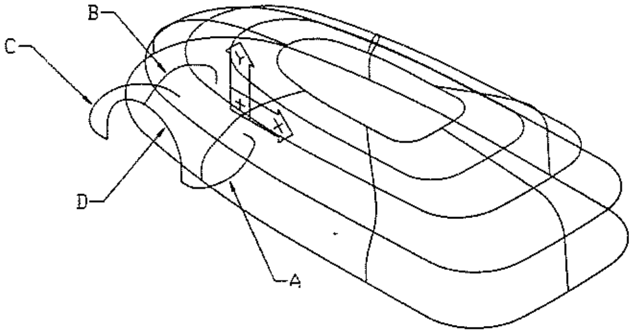


Figure 2.204 Three splines constructed

The wires for the sweep surface are complete. Set the current layer to Surface. Then use the AMSWEEPSF command to construct a sweep surface from three cross sections and one rail. (See Figure 2.205.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Sweep>**

Command: **AMSWEEPSF**
Select cross sections: [**Select A, B, and C (Figure 2.204).**]
Select cross sections: [**Enter**]
Select rails: [**Select D (Figure 2.204).**]
Select rails: [**Enter**]

[Orientation **Normal**
OK]

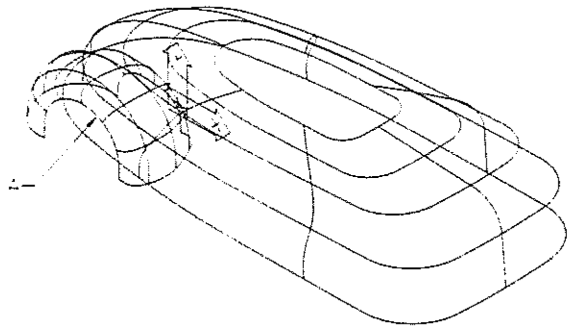


Figure 2.205 Front fender constructed

The base surface for one of the front fenders is complete. Turn off layer Front. Then use the MIRROR3D command to construct a mirror copy of the front fender. (See Figure 2.206.)

[Mechanical Main] **[Layer Control]**

Off layer: **Front**
Current layer: **Surface**

<Construct> **<3D Translations>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select A (Figure 2.205).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **XY**
Point on ZX plane: **0,0,0**
Delete old objects? <N> **[Enter]**

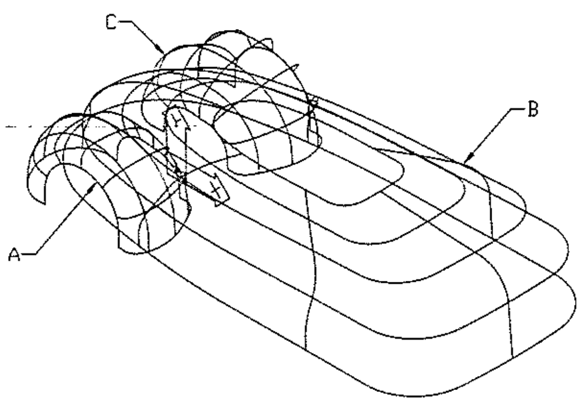


Figure 2.206 Front fender mirrored

To form intersections between the front fender surfaces and the body surface, use the AMINTERSF command. (See Figure 2.207, the Surface Intersection dialog box.) You can output a polyline at the intersection of the surfaces or trim one or both of the intersecting surfaces. Here, trim both surfaces. After trimming one side of the car, repeat the AMINTERSF command to intersect the other front fender with the body surface. (See Figure 2.208.) The front fenders are complete.

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.206).]**
Select second surface: **[Select B (Figure 2.206).]**

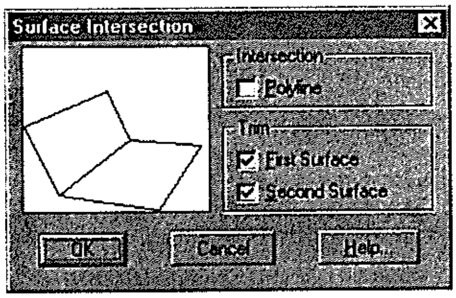


Figure 2.207 Surface Intersection dialog box

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**
Select first surface: [**Select B (Figure 2.206).**]
Select second surface: [**Select C (Figure 2.206).**]

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

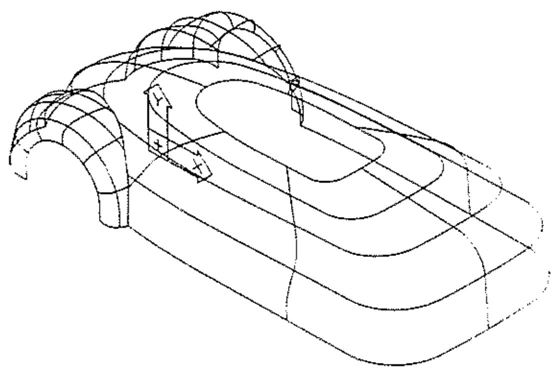


Figure 2.208 Front fenders intersected with the main body

Rear Fenders

Like the front fenders, the two rear fenders are alike and symmetrical. They are constructed from surfaces of revolution and extrusion. You will construct a polyline to define the cross section, rotate it to form a revolve surface, and extrude it to form an extrude surface. Then you will mirror the extrude surface and join the surfaces together. To make the other fender, you will make a mirror copy.

Set the current layer to Rear and set the UCS to World. Then use the PLINE command to construct a polyline. (See Figure 2.209.)

[Mechanical Main] **[Layer Control]**

Current layer: **Rear**

<Assist> <UCS> <World>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Design> <Polyline>

Command: **PLINE**
From point: **222,-100,12**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@10<180**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **A**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **@-10,10**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **L**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@20<90**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **[Enter]**

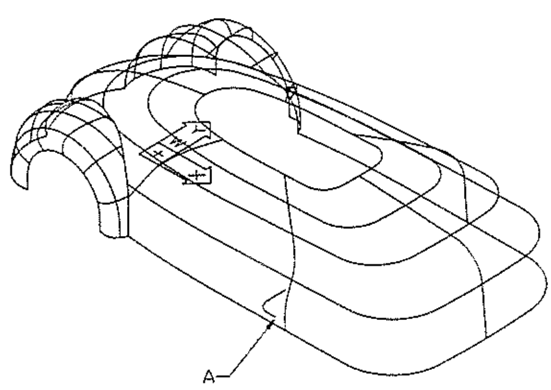


Figure 2.209 UCS set to World, polyline constructed

Set the current layer to Surface. Then use the AMREVOLVESF command to construct a surface of revolution. After that, use the AMEXTRUDESF command to construct a surface of extrusion. (See Figure 2.210.)

[Mechanical Main] [Layer Control]

Current layer: **Surface**

<Surface> <Create Surface> <Revolve>

Command: **AMREVOLVESF**
Select path curves: **[Select A (Figure 2.209).]**
Select path curves: **[Enter]**
Axis of revolution: Wire/<Start point of axis>: **FROM**
Base point: **END** of **[Select A (Figure 2.209).]**
<Offset>: **@38<0**
End point of axis: **@1<90**
Start angle: **0**
Included angle (+=ccw, -=cw) <Full circle>: **180**

<Surface> <Create Surface> <Extrude>

Command: **AMEXTRUDESF**
Select wires: **[Select A (Figure 2.209).]**
Select wires: **[Enter]**
Direction: Viewdir/Wire/X/Y/Z/<Start point>: **Z**
Distance: **-12**
Flip/<Accept>: **[Accept if the arrow is pointing downward.]**
Taper angle: **0**

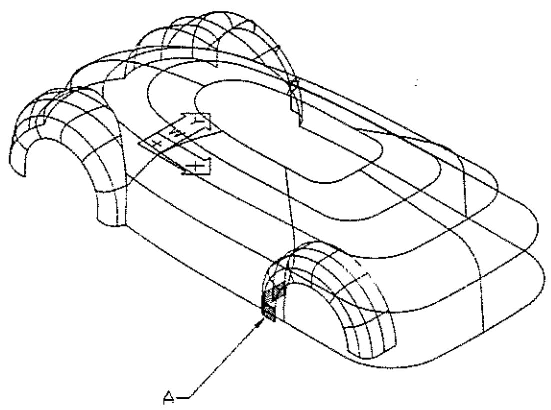


Figure 2.210 Revolve and extrude surfaces constructed

Make a mirror copy of the extrude surface. (See Figure 2.211.)

<Construct> <3D Translations> <3D Mirror>

Command: **MIRROR3D**
Select objects: **[Select A (Figure 2.210).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **YZ**
Point on YZ plane: **260,0**
Delete old objects? <N> **[Enter]**

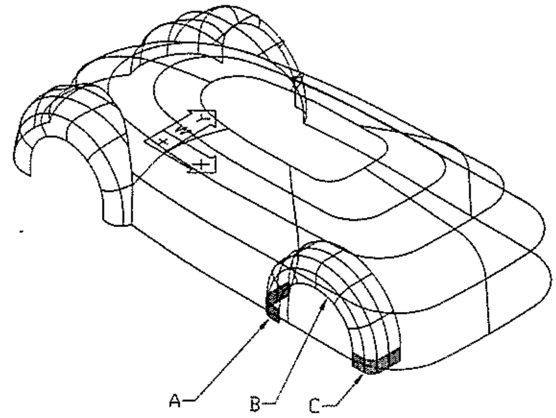


Figure 2.211 Extrude surface mirrored

Use the AMJOINSF command to join the extrude surfaces and the revolve surface into a single surface. Then use the MIRROR3D command to mirror the joined surface to form the other fender. (See Figure 2.212.)

<Surface> **<Edit Surface>** **<Join>**

Command: **AMJOINSF**
Select surfaces to join: [**Select A, B, and C (Figure 2.211).**]
Select surfaces to join: [**Enter**]

<Construct> **<3D Translations>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: [**Select A (Figure 2.211).**]
Select objects: [**Enter**]
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **ZX**
Point on YZ plane: **0,0**
Delete old objects? <N> [**Enter**]

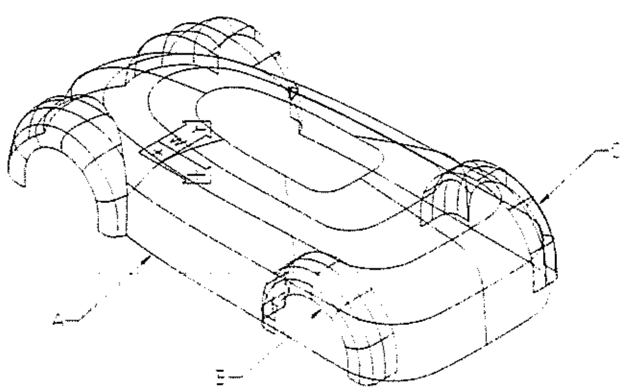


Figure 2.212 Revolve and extrude surfaces joined together and mirrored

To complete the rear fenders, use the AMINTERSF command twice to intersect the fenders with the main body. (See Figure 2.213.)

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.212).]**
Select second surface: **[Select B (Figure 2.212).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select A (Figure 2.212).]**
Select second surface: **[Select C (Figure 2.212).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

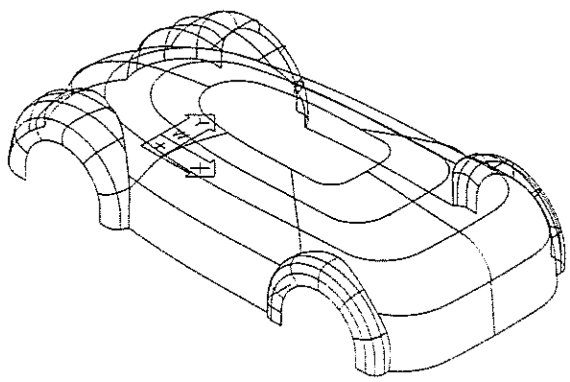


Figure 2.213 Rear fenders intersected with the main body

The wires for the rear fenders are not required. Turn off layer Rear and set the current layer to Top.

[Mechanical Main] **[Layer Control]**

Off layer: **Rear**
Current layer: **Top**

Greenhouse

The greenhouse of the model car is a loft u surface. It is constructed from five wires. Set the UCS to rotate 90° about the X axis. Then set the display to the front view by using the shortcut key [9]. After that, use the SPLINE command to construct three splines. (See Figure 2.214.)

<Assist> <UCS> <X Axis Rotate>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

Command: **9**

<Design> <Spline>

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-11,55	**70,110**	**145,120**	**215,105**	**255,65**	[Enter]	[Enter]	[Enter]
19,51	**80,104**	**145,116**	**200,102**	**232,58**	[Enter]	[Enter]	[Enter]
90,40	**104,57**	**135,70**	**166,57**	**180,40**	[Enter]	[Enter]	[Enter]

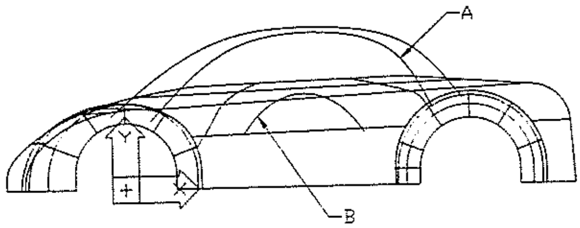


Figure 2.214 Three splines constructed

Use the MOVE command to move two splines. Then use the COPY command to copy two splines. After that, set the display to an isometric view. (See Figure 2.215.)

<Construct> <Move>

Command: **MOVE**
Select objects: **[Select A (Figure 2.214).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,42**
Second point of displacement: **[Enter]**

<Construct> <Move>

Command: **MOVE**
Select objects: **[Select B (Figure 2.214).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,65**
Second point of displacement: **[Enter]**

<Construct> **<Copy>**

Command: **COPY**
Select objects: **[Select A (Figure 2.214).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,-84**
Second point of displacement: **[Enter]**

<Construct> **<Copy>**

Command: **COPY**
Select objects: **[Select B (Figure 2.214).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,-130**
Second point of displacement: **[Enter]**

Command: **8**

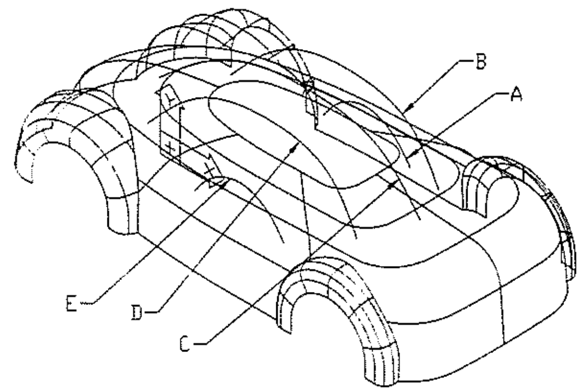


Figure 2.215 Splines copied and moved

The wires for the loft u surface are complete. Set the current layer to Surface. Then use the AMLOFTU command to construct a loft u surface. (See Figure 2.216.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Loft U...>**

Command: **AMLOFTU**
Select U wires: **[Select A, B, C, D, and E (Figure 2.215).]**
Select U wires: **[Enter]**

[Curve Direction: **Align**
Curve Fit: **Smooth**
Respace
OK]

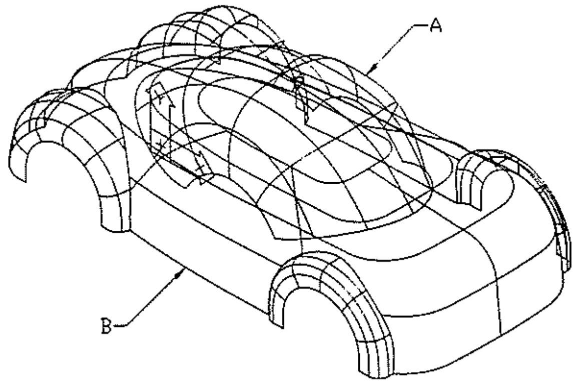


Figure 2.216 Loft u surface constructed

To complete the model, turn off layer Top. Then use the AMINTERSF command to intersect the greenhouse with the main body. (See Figure 2.217.)

[Mechanical Main] **[Layer Control]**

Off layer: **Top**
Current layer: **Surface**

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**
Select first surface: [**Select A (Figure 2.216).**]
Select second surface: [**Select B (Figure 2.216).**]

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

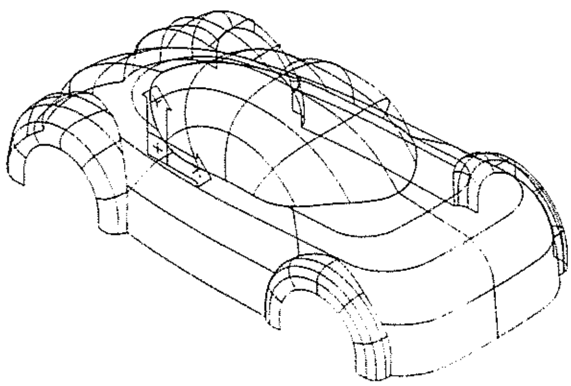


Figure 2.217 Greenhouse intersected with the main body

The model is complete. Save your drawing.

<File> <Save>

File name: **Car1.dwg**

In constructing this model, you learned how to break down a complex surface model into individual free-form surfaces, join surfaces together, and refine a surface. You also learned the difference between stretch sweeping and scale sweeping.

2.11 Scale Model Car Project 2

To further enhance your knowledge of applying surface modeling tools to real-world projects, you will work on another scale model car. Figure 2.218 shows the rendered image of the model together with the assembly of the solid parts that you will construct in Chapters 3 and 4. Figure 2.219 shows the surfaces that you will construct in this chapter.

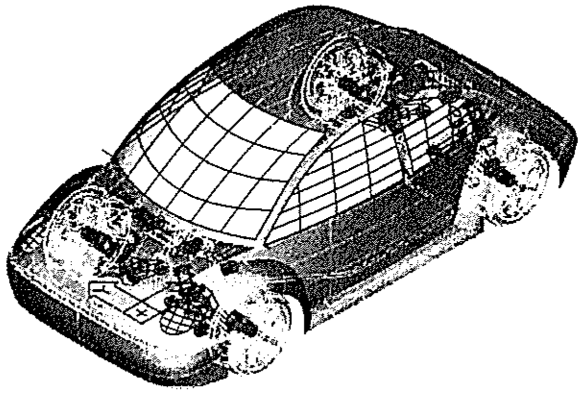


Figure 2.218 1/10 scale model car

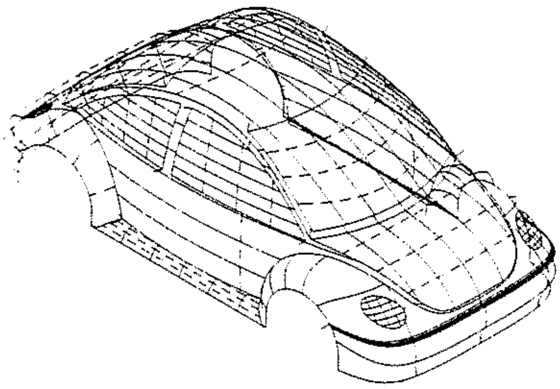


Figure 2.219 Surface model of the body of the model car

Analysis

Before you start to build this model, take some time to analyze the model and think about what surfaces are required and what wires are needed for making the surfaces. Here, your goal is to make a set of surfaces that make up the model of the car body. Basically, you can treat the windows and lights separately and make them at a later stage. Figure 2.220 shows the model with the windows and lights removed.

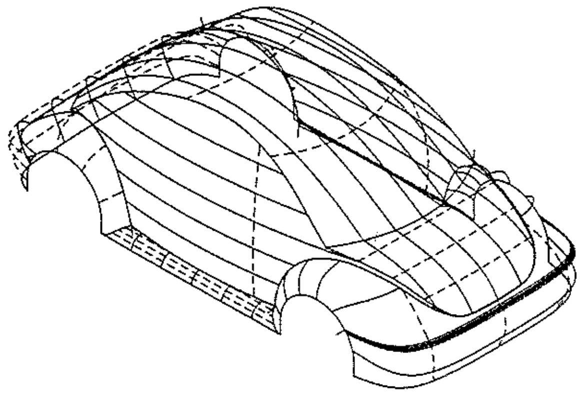


Figure 2.220 Windows and lights removed from the model

To help you see more clearly what surfaces are required, Figure 2.221 shows an exploded view of these surfaces.

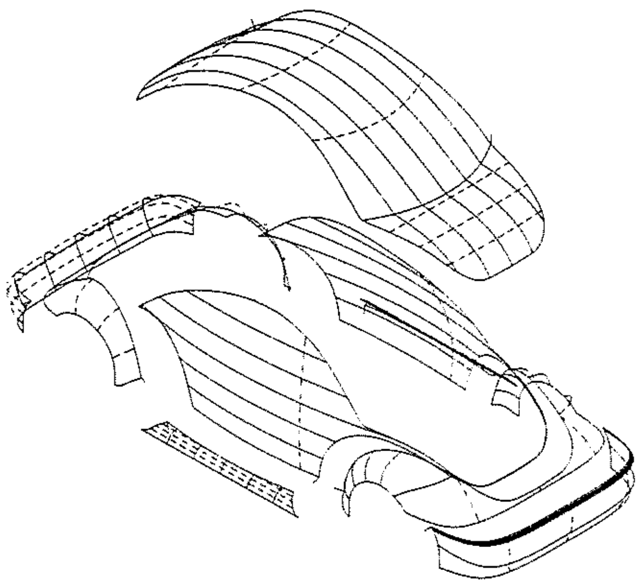


Figure 2.221 Exploded view of the surfaces

From the exploded view, you can see that most of the surfaces are trimmed surfaces. To make these trimmed surfaces, you have to make the base surfaces larger than the sizes of the trimmed boundaries. Figure 2.222 shows an exploded view of the base surfaces.

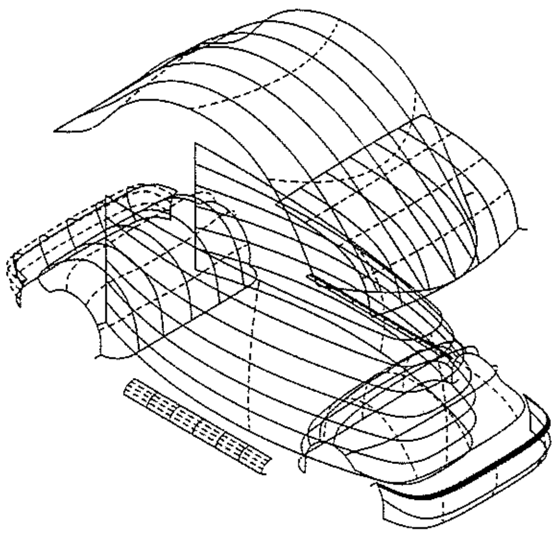


Figure 2.222 Exploded view of the base surfaces

Knowing the general shape of the base surfaces, you can start to think about the wires. Figures 2.223 through 2.227 show the trimmed surfaces, base surfaces, and wires required to make the various surfaces of the model car.

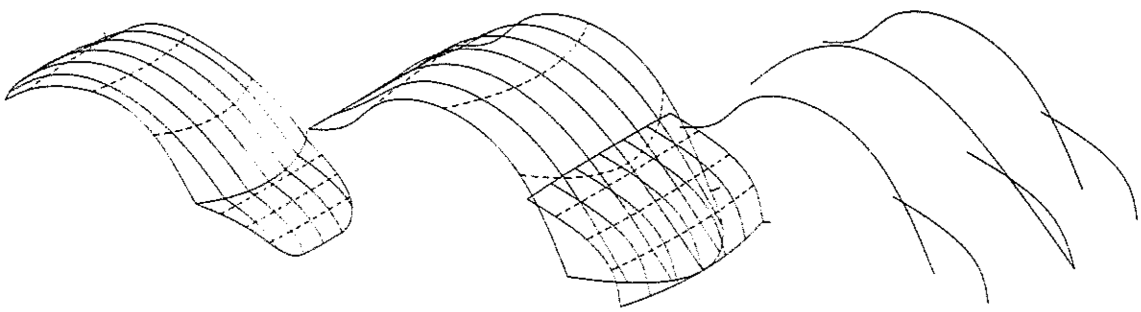


Figure 2.223 Trimmed surfaces, base surfaces, and wires for the top surfaces

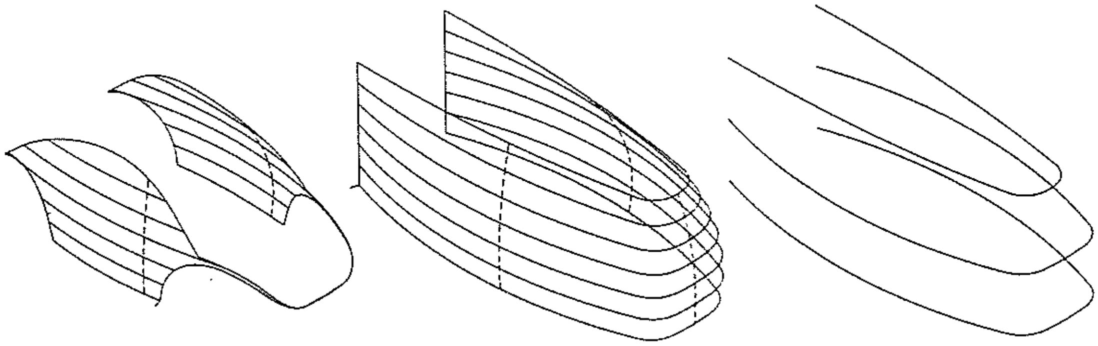


Figure 2.224 Trimmed surface, base surface, and wires for the side surface

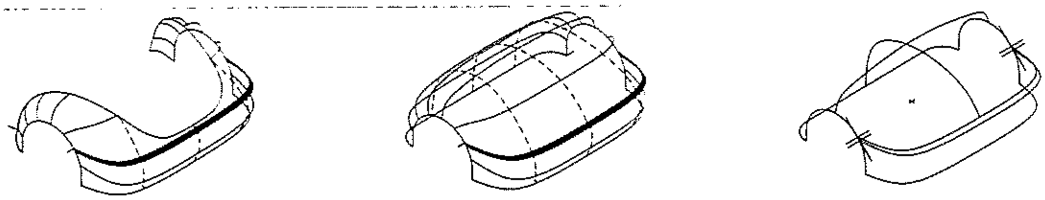


Figure 2.225 Trimmed surfaces, base surfaces, and wires for the front surfaces

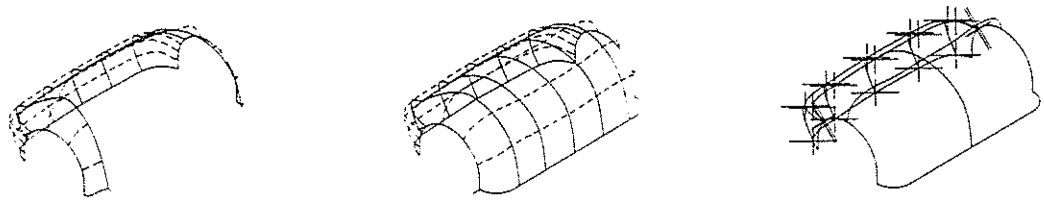


Figure 2.226 Trimmed surfaces, base surfaces, and wires for the rear surfaces

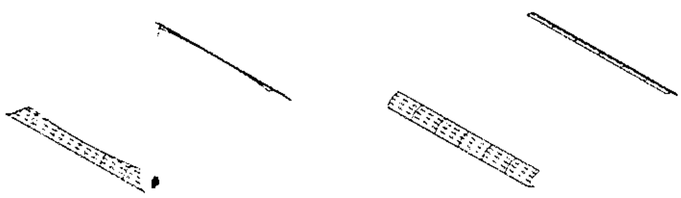


Figure 2.227 Trimmed surfaces, base surfaces, and wire for the skirt surfaces

In Figure 2.220 and Figures 2.223 through 2.227, you can see that the shapes of the trimmed edges correlate with the profiles, shapes, and relative positions of the adjacent surfaces. Because of this intercorrelation, it would be very hard to determine the wires for any individual surface in isolation from the others. You have to think of the wires for all the surfaces holistically. However, you cannot work on all the surfaces simultaneously. You must start from one surface and then improvise. To obtain a satisfactory result, you may have to iterate a number of times, constructing the wires, making the surfaces, intersecting the surfaces to obtain the trimmed edges, checking the profiles and shapes of the trimmed surfaces, and starting all over again until the required surface profiles and trimmed edges are obtained.

Here, the iterations have been done for you. The wires are determined after some trial and error and fine-tuning. Follow the steps shown below to make the wires, construct the surfaces, and intersect the surfaces. After making the surfaces and properly trimming them, you will get a model that resembles Figure 2.220.

To make the window and light openings, you need some more wires. Figure 2.228 shows the boundaries of the openings.

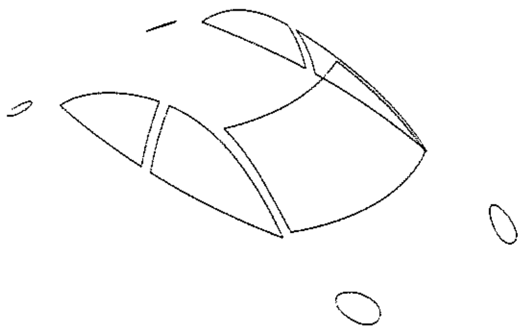


Figure 2.228 Boundary edges of the windows and lights

Briefly, you will perform the following tasks:

1. Set up the drawing.
2. Make the wires and base surfaces of the top of the car.
3. Make the wires and base surface of the side of the car.
4. Intersect and trim the top and side surfaces.
5. Make the wires and base surfaces of the front of the car.
6. Intersect and trim the front and side surfaces.
7. Make the wires and base surfaces of the rear of the car.
8. Intersect and trim the rear, top, and side surfaces.
9. Make the wires and base surfaces of the skirts of the car.
10. Intersect and trim the skirt, side, front, and rear surfaces.
11. Make the wires for the window and light openings.
12. Project and trim the window and light openings.
13. Make the window and light panels.

Drawing Setup

Start a new drawing from scratch and use metric defaults.

<File> **<New...>**

Construct seven additional layers: Surface, Front, Rear, Side, Skirt, Top, and Window. Layer Surface is used for holding the surfaces and other layers are used for holding the wires of the parts of the model. Set the current layer to Top. Then set the linetype scale to 20.

<Assist> **<Format>** **<Layer...>**

Command: **LAYER**

[Layer

Name	Color	Linetype
0	White	Continuous
Front	Magenta	Continuous
Rear	Green	Continuous
Side	Blue	Continuous
Skirt	Red	Continuous
Surface	Yellow	Continuous
Top	Green	Continuous
Window	White	Continuous

Current Layer: **Top**

OK]

Command: **LTSCALE**
New scale factor: **20**

Use the AMPREFS command to set the related system variables. From the Surfaces tab of the Desktop Preferences dialog box, select the [Model Size...] button to bring up the Approximate Model Size dialog box. Then set the model size to 3000 millimeters and select the [OK] button. On returning to the Desktop Preferences dialog box, set the vector length to 10 units and select the [OK] button to exit.

<Surface> **<Preferences...>**

Command: **AMPREFS**

Now set the UCS icon to be displayed at the origin position.

<Assist> <√ **Icon at Origin**>

The drawing setup is now complete.

Top Surfaces

To begin, you will work on the top part of the model car. The top part has two surfaces, the top and the hood. To define the contours and profiles of the surfaces, you will construct six smooth splines. To see the splines better, set the UCS to rotate 90° about the X axis. Then set the display to the plan view of the new UCS by using the shortcut key [9].

<Assist> **<UCS>** **<X Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

Command: **9**

On the new UCS, construct two splines. As we explained earlier, the iterations of trying out the splines, making the surfaces, and trimming the surfaces have been done for you. For these splines and other wires that you will construct, you need only input the given coordinates. (See Figure 2.229.)

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-98,0	**0,72**	**82,115**	**200,115**	**260,84**	**344,0**	**[Enter]**	**[Enter]**	**[Enter]**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-9,0	**12,35**	**82,105**	**200,107**	**255,72**	**292,30**	**344,0**	**[Enter]**	**[Enter]**	**[Enter]**

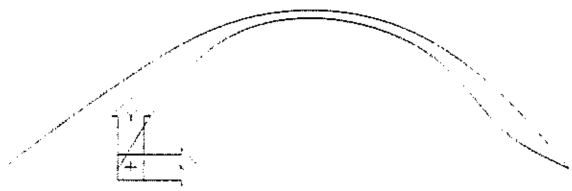


Figure 2.229 Two splines constructed

Repeat the SPLINE command to construct two more splines. (See Figure 2.230.)

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-98,0	**-77,42**	**0,68**	**47,70**	**[Enter]**	**[Enter]**	**[Enter]**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-82,0	**-72,36**	**0,65**	**47,67**	**[Enter]**	**[Enter]**	**[Enter]**

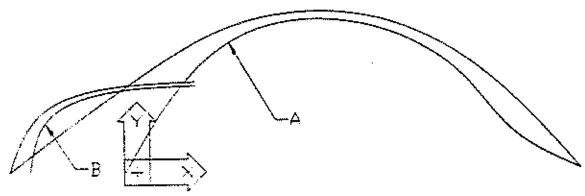


Figure 2.230 Two more splines constructed

Move and copy two splines. Then set the display to an isometric view by using the shortcut key [8]. (See Figure 2.231.)

<Construct> **<Move>**

Command: **MOVE**
Select objects: [**Select A and B (Figure 2.230).**]
Select objects: [**Enter**]
Base point or displacement: **0,0,100**
Second point of displacement: [**Enter**]

<Construct> **<Copy>**

Command: **COPY**
Select objects: [**Select A and B (Figure 2.230).**]
Select objects: [**Enter**]
Base point or displacement: **0,0,-200**
Second point of displacement: [**Enter**]

Command: **8**

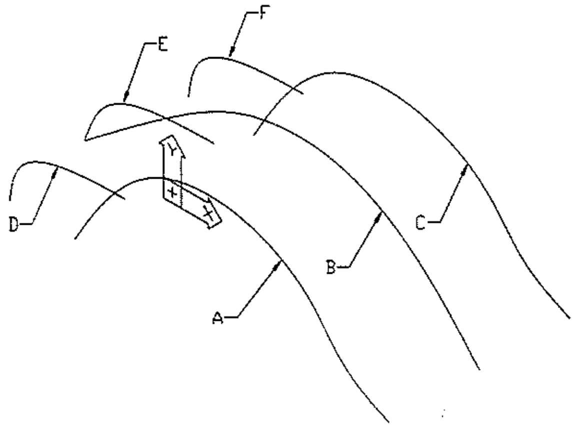


Figure 2.231 Splines moved and copied, display set to isometric

The wires for the top surfaces are complete. Now set the current layer to Surface. Then use the AMLOFTU command to make two loft u surfaces. (See Figure 2.232.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Loft U...>**

Command: **AMLOFTU**
Select U wires: [**Select A, B, and C (Figure 2.231).**]
Select U wires: [**Enter**]

[Curve Direction: **Align**
Curve Fit: **Smooth**
Respace
OK]

Command: [**Enter**]
AMLOFTU
Select U wires: [**Select D, E, and F (Figure 2.231).**]
Select U wires: [**Enter**]

[Curve Direction: **Align**
Curve Fit: **Smooth**
Respace
OK]

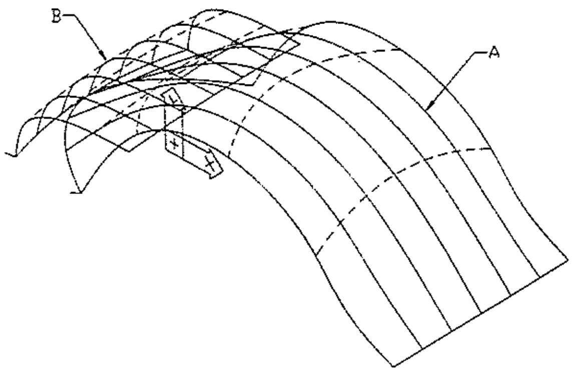


Figure 2.232 Two loft u surfaces constructed

Now you have two intersecting loft u surfaces. To trim away the unwanted portions of the surfaces, use the AMINTERSF command. While trimming, select the portion you want to retain. (See Figure 2.233.)

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**

Select first surface: **[Select A (Figure 2.232).]**
Select second surface: **[Select B (Figure 2.232).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

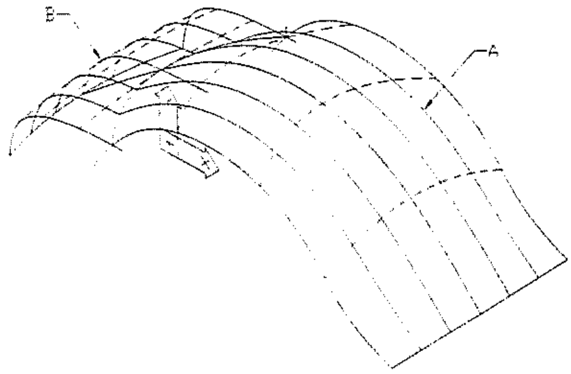


Figure 2.233 Intersecting loft u surfaces trimmed

The two intersecting loft u surfaces are trimmed. However, the surfaces are not yet complete, because they have to intersect with and be trimmed by the side and rear surfaces of the car. Now use the AMVISIBLE command to hide the top surfaces. You will come back to them later.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Objects
Hide **Select<** **]**

Select objects to hide: **[Select A and B (Figure 2.233).]**
Select objects to hide: **[Enter]**

[OK]

Side Surface

Turn off layer Top and set the current layer to Side. Then set the UCS orientation to World. After that, set the UCS origin to a new position.

[Mechanical Main] **[Layer Control]**

Off layer: **Top**
Current layer: **Side**

<Assist> **<UCS>** **<World>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Assist> **<UCS>** **<Origin>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **O**
Origin point: **0,0,-12**

On the new UCS, construct six arc segments. (See Figure 2.234.)

<Design> **<Arc>** **<3 Points>**

Command: **ARC**

Center/<Start point>:	Center/End/<Second point>:	End point:
315,-60	**130,-90**	**-78,-49**
-78,-49	**-83,0**	**-78,49**
-78,49	**130,90**	**315,60**
315,-60	**130,-55**	**-58,-29**
-58,-29	**-63,0**	**-58,29**
-58,29	**130,55**	**315,60**

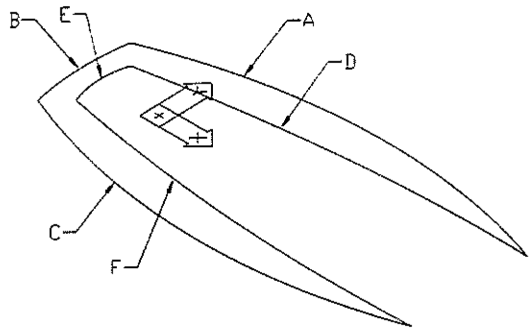


Figure 2.234 Six arcs constructed

Now use the AMFILLET3D command to round off four corners. The fillet radii of the two outer corners are 35 units and the fillet radii of the two inner corners are 30 units. (See Figure 2.235.)

<Surface> **<Edit Wireframe>** **<Fillet>**

Command: **AMFILLET3D**
Radius/Trim/<Select first object>: **R**
Radius: **35**
Radius/Trim/<Select first object>: **[Select A (Figure 2.234).]**
Radius/Trim/<Select second object>: **[Select B (Figure 2.234).]**

Command: **[Enter]**
AMFILLET3D
Radius/Trim/<Select first object>: **[Select B (Figure 2.234).]**
Radius/Trim/<Select second object>: **[Select C (Figure 2.234).]**

Command: **[Enter]**
AMFILLET3D
Radius/Trim/<Select first object>: **R**
Radius: **30**
Radius/Trim/<Select first object>: **[Select D (Figure 2.234).]**
Radius/Trim/<Select second object>: **[Select E (Figure 2.234).]**

Command: **[Enter]**
AMFILLET3D
Radius/Trim/<Select first object>: **[Select E (Figure 2.234).]**
Radius/Trim/<Select second object>: **[Select F (Figure 2.234).]**

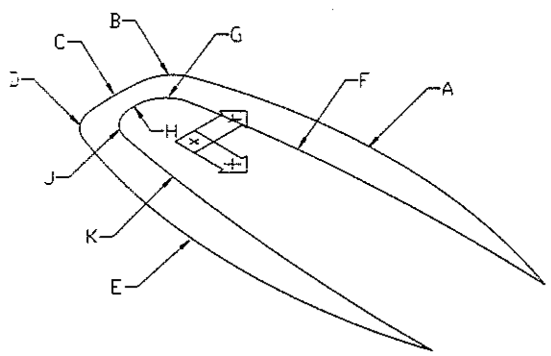


Figure 2.235 Corners filleted

Now you have ten arc segments. To join them into two splines, use the AMJOIN3D command.

<Surface> **<Edit Wireframe>** **<Join...>**

Command: **AMJOIN3D**

[Join3D
Automatic **Spline**
OK]

Select start wire: [**Select A (Figure 2.235).**]
Select wires to join: [**Select B, C, D, and E (Figure 2.235).**]
Select wires to join: [**Enter**]
Reverse? Yes/<No>: [**Enter**]

Command: [**Enter**]
AMJOIN3D

[Mode: **Automatic**
Output: **Spline**
OK]

Select start wire: [**Select F (Figure 2.235).**]
Select wires to join: [**Select G, H, J, and K (Figure 2.235).**]
Select wires to join: [**Enter**]
Reverse? Yes/<No>: [**Enter**]

By default, the DELOBJ variable is set to 1. Therefore, the selected arcs are erased after they are used to construct the splines. Now your screen shows two splines instead of ten arcs.

Because the side surface needs three splines that are spaced at intervals in the Z direction, use the COPY command to copy the outer spline and use the MOVE command to move the inner spline. (See Figure 2.236.)

<Construct> **<Copy>**

Command: **COPY**

Select objects: **[Select A (Figure 2.235).]**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0,72**
Second point of displacement: **[Enter]**

<Construct> **<Move>**

Command: **MOVE**
Select objects: **[Select F (Figure 2.235).]**
Select objects: **[Enter]**
Base point or displacement: **0,0,144**
Second point of displacement: **[Enter]**

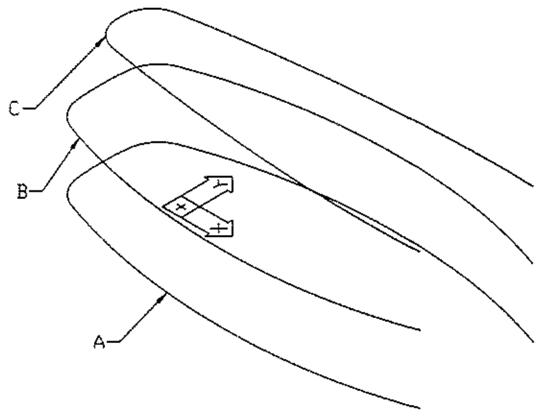


Figure 2.236 A spline copied and a spline moved

Set the current layer to Surface. Then use the AMLOFTU command to construct a loft u surface. After that, turn off layer Side. (See Figure 2.237.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Loft U...>**

Command: **AMLOFTU**
Select U wires: **[Select A, B, and C (Figure 2.236).]**
Select U wires: **[Enter]**

[Curve Direction: **Align**
Curve Fit: **Smooth**
Respace
OK]

[Mechanical Main] **[Layer Control]**

Off layer: **Side**
Current layer: **Surface**

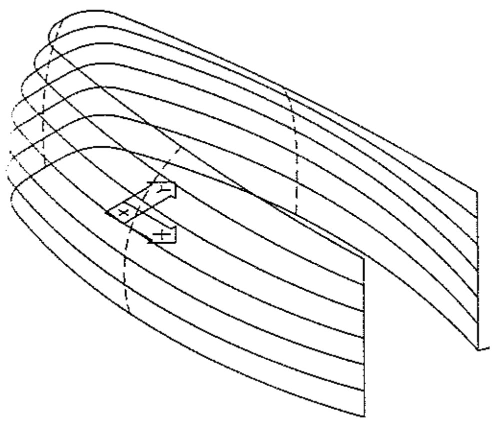


Figure 2.237 Loft u surface constructed

Now unhide the hidden top surfaces. (See Figure 2.238.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Surfaces**
OK]

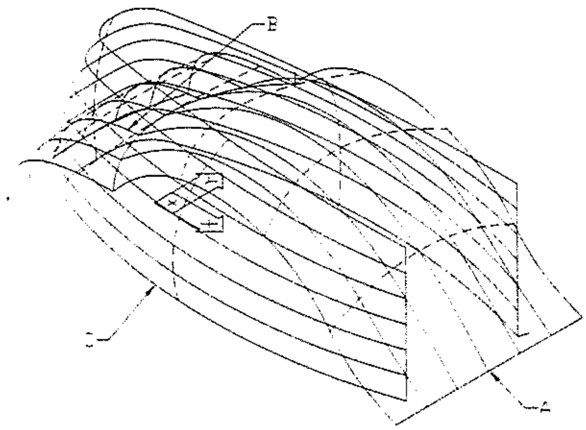


Figure 2.238 Top surfaces unhidden

To trim away the unwanted portions of the surfaces, take the following steps: Intersect the side surface and one of the top surfaces, trimming only the top surface; intersect the side surface and the second top surface, trimming only the top surface; copy the trimmed edges of the top surfaces; and use copied edges to trim the side surface.

Now run the AMINTERSF command on the side surface and a top surface. Trim only the top surface. (See Figure 2.239.)

<Surface> <Edit Surface> <Intersect Trim...>

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.238).]**
Select second surface: **[Select C (Figure 2.238).]**

Here you selected the top surface first. Therefore, you should select the [First Surface] button but not the [Second Surface] button to trim the first selected surface and leave the second selected surface intact.

[Surface Intersection
Trim **First Surface**
OK]

To reiterate, the top surface is selected first. It is trimmed. The side surface is selected second. It is not trimmed. Repeat the AMINTERSF command. Again, select the top surface first and select the [First Surface] button.

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select B (Figure 2.238).]**
Select second surface: **[Select C (Figure 2.238).]**

[Surface Intersection
Trim **First Surface**
OK]

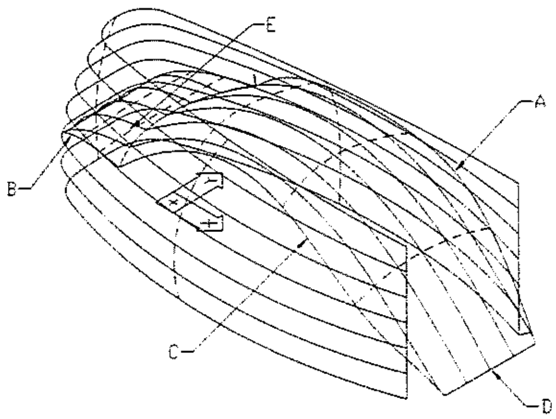


Figure 2.239 Top surfaces trimmed

Now the top surfaces are trimmed but the side surface is intact. To trim the side surface, copy the trimmed edges of the top surfaces and use the copied edges as projection wires for trimming the side surface in a single operation. Use the AMEDGE command to copy the edges. Then use the AMVISIBLE command to hide the top surfaces. (See Figure 2.240.)

<Surface> **<Edit Trim Edges>** **<Copy Edge>**

Command: **AMEDGE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **OUTPUT**
Polyline/<Spline>: **SPLINE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **COPY**
Select surface edge: **[Select A, B, and C (Figure 2.239).]**
Select surface edge: **[Enter]**

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: **[Select D and E (Figure 2.239).]**
Select objects to hide: **[Enter]**

[**OK**]

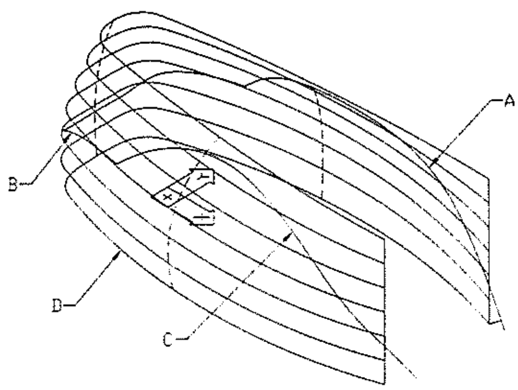


Figure 2.240 Edges copied, top surfaces hidden

Now use the AMPROJECT command to project the copied edges to trim the side surface. Select the lower part of the surface as the target surface. (See Figure 2.241.)

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: **[Select A, B, and C (Figure 2.240).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select D (Figure 2.240).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

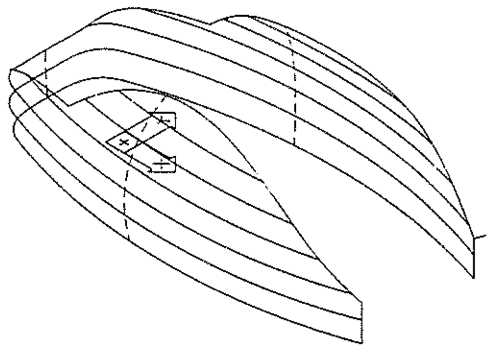


Figure 2.241 Side surface trimmed

Front Surfaces

Now you will work on the front fender and front bumper of the car. The front fender is a loft uv surface that requires two sets of wires in two orthogonal directions, and the front bumper is a sweep surface that requires three section wires and two rail wires. Set the current layer to Front. Then use the UCS command to rotate the UCS 90° about the X axis. After that, set the display to the plan view by using the shortcut key [9]. (See Figure 2.242.)

[Mechanical Main] **[Layer Control]**

Current layer: **Front**

<Assist> **<UCS>** **<X Axis Rotate>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **X**
Rotation angle about X axis: **90**

Command: **9**

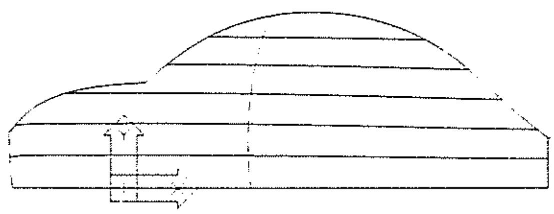


Figure 2.242 UCS rotated, display set to front view

On the XY plane of the new UCS, construct a polyline. Then construct two horizontal lines of infinite length by using the XLINE command. (See Figure 2.243.)

<Design> **<Polyline>**

Command: **PLINE**
From point: **38,0**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<90**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **A**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **@76<180**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **L**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<270**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **[Enter]**

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **H**
Through point: **0,37**
Through point: **0,34**
Through point: **[Enter]**

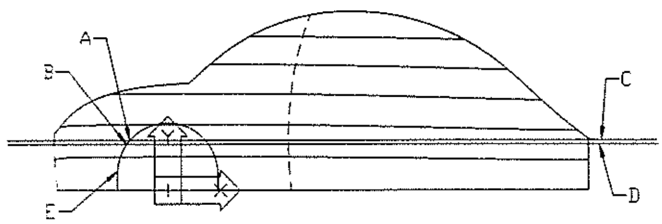


Figure 2.243 Polyline and xlines constructed

Set the point display mode to 3. Then use the POINT command to construct two points at the intersections between the polyline and the construction lines. After constructing the points, delete the construction lines. Then use the AMFITSPLINE command to fit the polyline into a spline. (See Figure 2.244.)

Command: **PDMODE**
New value for PDMODE: **3**

<Design> **<Point>** **<Single Point>**

Command: **POINT**
Point: **INT** of **[Select A, the intersection of C and E (Figure 2.243).]**

Command: **[Enter]**
POINT
Point: **INT** of **[Select B, the intersection of D and E (Figure 2.243).]**

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select C and D (Figure 2.243).]**
Select objects: **[Enter]**

<Surface> **<Edit Wireframe>** **<Spline Fit...>**

Command: **AMFITSPLINE**
Select wires: [**Select E (Figure 2.243).**]
Select wires: [**Enter**]

[Fit Spline
Spline Settings **Tolerance**
OK]

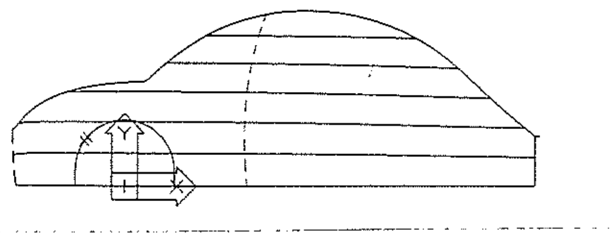


Figure 2.244 Points constructed, spline fitted

Because the default setting of the DELOBJ variable is 1, the polyline is erased after fitting to a spline. Now use the SPLINE command to construct a spline. (See Figure 2.245.)

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-87,37	**-76,52**	**48,52**	**60,0**	**[Enter]**	**[Enter]**	**[Enter]**

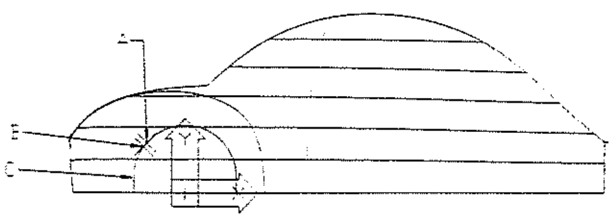


Figure 2.245 Spline constructed

Move and copy points A and B (Figure 2.245) and the fitted spline C (Figure 2.245). Then set the display to the left isometric view. After that, set the UCS orientation to World. (See Figure 2.246.)

<Construct> **<Move>**

Command: **MOVE**
Select objects: [**Select A, B, and C (Figure 2.245).**]
Select objects: [**Enter**]

Base point or displacement: **0,0,100**
Second point of displacement: **[Enter]**

<Construct> **<Copy>**

Command: **COPY**
Select objects: [**Select A, B, and C (Figure 2.245).**]
Select objects: [**Enter**]
Base point or displacement: **0,0,-200**
Second point of displacement: [**Enter**]

<View> **<Model Views>** **<Left Isometric>**

<Assist> **<UCS>** **<World>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

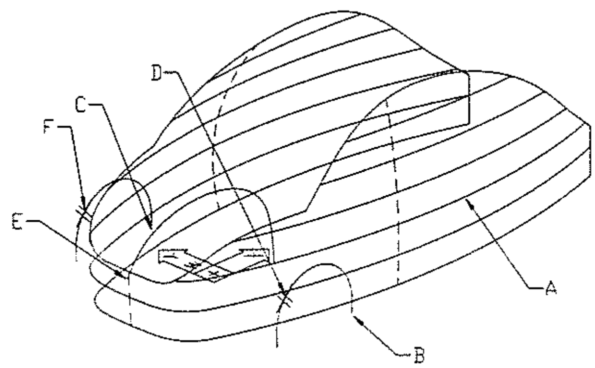


Figure 2.246 Points and spline copied and moved, display set to isometric view, and UCS set to World

The side surface is not required. Use the AMVISIBLE command to hide it. Then use the PLINE command to construct a polyline as part of the wires for the front surfaces. As we explained earlier, you can better control the shape of a wire by constructing polylines. After that, use the AMFITSPLINE command to fit the polyline into a spline and use the SPLINE command to construct two more splines. (See Figure 2.247.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: [**Select A (Figure 2.246).**]
Select objects to hide: [**Enter**]

[**OK**]

<Design> **<Polyline>**

Command: **PLINE**
From point: **END** of **[Select B (Figure 2.246).]**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **ARC**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **@22,22**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **LINE**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@156<90**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **ARC**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **END** of **[Select C (Figure 2.246).]**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **[Enter]**

<Surface> **<Edit Wireframe>** **<Spline Fit...>**

Command: **AMFITSPLINE**
Select wires: **LAST**
Select wires: **[Enter]**

[Fit Spline
Spline Settings **Tolerance**
OK]

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
0,-100,38	**0,-94,52**	**0,-80,58**	**0,80,58**	**0,94,52**	**0,100,38**	**[Enter]**	**[Enter]**	**[Enter]**

Command: **SPLINE**
Object/<Enter first point>: **NODE** of **[Select D (Figure 2.246).]**
Enter point: **-75,-85,25**
Close/Fit Tolerance/<Enter point>: **-85,-55,25**
Close/Fit Tolerance/<Enter point>: **END** of **[Select E (Figure 2.246).]**
Close/Fit Tolerance/<Enter point>: **-85,55,25**
Close/Fit Tolerance/<Enter point>: **-75,85,25**
Close/Fit Tolerance/<Enter point>: **NODE** of **[Select F (Figure 2.246).]**
Close/Fit Tolerance/<Enter point>: **[Enter]**
Enter start tangent: **[Enter]**
Enter end tangent: **[Enter]**

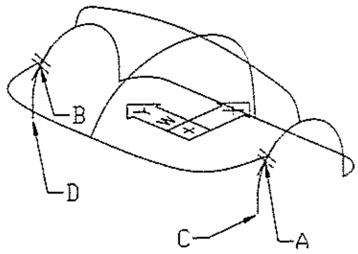


Figure 2.247 A spline fitted, two splines constructed

Repeat the SPLINE command to construct two more splines. (See Figure 2.248.)

Command: **[Enter]**
SPLINE
Object/<Enter first point>: **NODE** of **[Select A (Figure 2.247).]**
Enter point: **-80,-85,22**
Close/Fit Tolerance/<Enter point>: **-90,-55,22**
Close/Fit Tolerance/<Enter point>: **-92,0,22**
Close/Fit Tolerance/<Enter point>: **-90,55,22**
Close/Fit Tolerance/<Enter point>: **-80,85,22**
Close/Fit Tolerance/<Enter point>: **NODE** of **[Select B (Figure 2.247).]**
Close/Fit Tolerance/<Enter point>: **[Enter]**
Enter start tangent: **[Enter]**
Enter end tangent: **[Enter]**

Command: **[Enter]**
SPLINE
Object/<Enter first point>: **END** of **[Select C (Figure 2.247).]**
Enter point: **-75,-85,-12**
Close/Fit Tolerance/<Enter point>: **-85,-55,-12**
Close/Fit Tolerance/<Enter point>: **-87,0,-12**
Close/Fit Tolerance/<Enter point>: **-85,55,-12**
Close/Fit Tolerance/<Enter point>: **-75,85,-12**
Close/Fit Tolerance/<Enter point>: **END** of **[Select D (Figure 2.247).]**
Close/Fit Tolerance/<Enter point>: **[Enter]**
Enter start tangent: **[Enter]**
Enter end tangent: **[Enter]**

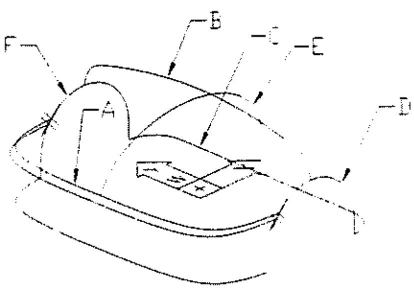


Figure 2.248 Two more splines constructed

In Figure 2.248, the splines B and E appear to intersect. However, this may not be so. To find out their 3D distance, you can use the AMDIST command. Note that the minimum distance shown here may differ from that on your screen.

Command: **AMDIST**
Select first set: Objects/<Instance>: **[Select B (Figure 2.248).]**
Next instance: **[Enter]**
Select second set: Objects/<Instance>: **[Select E (Figure 2.248).]**
Next instance: **[Enter]**
Output: Line/<Display>: **D**
Minimum distance:<5.438257>

On your screen, you will see a red line showing the minimum distance. Before you make a loft uv surface from these wires, you can either modify the splines to reduce their 3D distance or use the AMPREFS command to set the Join Gap tolerance to a larger value, say 6 units. Now set the current layer to Surface. Then use the AMLOFTUV command to construct a loft uv surface. (See Figure 2.249.)

<Surface> <Preferences...>

Command: **AMPREFS**

[Desktop Preferences
Surfaces
Join Gap Tolerance: **6**
OK]

[Mechanical Main] [Layer Control]

Current layer: **Surface**

<Surface> <Create Surface> <LoftUV>

Command: **AMLOFTUV**
Select U wires: **[Select A, B, and C (Figure 2.248).]**
Select U wires: **[Enter]**
Select V wires: **[Select D, E, and F (Figure 2.248).]**
Select V wires: **[Enter]**

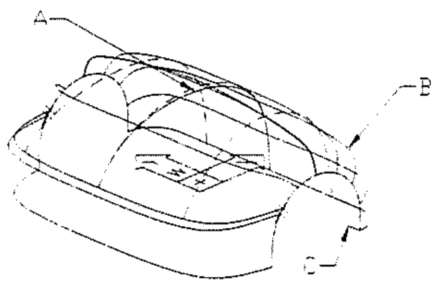


Figure 2.249 Loft uv surface constructed

Before making this surface from the U- and V-lines, you checked the distance and set the tolerance. Now you may want to know how the final surface deviates from the input wires. Use the AMCHECKFIT command. Note that the maximum distance of the selected wire from the surface is 2.7188 units; this value may differ from yours. To show how the surface deviates from the input wires, you can use the G option to display a graph. To see the graph more clearly, use the S option to set the magnifying scale factor to 10. (See Figure 2.250.)

<Surface> **<Utilities>** **<Check fit>**

Command: **AMCHECKFIT**
Select check wires: [**Select A (Figure 2.249).**]
Select check wires: [**Enter**]
Select target wire or surface: [**Select B (Figure 2.249).**]
Minimum distance = 0, Maximum distance = 2.7188
Graph/Scale/Table/<eXit>: **S**
Scale: **10**
(Scale = 10)
Graph/Scale/Table/<eXit>: **G**
Graph/Scale/Table/<eXit>: **X**

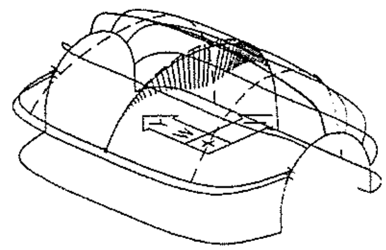


Figure 2.250 Magnified graph showing the distance between the input wire and the output surface

The normal direction of a surface depends on the direction of the input wires. Here, the normal vector C (Figure 2.249) is pointing toward the inside of the surface. To change the normal direction, use the AMEDITSF command.

<Surface> **<Edit Surface>** **<Flip normal>**

Command: **AMEDITSF**
Select surfaces: [**Select B (Figure 2.249).**]
Select surfaces: [**Enter**]
Direction/Preview/Span/Truncate/U grips/V grips/<eXit>: **D**
Direction/Preview/Span/Truncate/U grips/V grips/<eXit>: **X**

Now use the AMVISIBLE command to unhide the side surface. After you select the [Select<] button, the hidden objects appear. On your screen, select the side surface to unhide it. (See Figure 2.251.)

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: **[Select the side surface.]**
Select objects to unhide: **[Enter]**

[OK]

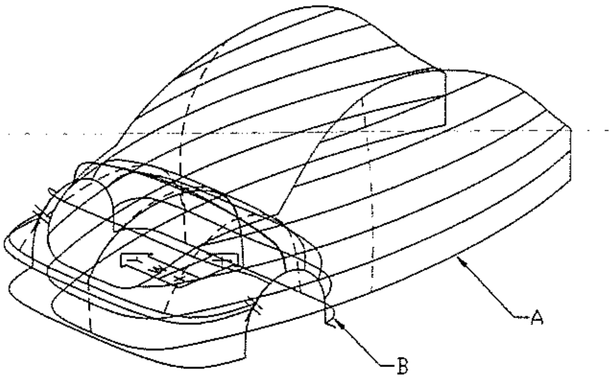


Figure 2.251 Surface normal flipped, side surface unhidden

To trim away the unwanted portions of the intersecting surfaces, use the AMINTERSF command. (See Figure 2.252.)

<Surface> <Edit Surface> <Intersect Trim...>

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.251).]**
Select second surface: **[Select B (Figure 2.251).]**

[Surface Intersection
Trim First
Trim Second
OK]

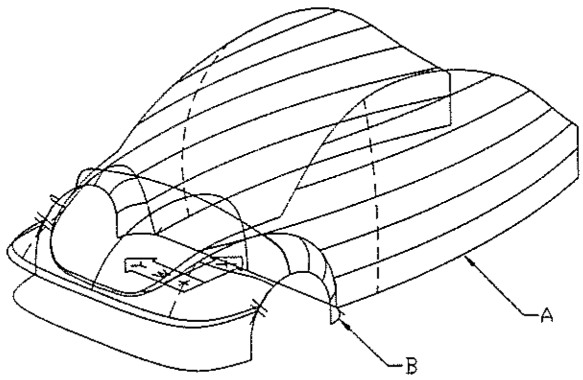


Figure 2.252 Intersecting side and front fender surfaces trimmed

To hide the intersected front and side surfaces temporarily, use the AMVISIBLE command. (See Figure 2.253.)

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: **[Select A and B (Figure 2.252).]**
Select objects to hide: **[Enter]**

[OK]

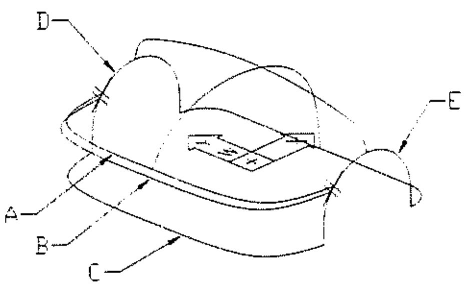


Figure 2.253 Side and front fender surfaces hidden

After hiding the surfaces, use the AMSWEEPSF command to construct a sweep surface by using three section wires and two rail wires. (See Figure 2.254.) The front surfaces are now complete.

<Surface> <Create Surface> <Sweep>

Command: **AMSWEEPSF**

Select cross sections: **[Select A, B, and C (Figure 2.253).]**
Select cross sections: **[Enter]**
Select rails: **[Select D (Figure 2.253).]**
Select rails: **[Select E (Figure 2.253).]**

[Transition: **Stretch**
OK]

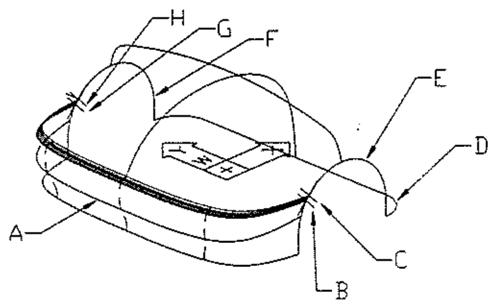


Figure 2.254 Sweep surface constructed

Rear Surfaces

Now you will work on the rear surfaces. Use the AMVISIBLE command to hide the front bumper surface.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: **[Select A (Figure 2.254).]**
Select objects to hide: **[Enter]**

[OK]

Because some wires for the rear surfaces are similar to those for the front surfaces, use the MIRROR3D command to mirror the three wires D, E, and F (Figure 2.254) and the four points B, C, G, and H (Figure 2.254). The mirror plane is a YZ plane that resides midway between the front and rear axles. The distance between the front and rear wheels is 260 units. (See Figure 2.255.)

<Construct> **<3D Transitions>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select B, C, D, E, F, G, and H (Figure 2.254).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **YZ**
Point on YZ plane: **130,0**

Delete old objects? <N> **N**

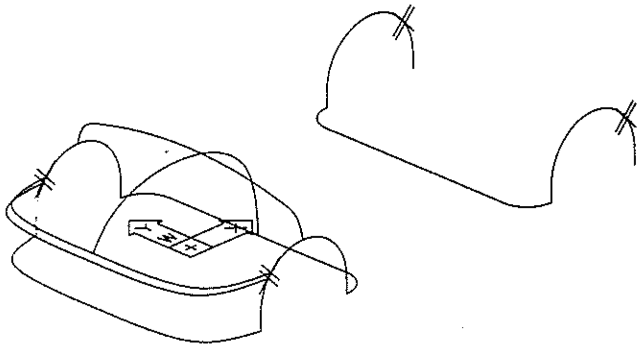


Figure 2.255 Front bumper surface hidden, wires and points mirrored

Use the shortcut key [8] to set the display to an isometric view. Then set the current layer to Rear. After that, use the POINT command to construct 15 points. (See Figure 2.256.)

Command: **8**

[Mechanical Main] **[Layer Control]**

Current layer: **Rear**

<Design> **<Point>** **<Single Point>**

Command: **POINT**

Input point coordinates according to the following table.

X	Y	Z
311	-93	-12
321	-55	-12
323	0	-12
321	55	-12
311	93	-12
311	-93	25
321	-55	25
323	0	25
321	55	25
311	93	25
321	-93	22
331	-55	22
333	0	22
331	55	22
321	93	22

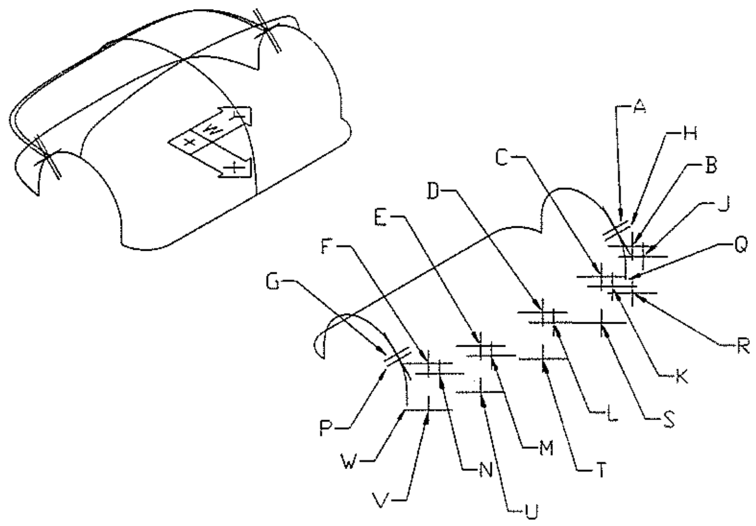


Figure 2.256 Points constructed

Set the running object snap mode to Node (snapping to a point). Then use the SPLINE command to construct eight splines. (See Figure 2.257.)

<Assist> **<Object Snap Settings...>**

Command: **OSNAP**

[Running Osnap **Node**
OK]

<Design> **<Spline>**

Command: **SPLINE**

Spline	First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
1	**A**	**B**	**C**	**D**	**E**	**F**	**G**	**[Enter]**	**[Enter]**	**[Enter]**
2	**H**	**J**	**K**	**L**	**M**	**N**	**P**	**[Enter]**	**[Enter]**	**[Enter]**
3	**End** of **Q**	**R**	**S**	**T**	**U**	**V**	**End** of **W**	**[Enter]**	**[Enter]**	**[Enter]**

Spline	First Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
4	**B**	**J**	**R**	**[Enter]**	**[Enter]**	**[Enter]**
5	**C**	**K**	**S**	**[Enter]**	**[Enter]**	**[Enter]**
6	**D**	**L**	**T**	**[Enter]**	**[Enter]**	**[Enter]**
7	**E**	**M**	**U**	**[Enter]**	**[Enter]**	**[Enter]**
8	**F**	**N**	**V**	**[Enter]**	**[Enter]**	**[Enter]**

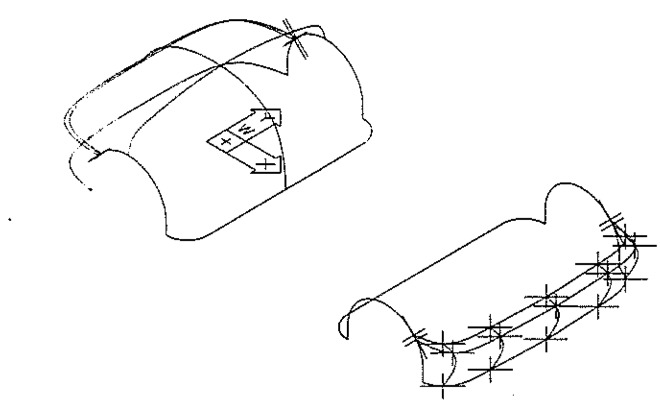


Figure 2.257 Eight splines constructed

Clear all the running object snap mode by using the OSNAP command. Then use the SPLINE command to construct two more splines. (See Figure 2.258.)

<Assist> **<Object Snap Settings...>**

Command: **OSNAP**

[Running Osnap **Clear All**
OK]

<Design> **<Spline>**

Command: **SPLINE**
Object/<Enter first point>: **323,0,25**
Enter point: **309,0,39**
Close/Fit Tolerance/<Enter point>: **260,0,56**
Close/Fit Tolerance/<Enter point>: **212,0,35**
Close/Fit Tolerance/<Enter point>: **200,0,-12**
Close/Fit Tolerance/<Enter point>: [**Enter**]
Enter start tangent: [**Enter**]
Enter end tangent: [**Enter**]

Command: [**Enter**]
SPLINE
Object/<Enter first point>: **260,100,38**
Enter point: **260,94,52**
Close/Fit Tolerance/<Enter point>: **260,80,58**
Close/Fit Tolerance/<Enter point>: **260,40,56**
Close/Fit Tolerance/<Enter point>: **260,0,56**
Close/Fit Tolerance/<Enter point>: **260,-40,56**
Close/Fit Tolerance/<Enter point>: **260,-80,58**
Close/Fit Tolerance/<Enter point>: **260,-94,52**
Close/Fit Tolerance/<Enter point>: **260,-100,38**
Close/Fit Tolerance/<Enter point>: [**Enter**]
Enter start tangent: [**Enter**]
Enter end tangent: [**Enter**]

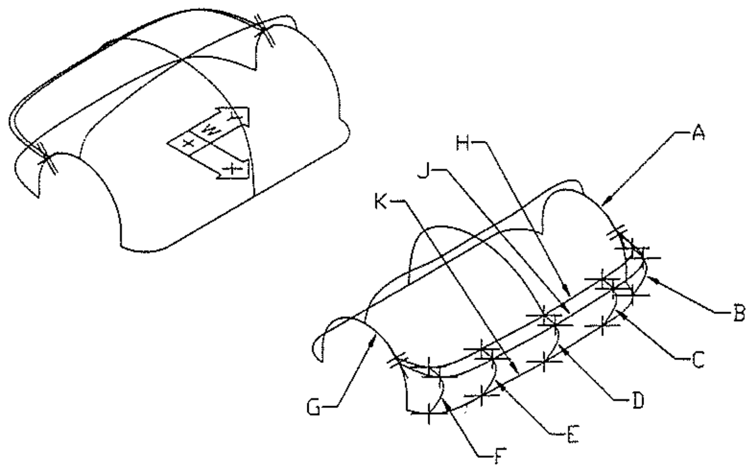


Figure 2.258 Two more splines constructed

Set the current layer to Surface. Then use the AMLOFTUV command to construct a loft uv surface. (See Figure 2.259.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<LoftUV>**

Command: **AMLOFTUV**
Select U wires: **[Select A, B, C, D, E, F, and G (Figure 2.258).]**
Select U wires: **[Enter]**
Select V wires: **[Select H, J, and K (Figure 2.258).]**
Select V wires: **[Enter]**

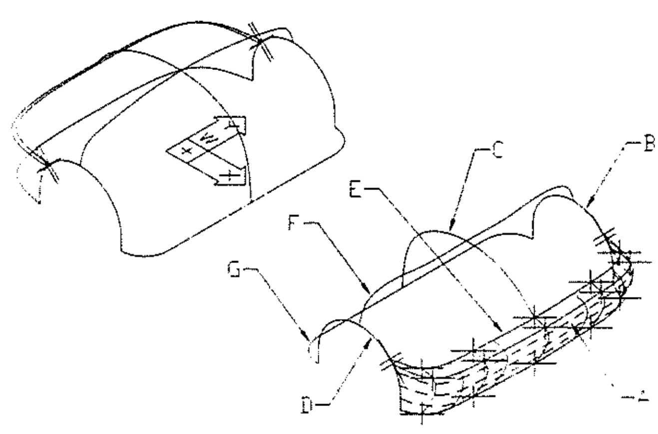


Figure 2.259 Loft uv surface constructed

Now use the AMVISIBLE command to hide the rear bumper surface A (Figure 2.259). Then use the AMLOFTUV command to construct another loft uv surface. After that, turn off layers Rear and Front and unhide the top and side surfaces. (See Figure 2.260.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: [**Select A (Figure 2.259).**]
Select objects to hide: [**Enter**]

[**OK**]

<Surface> **<Create Surface>** **<LoftUV>**

Command: **AMLOFTUV**
Select U wires: [**Select B, C, and D (Figure 2.259).**]
Select U wires: [**Enter**]
Select V wires: [**Select E, F, and G (Figure 2.259).**]
Select V wires: [**Enter**]

[Mechanical Main] **[Layer Control]**

Off layers: **Rear and Front**
Current layer: **Surface**

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: [**Select B and C (Figure 2.260).**]
Select objects to unhide: [**Enter**]

[**OK**]

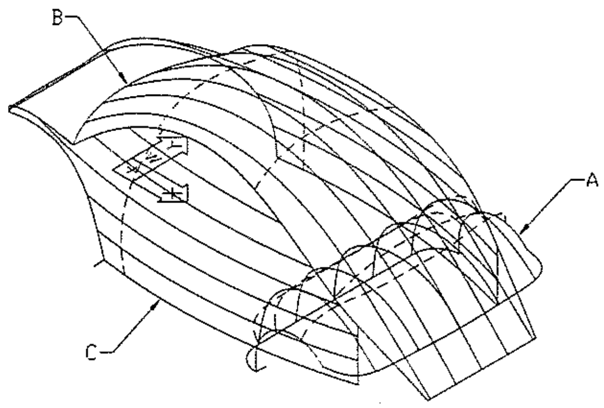


Figure 2.260 Rear bumper surface hidden, rear fender surface constructed, and top and side surfaces unhidden

To trim away the unwanted portions of the three intersecting surfaces, you have to perform a few steps: intersect the rear surface with the top surface, trimming only the top surface; intersect the rear surface with the side surface, trimming only the side surface; copy the trimmed edges of the top and side surfaces; hide the top and side surfaces; and use the copied edges to trim the rear surface. Now apply the AMINTERSF command on the rear and top surfaces. Trim only the top surface. After that, repeat the AMINTERSF command on the rear and side surfaces. Trim only the side surface. (See Figure 2.261.)

<Surface> <Edit Surface> <Intersect Trim...>

Command: **AMINTERSF**
Select first surface: [**Select A (Figure 2.260).**]
Select second surface: [**Select B (Figure 2.260).**]

[Surface Intersection
Trim **Second Surface**
OK]

<Surface> <Edit Surface> <Intersect Trim...>

Command: **AMINTERSF**
Select first surface: [**Select A (Figure 2.260).**]
Select second surface: [**Select C (Figure 2.260).**]

[Surface Intersection
Trim **Second Surface**
OK]

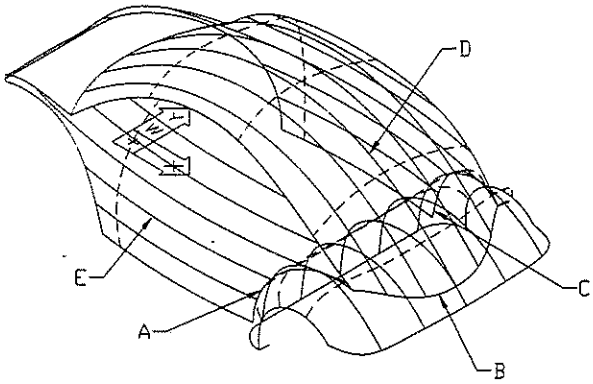


Figure 2.261 Top and side surfaces trimmed

Copy the trimmed edges of the top and side surfaces. Then hide these two surfaces. (See Figure 2.262.)

<Surface> **<Edit Trim Edges>** **<Copy Edge>**

Command: **AMEDGE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **OUTPUT**
Polyline/<Spline>: **SPLINE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **COPY**
Select surface edge: [**Select A, B, and C (Figure 2.261).**]
Select surface edge: [**Enter**]
Edge(s) copied.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Select<**]

Select objects to hide: [**Select D and E (Figure 2.261).**]
Select objects to hide: [**Enter**]

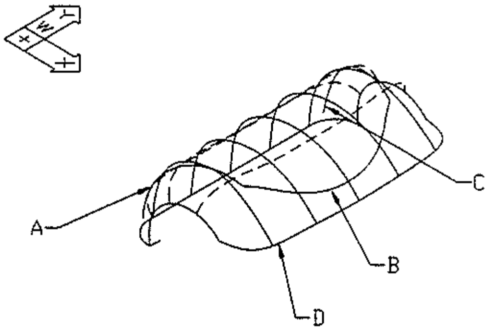


Figure 2.262 Trimmed edges copied, top and side surfaces hidden

Use the AMPROJECT command to project the copied edges in a direction normal to the rear fender surface to trim away the unwanted portion. (See Figure 2.263.)

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: [**Select A, B, and C (Figure 2.262).**]
Select wires to project: [**Enter**]
Select target surfaces: [**Select D (Figure 2.262).**]
Select target surfaces: [**Enter**]

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

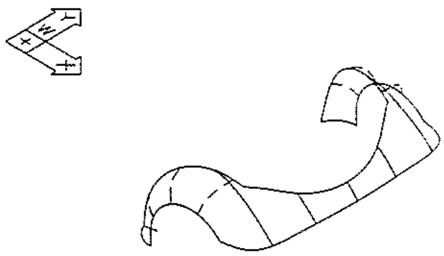


Figure 2.263 Rear fender surface trimmed

The rear fender surface is now complete. Use the AMVISIBLE command to unhide all the surfaces. (See Figure 2.264.)

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide All

OK]

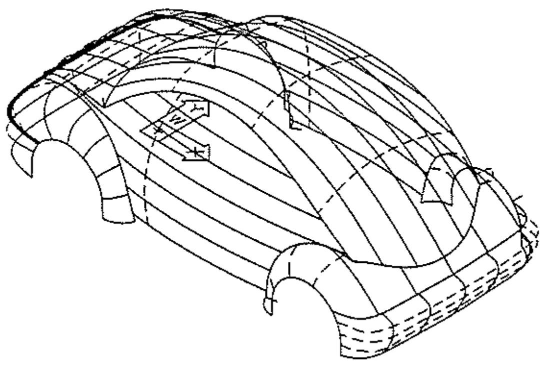


Figure 2.264 All hidden surfaces unhidden

Skirt Surfaces

Now proceed to the skirt surfaces. Set the current layer to Skirt. Then use the SPLINE command to construct a spline. (See Figure 2.265.)

[Mechanical Main] **[Layer Control]**

Current layer: **Skirt**

<Design> **<Spline>**

Command: **SPLINE**
Object/<Enter first point>: **38,-100,-12**
Enter point: **38,-95,-5**
Close/Fit Tolerance/<Enter point>: **38,-85,-2**
Close/Fit Tolerance/<Enter point>: **[Enter]**
Enter start tangent: **[Enter]**
Enter end tangent: **[Enter]**

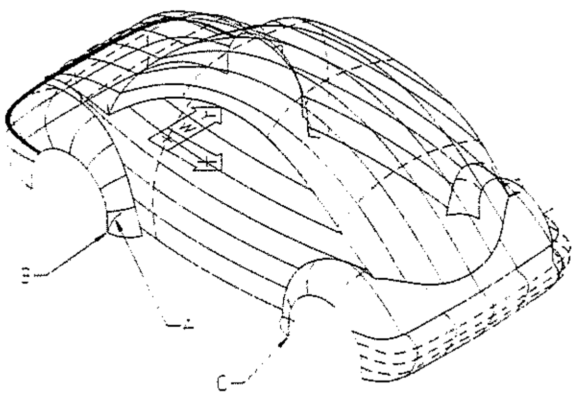


Figure 2.265 Spline for skirt surface constructed

Set the current layer to Surface. Then construct an extrude surface. (See Figure 2.266.)

[Mechanical Main] **[Layer Control]**

Current layer: **Surface**

<Surface> **<Create Surface>** **<Extrude>**

Command: **AMEXTRUDESF**
Select wires: **[Select A (Figure 2.265).]**
Select wires: **[Enter]**
Direction: Viewdir/Wire/X/Y/Z/<Start point>: **END** of **[Select B (Figure 2.265).]**
End point: **END** of **[Select C (Figure 2.265).]**
Taper angle: **0**

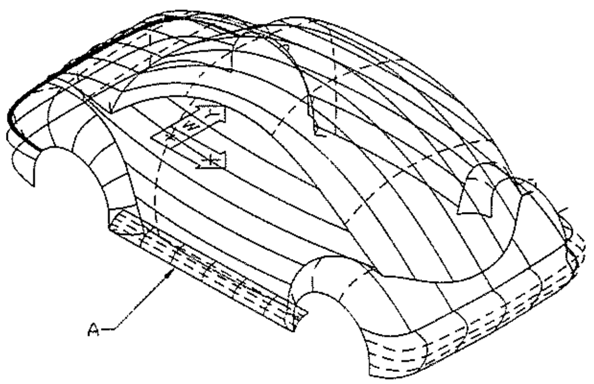


Figure 2.266 Extrude surface constructed

Mirror the extrude surface. (See Figure 2.267.)

<Construct> **<3D Translations>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select A (Figure 2.266).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **ZX**
Point on ZX plane <0,0,0>: **[Enter]**
Delete old objects? <N> **[Enter]**

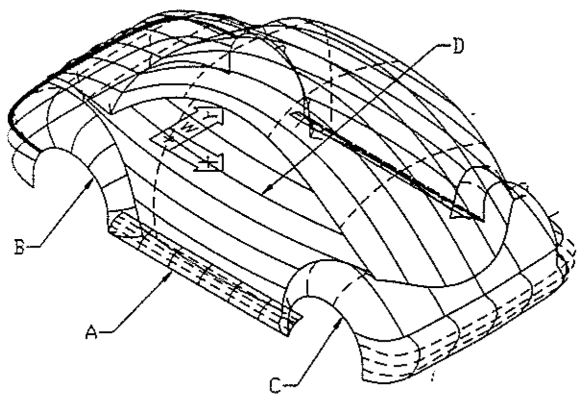


Figure 2.267 Extrude surface mirrored

Use the AMINTERSF command to trim the surfaces of one side of the car. (See Figure 2.268.)

<Surface> <Edit Surface> <Intersect Trim...>

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.267).]**
Select second surface: **[Select B (Figure 2.267).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select A (Figure 2.267).]**
Select second surface: **[Select C (Figure 2.267).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select A (Figure 2.267).]**
Select second surface: **[Select D (Figure 2.267).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

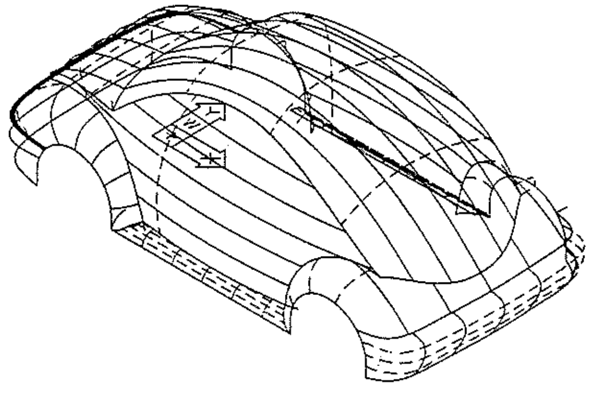


Figure 2.268 Skirt, side, front, and rear surfaces trimmed

Set the display to a back left isometric view. (See Figure 2.269.)

<View> **<Model View>** **<Back Left Iso>**

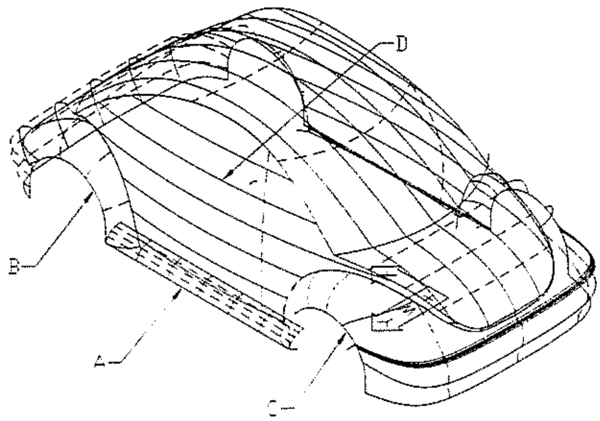


Figure 2.269 Back left isometric view

Use the AMINTERSF command to trim the other side of the car. (See Figure 2.270.)

<Surface> **<Edit Surface>** **<Intersect Trim...>**

Command: **AMINTERSF**
Select first surface: **[Select A (Figure 2.269).]**
Select second surface: **[Select B (Figure 2.269).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select A (Figure 2.269).]**

Select second surface: **[Select C (Figure 2.269).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

Command: **[Enter]**
AMINTERSF
Select first surface: **[Select A (Figure 2.269).]**
Select second surface: **[Select D (Figure 2.269).]**

[Surface Intersection
Trim **First Surface**
Second Surface
OK]

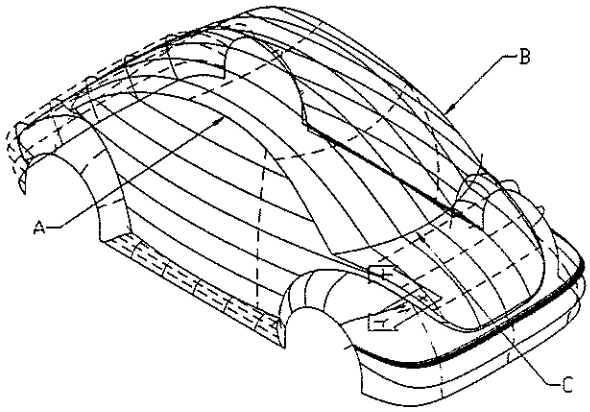


Figure 2.270 Skirt, side, front, and rear surfaces trimmed

The main surfaces of the car are complete. The wire on layer Skirt is not required. Turn off layer Skirt. As we have explained, the profile and shape of the trimmed edges rest on the location and shape of the adjacent intersecting surfaces. Therefore, if you are not satisfied with the shape of this model car, you can erase the surfaces, fine-tune the wires given in the foregoing paragraphs, construct the surfaces by using the revised wires, and intersect and trim the surfaces as may be required. Then save your drawing.

[Mechanical Main] **[Layer Control]**

Off layer: **Skirt**

<File> **<Save>**

Command: **Car2.dwg**

Window and Light Openings

To make the window and light openings, you have to construct wires and use the wires to trim the surfaces. For the window openings, you will copy the edges of the trimmed top and side surfaces. Then you will offset the wires from these copied edges and construct additional wires to describe the shapes of the windows. For the light openings, you will construct ellipses and then project them to the front and rear surfaces.

Now set the current layer to Window. Then use the AMEDGE command to copy three trimmed edges and output them as 3D polylines. After that, use the AMVISIBLE command to hide all the surfaces and set the UCS to a new position. (See Figure 2.271.)

[Mechanical Main] **[Layer Control]**

Current layer: **Window**

<Surface> **<Edit Trim Edges>** **<Copy Edge>**

Command: **AMEDGE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **OUTPUT**
Polyline/<Spline>: **POLYLINE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **COPY**
Select surface edge: **[Select A, B, and C (Figure 2.270).]**
Select surface edge: **[Enter]**
Edge(s) copied.

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Surfaces**
OK]

<Assist> **<UCS>** **<Z Axis Vector>**

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **ZAXIS**
Origin point: **0,0,0**
Point on positive portion of Z-axis: **-1,0**

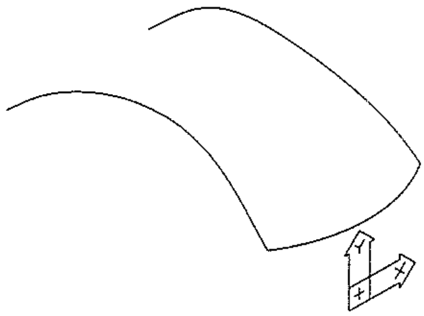


Figure 2.271 Edges copied, surfaces hidden, and UCS changed

To construct 3D offset wires, you use the AMOFFSET3D command. Because this command offsets the 3D polylines in a plane parallel to the plane of the current display view, not the UCS plane, use the shortcut key [9] to set the display to the front view of the model car. (See Figure 2.272.)

Command: **9**

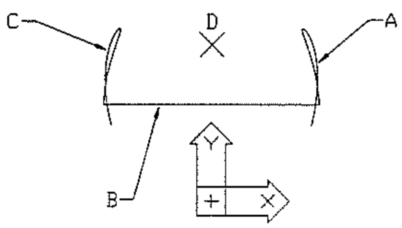


Figure 2.272 Front view of the model car

Note that the AMOFFSET3D command offsets 3D polylines (not splines) in 3D space. If you have mistakenly output the trimmed edges as splines instead of 3D polylines, you can convert the splines to a 3D polyline by using the AMUNSPLINE command.

<Surface> **<Edit Wireframe>** **<Unspline>**

Command: **AMUNSPLINE**
Select splines: [**Select A, B, and C (Figure 2.272).**]

If the selected objects are not splines, you will receive the following message:

3 was filtered out.
Select splines: [**Enter**]

Now use the AMOFFSET3D command to offset three 3D polylines. Then use the XLINE command to create a construction line. (See Figure 2.273.)

<Surface> **<Create Wireframe>** **<Offset Wire>**

Command: **AMOFFSET3D**
Offset distance or Through: **4**
Select 3D polyline to offset: **[Select A (Figure 2.272).]**
Side to offset?: **[Select D (Figure 2.272).]**
Select 3D polyline to offset: **[Select B (Figure 2.272).]**
Side to offset?: **[Select D (Figure 2.272).]**
Select 3D polyline to offset: **[Select C (Figure 2.272).]**
Side to offset?: **[Select D (Figure 2.272).]**
Select 3D polyline to offset: **[Enter]**

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **H**
Through point: **0,116**
Through point: **[Enter]**

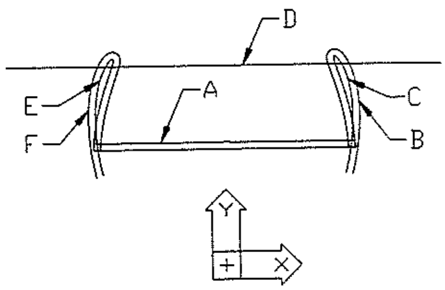


Figure 2.273 Polyline offset, construction line constructed

To obtain a set of wires for making the window openings, you need to trim the offset 3D polylines and the construction line. Because the 3D polylines may become segmented, trimming them can be laborious. Therefore, you will fit the 3D polylines into splines and then use the TRIM command to trim the fitted splines at the appropriate apparent intersection points. (See Figure 2.274.)

<Surface> **<Edit Wireframe>** **<Spline Fit...>**

Command: **AMFITSPLINE**
Select wires: **[Select A, B, C, E, and F (Figure 2.273).]**
Select wires: **[Enter]**

[Fit Spline
Fit Tolerance: **0.2**
OK]

<Modify> **<Trim>**

Command: **TRIM**
Select cutting edges:
Select objects: **[Select A, B, D, and F (Figure 2.273).]**
Select objects: **[Enter]**

<Select object to trim>/Project/Edge/Undo: **[As shown in Figure 2.274, select the parts that are to be trimmed.]**
<Select object to trim>/Project/Edge/Undo: **[Enter]**

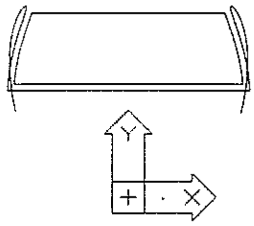


Figure 2.274 Wires trimmed

Set the display to a back left isometric view. Then unhide the top surface as in Figure 2.275.

<View> **<Model View>** **<Back Left Iso>**

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: **[Select E (Figure 2.275).]**
Select objects to unhide: **[Enter]**

[OK]

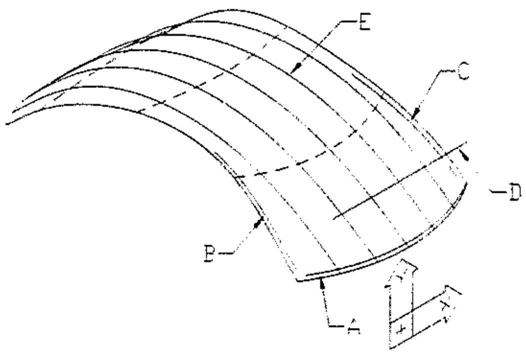


Figure 2.275 Display set to back left isometric, top surface unhidden

In Figure 2:275, D is the trimmed construction line and A, B, and C are the trimmed splines. You can see that they are not joining at their ends. However, they form a closed loop if you view them in a direction normal to the UCS. Note also in Figure 2.275 that if you project and trim the top surface with the wires, you will cut two openings, one at the front and one at the rear. To prevent cutting an opening at the rear, take three steps: project the wires onto the surface (this produces two sets of wires, one at the front and one at the rear), remove the unwanted wires at the rear, and use the wires at the front to cut an opening. Now use the AMPROJECT command to project a set of wires on the top surface. (See Figure 2.276.)

<Surface> **<Create Wireframe>** **<Project Wire...>**

Command: **AMPROJECT**
Select wires to project: [**Select A, B, C, and D (Figure 2.275).**]
Select wires to project: [**Enter**]
Select target surfaces: [**Select E (Figure 2.275).**]
Select target surfaces: [**Enter**]

[Project to Surface
Direction: **Normal to UCS**
Output Type: **Polyline**
OK]

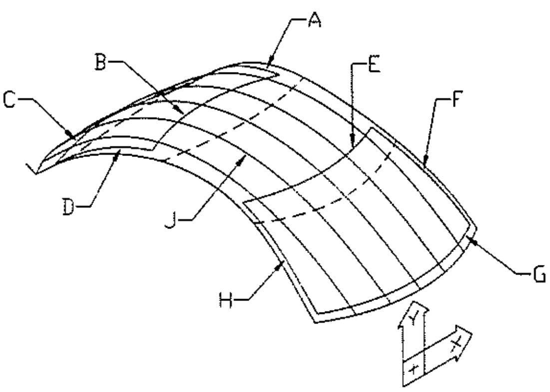


Figure 2.276 Wires projected on the top surface

In Figure 2.276, two sets of wires are obtained after projection, one set at the front (E, F, G, and H) and another set at the rear (A, B, C, and D). Because only the front set of wires is required, carefully select and erase the wires at the rear of the car. After that, project the remaining wires in a direction normal to the surface to trim the surface. (See Figure 2.277.)

<Modify> **<Erase>**

Command: **ERASE**
Select objects: [**Select A, B, C, and D (Figure 2.276).**]
Select objects: [**Enter**]

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: [**Select E, F, G, and H (Figure 2.276).**]
Select wires to project: [**Enter**]

Select target surfaces: **[Select J (Figure 2.276).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

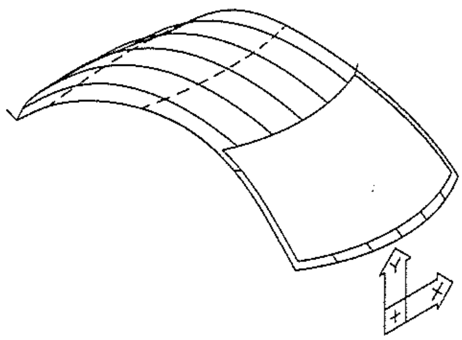


Figure 2.277 Wires at the rear erased, wires at the front projected to cut an opening

The front window opening is complete. Before you make the wires for the rear window opening, use the AMVISIBLE command to hide the top surface. (See Figure 2.278.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Surfaces**
OK]

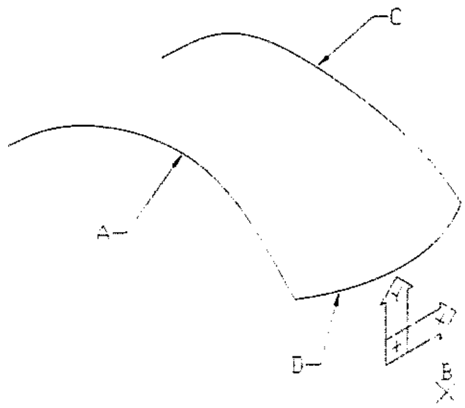


Figure 2.278 Top surface hidden

Use the BREAK command to break the two wires A and C (Figure 2.278) at their midpoints, and use the ERASE command to erase the unwanted wire D (Figure 2.278). (See Figure 2.279.)

<Modify> **<Break>**

Command: **BREAK**
Select object: **MID** of **[Select A (Figure 2.278).]**
Enter second point (or F for first point): **[Select B (Figure 2.278).]**

Command: **[Enter]**
BREAK
Select object: **MID** of **[Select C (Figure 2.278).]**
Enter second point (or F for first point): **[Select B (Figure 2.278).]**

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select D (Figure 2.278).]**
Select objects: **[Enter]**

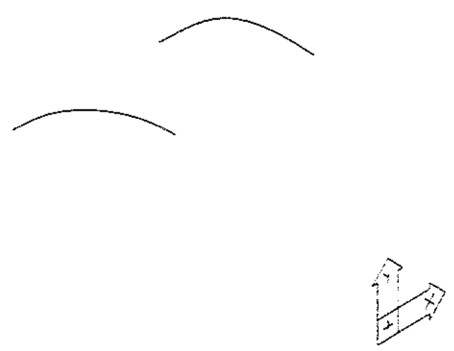


Figure 2.279 Wires edited

Use the shortcut key [9] to set the display to the plan view of the current UCS. Then construct two horizontal construction lines. (See Figure 2.280.)

Command: **9**

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **H**
Through point: **0,116**
Through point: **0,70**
Through point: **[Enter]**

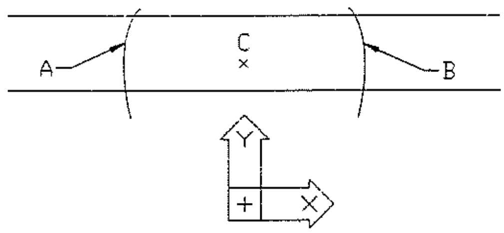


Figure 2.280 Display set, horizontal construction lines constructed

Use the AMUNSPLINE command to convert splines A and B (Figure 2.280) to polylines. Then construct two offset wires. (See Figure 2.281.)

<Surface> <Edit Wireframe> <Unspline>

Command: **AMUNSPLINE**
Select splines: **[Select A and B (Figure 2.280).]**
Select splines: **[Enter]**
Tolerance <???>: **[Enter]**

<Surface> <Create Wireframe> <Offset Wire>

Command: **AMOFFSET3D**
Offset distance or Through: **4**
Select 3D polyline to offset: **[Select A (Figure 2.280).]**
Side to offset?: **[Select C (Figure 2.280).]**
Select 3D polyline to offset: **[Select B (Figure 2.280).]**
Side to offset?: **[Select C (Figure 2.280).]**
Select 3D polyline to offset: **[Enter]**

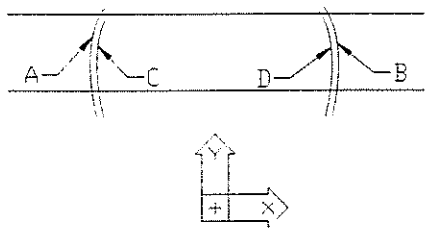


Figure 2.281 Offset wires constructed

Erase the original wires that were used to make the offset wires. Then fit the offset wires into splines. (See Figure 2.282.)

<Modify> <Erase>

Command: **ERASE**
Select objects: **[Select A and B (Figure 2.281).]**
Select objects: **[Enter]**

<Surface> <Edit Wireframe> <Spline Fit...>

Command: **AMFITSPLINE**
Select wires: **[Select C and D (Figure 2.281).]**
Select wires: **[Enter]**

[Fit Spline
Fit Tolerance: **0.2**
OK]

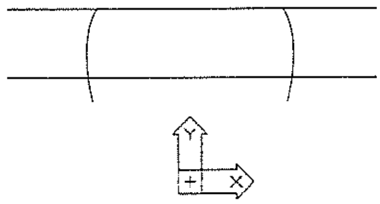


Figure 2.282 Wires erased, offset wires fitted into splines

Trim the wires as shown in Figure 2.283.

<Modify> <Trim>

Command: **TRIM**

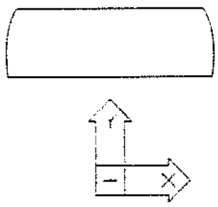


Figure 2.283 Wires trimmed

Wires for the silhouettes of the rear window are complete. Use the shortcut key [8] to set the display to an isometric view. Then use the AMVISIBLE command to unhide the top surface. (See Figure 2.284.)

Command: **8**

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: **[Select the top surface from your screen.]**
Select objects to unhide: **[Enter]**

[OK]

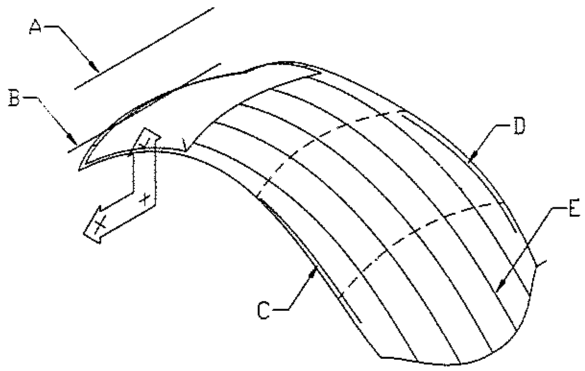


Figure 2.284 Top surface unhidden, display set

Now project the wires onto the top surface. As with the front windshield opening, you get two sets of wires projected on the top surface. Hide the surface so that you can delete the unwanted wires. (See Figure 2.285.)

<Surface> **<Create Wireframe>** **<Project Wire...>**

Command: **AMPROJECT**
Select wires to project: **[Select A, B, C, and D (Figure 2.284).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select E (Figure 2.284).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to UCS**
Output Type: **Polyline**
OK]

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Surfaces**
OK]

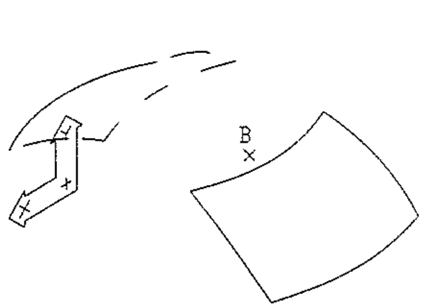


Figure 2.285 Wires projected, surface hidden

Erase the unwanted wires. Then unhide the top surface. (See Figure 2.286.)

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select A (Figure 2.285).]**
Other corner: **[Select B (Figure 2.285).]**
Select objects: **[Enter]**

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: **[Select the top surface from your screen.]**
Select objects to unhide: **[Enter]**

[OK]

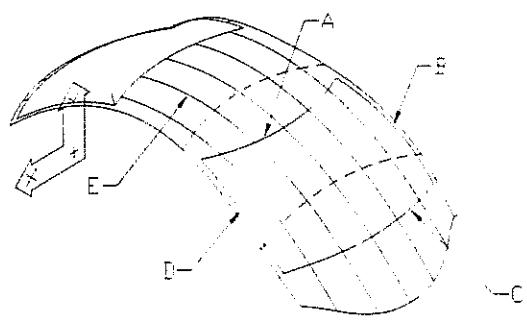


Figure 2.286 Unwanted wires erased, top surface unhidden

Now project the wires in a direction normal to the surface to cut an opening for the rear window. (See Figure 2.287.)

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select A, B, C, and D (Figure 2.286).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select E (Figure 2.286).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

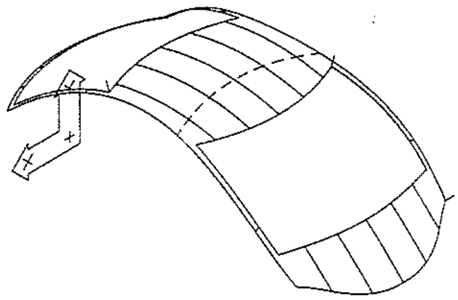


Figure 2.287 Rear window cut

The rear window opening is complete. Now you will work on the side window openings, using a different approach. Here, you will construct a planar surface on which you will project the edge of the top surface. Then you will offset the projected wire, add more wires, and modify them.

Rotate the UCS 90° about the Y axis. Then use the AMPLANE command to make a planar surface on the new UCS plane. (See Figure 2.288.)

<Assist> <UCS> <Y Axis Rotate>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **Y**
Rotation angle about Y axis: **90**

<Surface> <Create Surface> <Planar>

Command: **AMPLANE**
Plane/Wires/<First corner>: **0,0**
Second corner: **320,140**

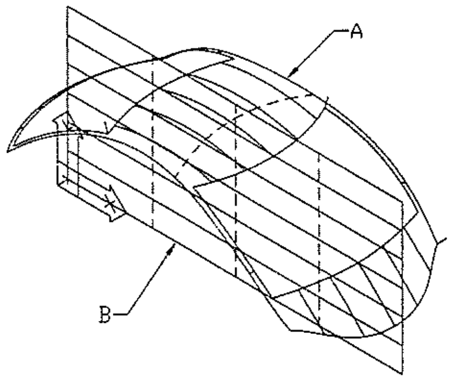


Figure 2.288 UCS rotated, planar surface constructed

Use the AMPROJECT command to project a trimmed edge of the top surface onto the planar surface to output a polyline. After that, erase the planar surface, hide the top surface, and set the display to the plan view. (See Figure 2.289.)

<Surface> <Create Wireframe> <Project Wire...>

Command: **AMPROJECT**
Select wires to project: [**Select A (Figure 2.288).**]
Select wires to project: [**Enter**]
Select target surfaces: [**Select B (Figure 2.288).**]
Select target surfaces: [**Enter**]

[Project to Surface
Direction: **Normal to surface**
Output Type: **Polyline**
OK]

<Modify> <Erase>

Command: **ERASE**
Select objects: [**Select B (Figure 2.288).**]
Select objects: [**Enter**]

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Object
Hide **Surfaces**
OK]

Command: **9**

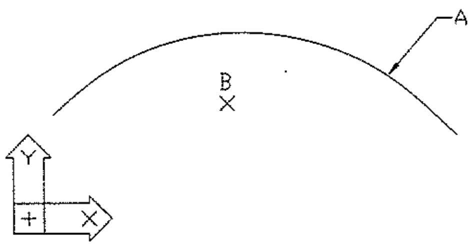


Figure 2.289 Edge projected, planar surface erased, top surface hidden, and display set

Using a planar surface as the target surface for projection, you obtain a 2D polyline. To offset the 2D polyline, you use the OFFSET command instead of the AMOFFSET3D command, because the latter works only with 3D polylines. Because the projected wire is not required, erase it. Then construct one horizontal and two vertical construction lines. (See Figure 2.290.)

<Construct> **<Offset>**

Command: **OFFSET**
Offset distance or Through: **4**
Select object to offset: [**Select A (Figure 2.289).**]
Side to offset? [**Select B (Figure 2.289).**]
Select object to offset: [**Enter**]

<Modify> **<Erase>**

Command: **ERASE**
Select objects: [**Select A (Figure 2.289).**]
Select objects: [**Enter**]

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **H**
Through point: **0,65**
Through point: [**Enter**]

Command: [**Enter**]
XLINE
Hor/Ver/Ang/Bisect/Offset/<From point>: **V**
Through point: **150,0**
Through point: **160,0**
Through point: [**Enter**]

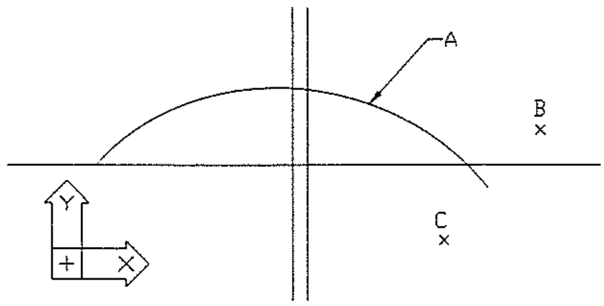


Figure 2.290 Wire offset, original wire erased, and construction lines constructed

Use the AMFITSPLINE command to fit the polyline into a spline with four control points. Then use the STRETCH command to modify the shape of the wire. After that, trim the wires as shown in Figure 2.291.

<Surface> <Edit Wireframe> <Spline Fit...>

Command: **AMFITSPLINE**
Select wires: [**Select A (Figure 2.290).**]
Select wires: [**Enter**]

[Fit Spline
Control Pts: **4**
OK]

<Modify> <Stretch>

Command: **STRETCH**
Select objects: [**Select B (Figure 2.290).**]
Other corner: [**Select C (Figure 2.290).**]
Select objects: [**Enter**]
Base point or displacement: **10<180**
Second point of displacement: [**Enter**]

<Modify> <Trim>

Command: **TRIM**

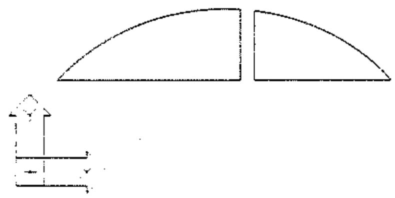


Figure 2.291 Wires trimmed

The wires for cutting the side windows are complete. Use the AMVISIBLE command to unhide the side surface. (See Figure 2.292.)

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **Select<**]

Select objects to unhide: **[Select the side surface from your screen.]**
Select objects to unhide: **[Enter]**

[OK **]**

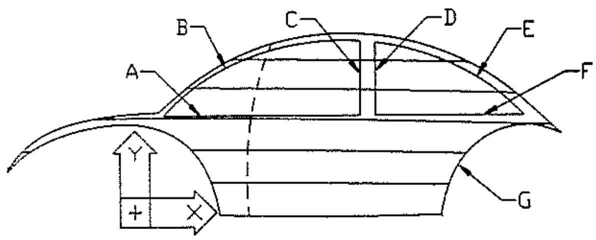


Figure 2.292 Side surface unhidden

To cut the side window openings, use the AMPROJECT command. (See Figure 2.293.)

<Surface> **<Edit Surface>** **<Project Trim...>**

Command: **AMPROJECT**
Select wires to project: **[Select A, B, C, D, E, and F (Figure 2.292).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select G (Figure 2.292).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to UCS**
Output Type: **Trim surface**
OK]

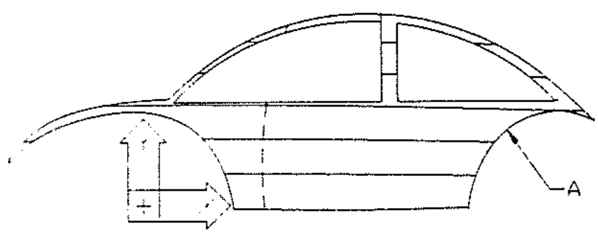


Figure 2.293 Side window openings cut

The side windows are complete. Now hide the side surface. Then unhide the front and rear surfaces. After that, set the display to an isometric view and set the UCS to rotate 90° about the Y axis. (See Figure 2.294.)

<Surface> <Visibility...>

Command: **AMVISIBLE**

[Desktop Visibility
Objects
Hide **Select<**]

Select objects to hide: **[Select A (Figure 2.293).]**
Select objects to hide: **[Enter]**

[Unhide **Select<**]

Select objects to unhide: **[Select the front and rear surfaces.]**
Select objects to unhide: **[Enter]**

[OK]

Command: **8**

<Assist> <UCS> <Y Axis Rotate>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **Y**
Rotation angle about X axis: **90**

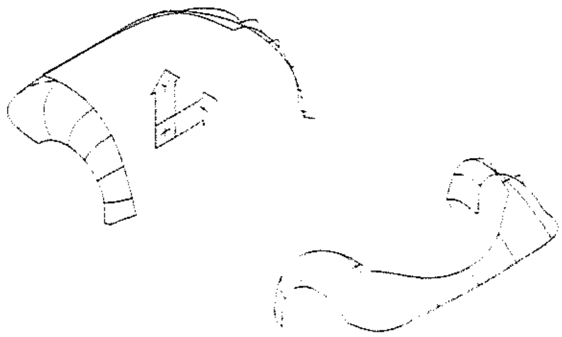


Figure 2.294 Side surface hidden, front and rear surfaces unhidden, and UCS set

On the new UCS plane, construct two ellipses for use as wires for making the front and rear light openings. (See Figure 2.295.)

<Design> <Ellipse> <Center>

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **-75,40**

Axis endpoint: **@12<0**
<Other axis distance>/Rotation: **8**

Command: **[Enter]**
ELLIPSE
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **75,40**
Axis endpoint: **@10<0**
<Other axis distance>/Rotation: **8**

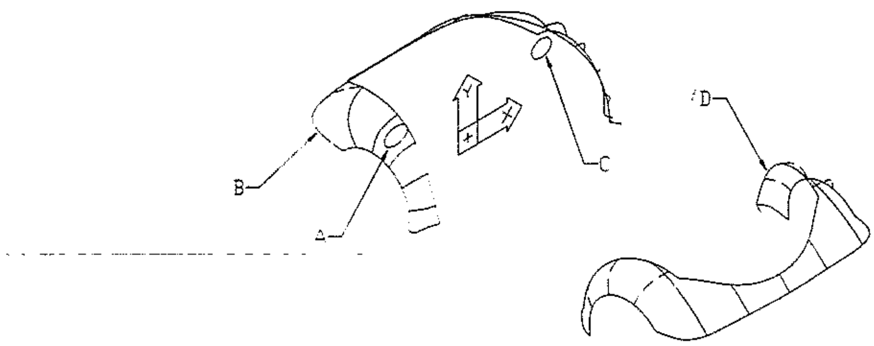


Figure 2.295 Two ellipses constructed

Now use the AMPROJECT command to project the ellipses onto the front and rear surfaces. (See Figure 2.296.)

<Surface> **<Create Wireframe>** **<Project Wire...>**

Command: **AMPROJECT**
Select wires to project: [**Select A (Figure 2.295).**]
Select wires to project: [**Enter**]
Select target surfaces: [**Select B (Figure 2.295).**]
Select target surfaces: [**Enter**]

[Project to Surface
Direction: **Normal to UCS**
Output Type: **Polyline**
OK]

Command: **[Enter]**
AMPROJECT
Select wires to project: [**Select C (Figure 2.295).**]
Select wires to project: [**Enter**]
Select target surfaces: [**Select D (Figure 2.295).**]
Select target surfaces: [**Enter**]

[Project to Surface
Direction: **Normal to UCS**
Output Type: **Polyline**
OK]

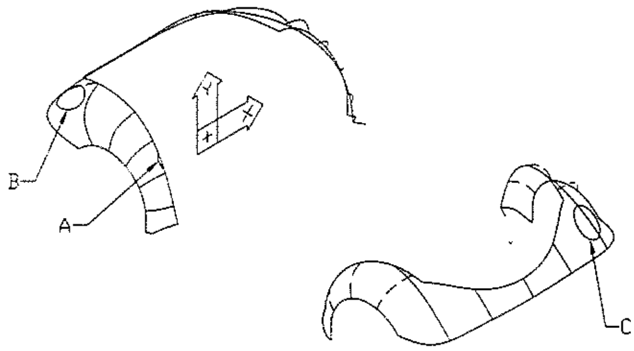


Figure 2.296 Ellipses projected

Erase wire A (Figure 2.296). Then use the MIRROR command to mirror the remaining projected wires. (See Figure 2.297.)

<Modify> **<Erase>**

Command: **ERASE**
Select objects: **[Select A (Figure 2.296).]**
Select objects: **[Enter]**

<Construct> **<3D Transitions>** **<3D Mirror>**

Command: **MIRROR3D**
Select objects: **[Select B and C (Figure 2.296).]**
Select objects: **[Enter]**
Plane by Object/Last/Zaxis/View/XY/YZ/ZX/<3points>: **YZ**
Point on YZ plane <0,0,0>: **[Enter]**
Delete old objects? <N> **[Enter]**

Figure 2.297 Unwanted wire erased, projected wires mirrored

The wires for making the light openings are complete. Use the AMPROJECT command to cut the light openings. To view the model more clearly, set the display to back left isometric view before cutting the front light openings. (See Figure 2.298.)

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select A and B (Figure 2.297).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select C (Figure 2.297).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

<View> <Model View> <Back Left Iso>

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select D and E (Figure 2.297).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select F (Figure 2.297).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

Figure 2.298 Light openings cut

The light openings are complete. Set the UCS to World. Then use the AMVISIBLE command to unhide all the hidden surfaces. After that, turn off layer Window. (See Figure 2.299.)

<Assist> <UCS> <World>

Command: **UCS**
Origin/ZAxis/3point/OBject/View/X/Y/Z/Prev/Restore/Save/Del/?/<World>: **W**

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[Desktop Visibility
Object
Unhide **All**
OK]

[Mechanical Main] **[Layer Control]**

Off layer: **Window**
Current layer: **Surface**

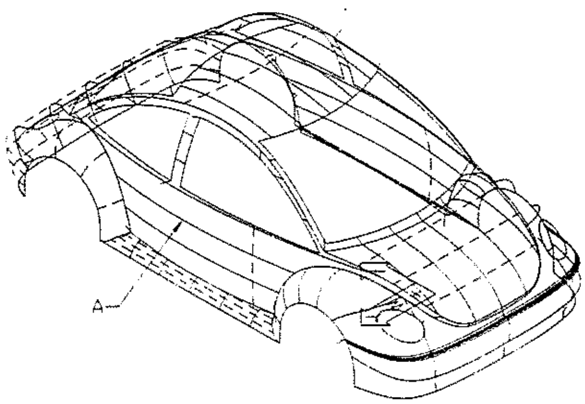


Figure 2.299 All surfaces unhidden

Window Panels and Light Lenses

To further add details to the model car, you will add the window panels and light lenses. Basically, these surfaces are the portions of the surfaces that are trimmed away when you cut the window and light openings. To make these surfaces, you will copy the trimmed surfaces, copy the trimmed edges of the openings, change the trimmed surfaces back to their base surfaces, and use the trimmed edges to trim the base surfaces.

Now use the COPY command to make a copy of the side surface, and use the GROUP command to put this copied surface into an entity group called Side. Because the copied surface and the original surface reside in the same location, you will not see any change on your screen. Now use the AMVISIBLE command to hide all the surfaces except the one you put in the group called Side. (See Figure 2.300.)

<Construct> **<Copy>**

Command: **COPY**
Select objects: [**Select A (Figure 2.299).**]
Select objects: [**Enter**]
<Base point or displacement>/Multiple: **0,0**

Second point of displacement: **[Enter]**

<Construct> **<Group...>**

Command: **GROUP**

[Object Grouping
Group Name: **SIDE**
New<]

Select objects: [**Select A (Figure 2.299).**]
Select objects: [**Enter**]

[**OK**]

<Surface> **<Visibility...>**

Command: **AMVISIBLE**

[**Objects**
Hide **Select<**]

Select objects to hide: **ALL**
Select objects to hide: **REMOVE**
Remove objects: **G**
Enter group name: **SIDE**
Remove objects: [**Enter**]

[**OK**]

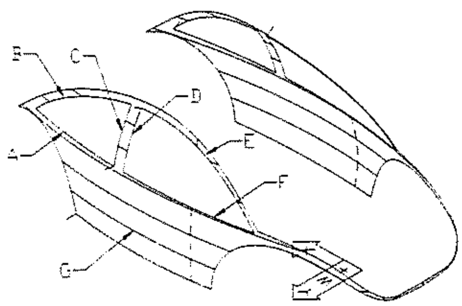


Figure 2.300 All surfaces except the copied side surface hidden

Copy the trimmed edges of the window openings of the side surface. Then untrim the side surface. (See Figure 2.301.)

<Surface> **<Create Wireframe>** **<Copy Edge>**

Command: **AMEDGE**
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **OUTPUT**
Polyline/<Spline>: [**Enter**]
(Output = Spline)
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **COPY**
Select surface edge: [**Select A, B, C, D, E, and F (Figure 2.300).**]

Select surface edge: **[Enter]**

<Surface> <Edit Trim Edges> <Untrim Surface>

Command: **AMEDGE**
(Output = Polyline)
Copy edge/Output/Show nodes/Untrim/<Extract loop>: **UNTRIM**
Select surfaces: **[Select G (Figure 2.300).]**
Select surfaces: **[Enter]**

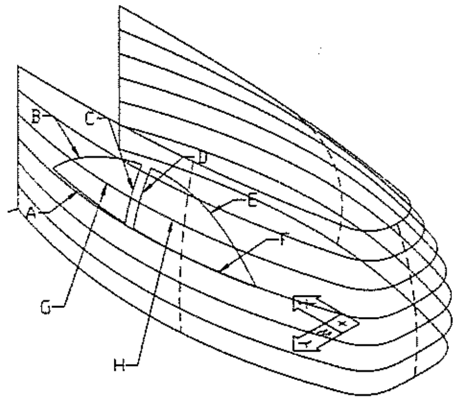


Figure 2.301 Trimmed edges copied, copied side surface untrimmed

Because there are two window panels on each side of the side surface, make a copy of the untrimmed surface. Then use the copied trimmed edges to trim the original surface and the copied surface. (See Figure 2.302.)

<Construct> <Copy>

Command: **COPY**
Select objects: **[Select G (Figure 2.301).]**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0**
Second point of displacement: **[Enter]**

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select A, B, and C (Figure 2.301).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select G (Figure 2.301).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

<Surface> <Edit Surface> <Project Trim...>

Command: **AMPROJECT**
Select wires to project: **[Select D, E, and F (Figure 2.301).]**
Select wires to project: **[Enter]**
Select target surfaces: **[Select H (Figure 2.301).]**
Select target surfaces: **[Enter]**

[Project to Surface
Direction: **Normal to surface**
Output Type: **Trim surface**
OK]

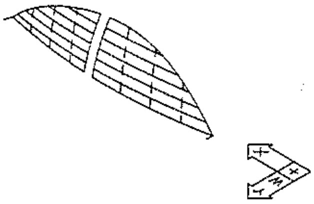


Figure 2.302 Window panels constructed

Use the AMVISIBLE command to unhide the hidden surfaces. Then use the MIRROR3D command to mirror the window panels. (See Figure 2.303.)

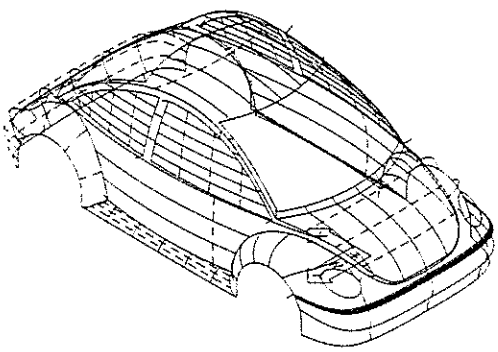


Figure 2.303 Surfaces unhidden, window panels mirrored

To reiterate, the process of making the window panels is: Copy the surface with openings cut in it, hide all other surfaces, copy the trimmed edges of the openings, untrim the copied surface, and use the copied edges to trim the untrimmed surface.

Now complete the front window panel, the rear window panel, and the light lenses by following a similar route. Figure 2.304 shows all the window panels and light lenses. After making all the window panels and light lenses, unhide all surfaces. (See Figure 2.305.)

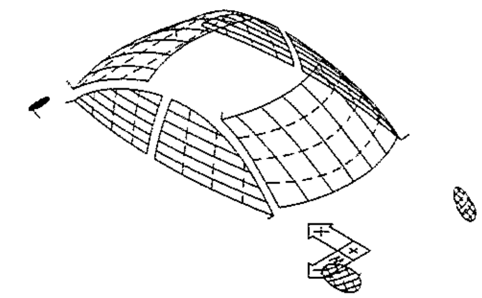


Figure 2.304 Window panels and light lenses constructed

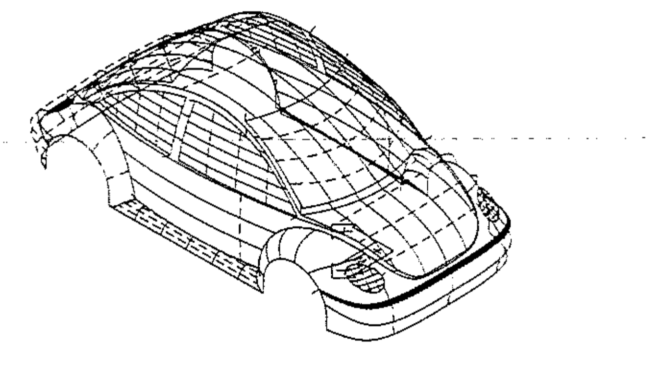


Figure 2.305 Complete car model

To reduce the memory size of the model, truncate all the trimmed surfaces. Then save your drawing.

<Surface> **<Edit Surface>** **<Truncate>**

Command: **AMEDITSF**
Select surfaces: **ALL**
Select surfaces: [**Enter**]
Direction/Preview/Span/Truncate/U grips/V grips/<eXit>: **TRUNCATE**
Direction/Preview/Span/Truncate/U grips/V grips/<eXit>: **EXIT**

<File> **<Save>**

The surface model is now complete. In Chapter 3, you will construct solid parts for this model car. In Chapter 4, you will assemble the solid parts. To add the assembled solid parts to this surface model, you can use the AMCATALOG command (which you will learn in Chapter 4) to attach the drawing Car.dwg. After attaching the drawing, you will use the MOVE command to position the solid parts as shown in Figure 2.306.

<Construct> **<Move>**

Command: **MOVE**
Select objects: [**Select the assembly of the solid parts.**]
Select objects: [**Enter**]

Base point or displacement: **MID** of [**Select A (Figure 2.306).**]
Second point of displacement: **-28,0,-10**

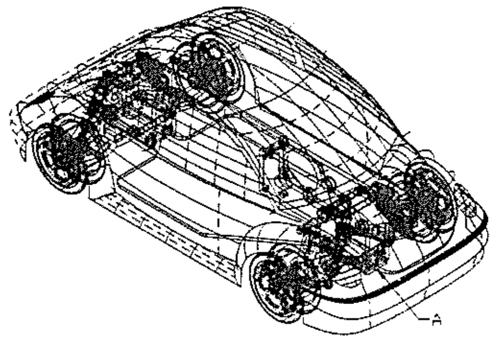


Figure 2.306 Surface model assembled with solid parts constructed later chapters

In making this surface model, you learned various techniques for copying trimmed edges, offsetting wires, editing wires, and using wires to construct and edit surfaces. While constructing the windows and lenses, you learned how to construct matching surfaces at the trimmed edges. To reiterate, you can offset a polyline or a 3D polyline, but not a 3D spline. Therefore, you have to unspline a spline or output a polyline to construct offset wires. Because a polyline has a number of line segments and breaking or trimming is laborious, you have to fit polylines to splines in order to facilitate editing.

2.12 Key Points and Exercises

Mechanical Desktop has four sets of engineering design tools: surface modeling tools, parametric solid modeling tools, assembly modeling tools, and associative drafting tools. In this chapter, you learned the key concepts of surface modeling and the techniques for using surface modeling tools to produce 3D surface models. You also learned ways to apply various utility tools and ways to incorporate a free-form surface in a native solid model. In the next chapter, you will learn how to construct parametric solids. After that, you will learn how to incorporate free-form surfaces into a parametric solid.

Using Mechanical Desktop, you can construct five kinds of surfaces: primitive surfaces, free-form surfaces, derived surfaces, trimmed surfaces, and converted surfaces. Primitive surfaces are surfaces of regular geometric shapes that you can construct by specifying their types, dimensions, and locations. They include the cylindrical surface, the spherical surface, the toroidal surface, and the planar surface. To make cylindrical and toroidal surfaces together, you can construct a tubular surface.

Free-form surfaces are the kind that you use most in making 3D surface models. They are the revolve surface, the extrude surface, the rule surface, the loft u surface, the sweep surface, and the loft uv surface. To make free-form surfaces, you have to define a set of wires. As you have learned in this chapter, making free-form surfaces is simple after the wires are constructed. However, making the wires is a tedious job.

A 3D surface model usually consists of a number of surfaces put together. To treat the joints among the surfaces, you can use the fillet surface, the corner surface, and the blend surface. These three kinds of surfaces, together with the offset surface, are called derived surfaces.

Smooth surfaces are constructed from smooth wires. However, most smooth surfaces that you use to form a 3D model have irregular boundary edges. To make a smooth surface with irregular edges, you build a primitive, free-form, or derived surface, and then use a wire to trim away the unwanted portions. The resulting surface is called a trimmed surface.

Some AutoCAD objects, such as 2D objects with thickness and native solids, and Mechanical Desktop solids can be converted to sets of surfaces. Using an existing object that you already have as a starting point for further developing a 3D surface model can save you a lot of time.

There are many ways to edit a surface. You can break a surface into two pieces and yet retain their original profile and silhouette; join two or more surfaces at their untrimmed edges to form a single smooth surface; lengthen a surface at its untrimmed edge; scale the size of a surface up or down; intersect and trim two surfaces; untrim a trimmed surface by removing the boundary edges; truncate a surface by reducing the size of the base surface to a minimum that is barely large enough to define the profile and silhouette of the trimmed surface; refine the uv patches of a surface to increase or decrease its accuracy in relation to the defining wires; set the number of control grip points where you can pull and stretch a surface to deform it; set the size of a circular area that is deformed when a control grip point is manipulated; and change the normal direction of a surface.

As explained earlier, you need to construct 3D wires in order to construct free-form surfaces. To enable you to build intricate 3D wires, 3D wire construction and editing tools are available for you to construct augmented lines; join wires to form augmented lines, polylines, or splines; fit wires to form splines; change a spline back to a polyline; copy the edges of surfaces; produce flow lines from surfaces; generate a parting line on a surface model; cut a series of cross section lines on a surface model; construct a wire at the intersection of two surfaces; project a wire onto a surface to obtain another wire; offset a wire in 3D view; fillet wires in 3D; change the direction of a wire; and refine the number of control points of a wire.

Surfaces can be incorporated into AutoCAD native solids and Mechanical Desktop parametric solids. You can treat the surface as a knife and the solid as a cake. To make a solid with a free-form surface, you use the knife to cut the cake. In preparing the surface, you have to ensure that all the boundary edges of the surface lie outside the solid. While cutting the solid, you have to decide which side of the solid is to be removed. In this chapter, you cut a native solid. In the next chapter, you will cut a parametric solid.

Every time you start to construct a surface model, remember to set the related system variables, because these settings affect the way the surfaces are constructed and the accuracy of the surfaces in relation to the inputting wires.

Now enhance your knowledge by working on the exercises that follow.

Exercise 2.1

Give a brief account of the types of surfaces that you can construct using Mechanical Desktop. Use simple sketches to illustrate the various kinds of primitive surfaces, free-form surfaces, derived surfaces, and trimmed surfaces. What kind of AutoCAD and Mechanical Desktop objects can you convert to surfaces? What are the editing processes that you can apply on a surface?

Exercise 2.2

In making the coil spring project in this chapter, you used an augmented line as a rail. Explain, with the aid of two other examples, how you can use augmented lines in constructing a surface model.

Exercise 2.3

What is the major difference between joining wires to form a spline and fitting wires to form splines? How does the direction of a wire affect a surface constructed from it?

Exercise 2.4

Figure 2.307 shows the rendered image of an oil cooler. Obviously, this oil cooler is a series of cylindrical and toroidal surfaces joined together. To make this surface model, you can construct a 3D polyline as the control wire and then use the AMTUBE command to construct the surfaces.

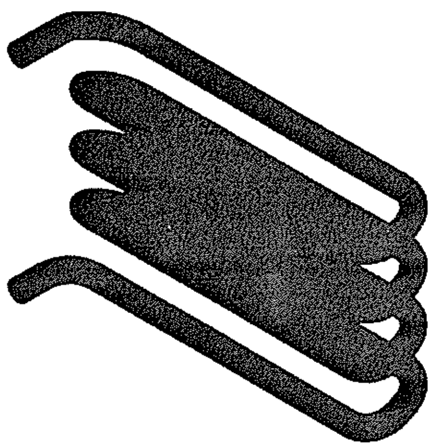


Figure 2.307 Rendered image of an oil cooler

Start a new drawing by using the template Surf_mdl.dwt that you saved in this chapter. Set the current layer to Wire and set the display to an isometric view. Then select the 3D Polyline item of the Design pull-down menu to use the 3DPOLY command to construct a wire. (See Figure 2.308.)

<Design> <3D Polyline>

Command: **3DPOLY**
From point: **0,0**
Close/Undo/<Endpoint of line>: **@50<90**
Close/Undo/<Endpoint of line>: **@300<0**
Close/Undo/<Endpoint of line>: **@0,50,25**
Close/Undo/<Endpoint of line>: **@300<180**
Close/Undo/<Endpoint of line>: **@0,-50,25**
Close/Undo/<Endpoint of line>: **@300<0**
Close/Undo/<Endpoint of line>: **@0,50,25**
Close/Undo/<Endpoint of line>: **@300<180**
Close/Undo/<Endpoint of line>: **@0,-50,25**
Close/Undo/<Endpoint of line>: **@300<0**
Close/Undo/<Endpoint of line>: **@0,50,25**
Close/Undo/<Endpoint of line>: **@300<180**
Close/Undo/<Endpoint of line>: **@0,-50,25**
Close/Undo/<Endpoint of line>: **@300<0**
Close/Undo/<Endpoint of line>: **@0,50,25**
Close/Undo/<Endpoint of line>: **@300<180**
Close/Undo/<Endpoint of line>: **@0,-50,25**
Close/Undo/<Endpoint of line>: **@50<270**
Close/Undo/<Endpoint of line>: **[Enter]**

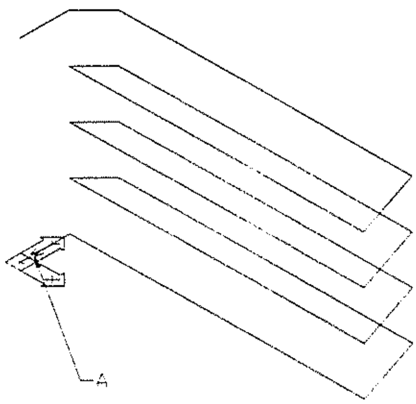


Figure 2.308 3D polyline constructed

Set the current layer to Surface. Then use the AMTUBE command. (See Figure 2.309.)

<Surface> <Create Surface> <Tubular>

Command: **AMTUBE**
Select wire: [**Select A (Figure 2.308).**]
Tube diameter: **20**
Automatic/<Manual>: **A**
Bend radius for all: **24**

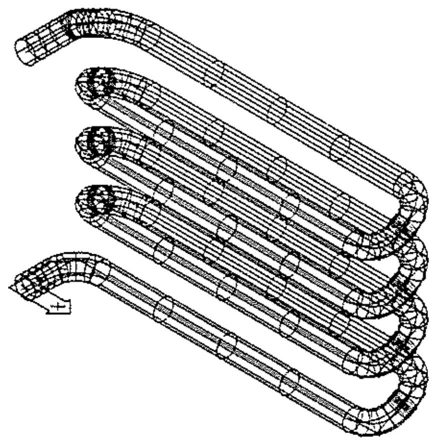


Figure 2.309 Surface model of the oil cooler

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Oilcool.dwg**

Exercise 2.5

Figure 2.310 shows the rendered image of the casing of a pager. Take some time to analyze the model to determine what surfaces are required and what wires are needed.

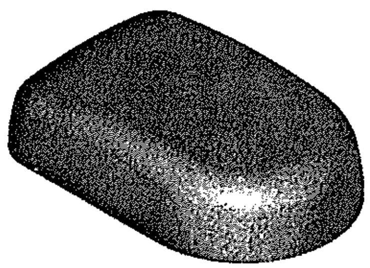


Figure 2.310 Rendered image of the pager

The surface model consists of side surfaces, a top surface, and fillet surfaces. Start a new drawing by using the template Surf_mdl.dwt. As shown in Figure 2.311, construct the wires for the rails of the sweep surfaces for the side surfaces.

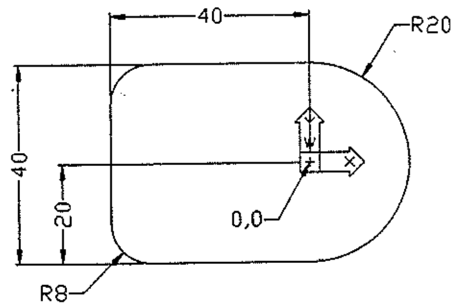


Figure 2.311 Wires for the rails of the sweep surfaces

Set the display to an isometric view. Then set the UCS to rotate 90° about the X axis. After that, construct a circle and trim it to an arc as shown in Figure 2.312.

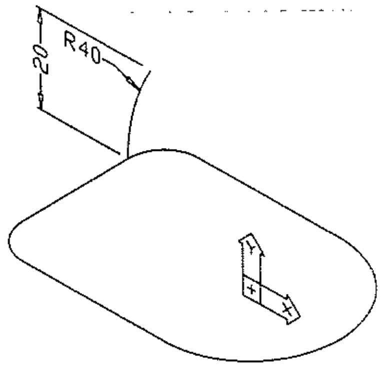


Figure 2.312 Wire for the cross section of the sweep surface

Set the current layer to Surface, then construct four sweep surfaces as shown in Figure 2.313.

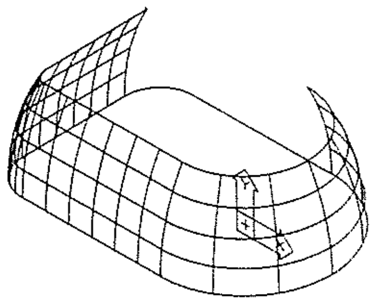


Figure 2.313 Four sweep surfaces constructed

Now construct two splines on layer Wire. (See Figure 2.314.)

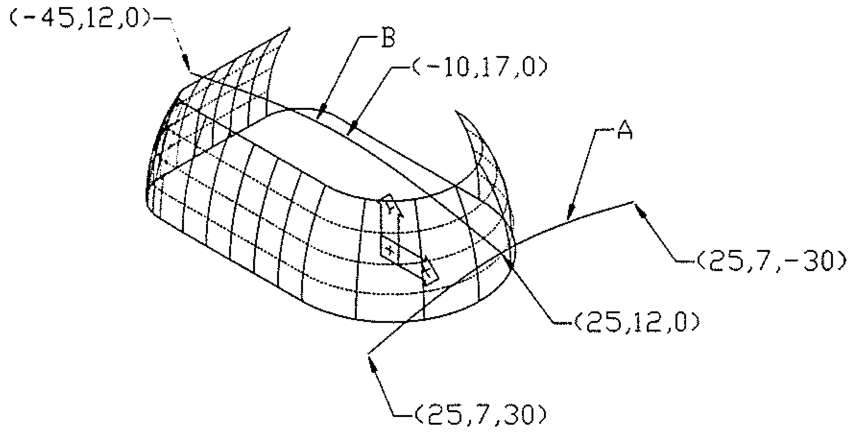


Figure 2.314 Wires for the top surfaces

Set the current layer to Surface. Then construct a sweep surface. (See Figure 2.315.)

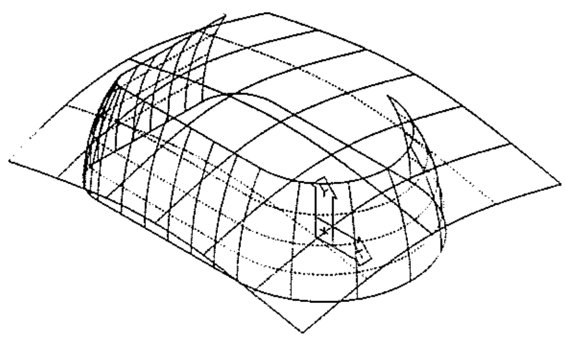


Figure 2.315 Sweep surface constructed on layer Surface

Turn off layer Wire. Then construct four fillet surfaces, A, B, C, and D, as shown in Figure 2.316. Fillets A and B are constant fillet surfaces of radius 5 units. Fillet C is a linear variable fillet with fillet radii of 5 units to 7 units. Fillet D is a constant fillet surface of radius 7 units. While filleting, trim the side surfaces but not the top surface.

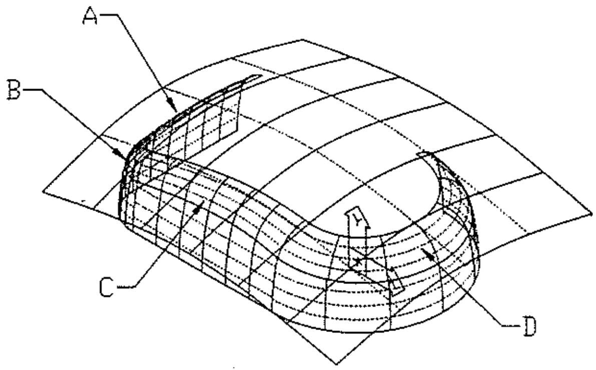


Figure 2.316 Fillet surfaces constructed

Mirror two fillet surfaces and two sweep surfaces as shown in Figure 2.317.

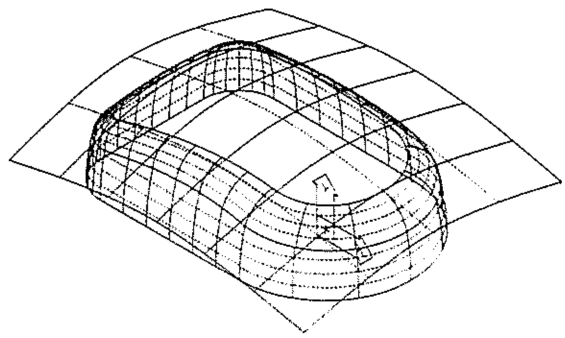


Figure 2.317 Fillet and sweep surfaces mirrored

Set the display to a top view by using the shortcut key [5]. Then copy the trimmed edges of the fillet surfaces. After that, hide all the surfaces except the top surface. (See Figure 2.318.)

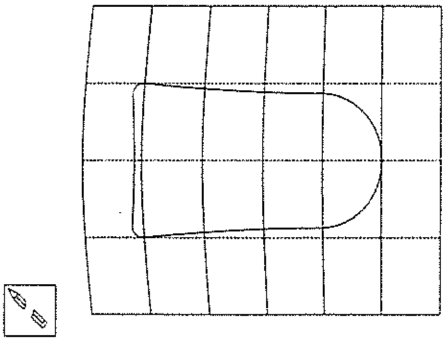


Figure 2.318 Trimmed edges copied, surfaces hidden

Use the copied edges to trim the top surface. Then unhide all the hidden surfaces. After that, set the display to an isometric view. (See Figure 2.319.)

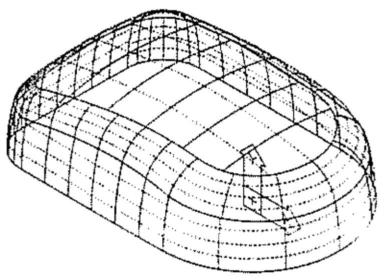


Figure 2.319 Top surface trimmed by copied edges, all surfaces unhidden

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Pager.dwg**

Exercise 2.6

Figure 2.320 shows the rendered image of the solid model of the axle of a scale model car. This solid model has three main features: a revolve solid, an extrude solid, and a helical sweep surface. You will construct a solid of extrusion as the main body, a solid of extrusion as the slot, and a helical sweep surface as the screw thread. To compose the solid model, you subtract the extrude solid from the revolve solid and then use the helical sweep surface to cut the solid. Figure 2.321 shows the solid of revolution, the solid of extrusion, and the helical surface exploded apart.

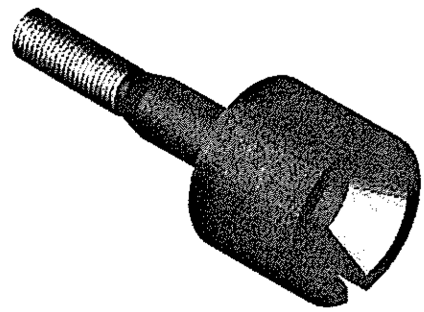


Figure 2.320 Rendered image of an axle

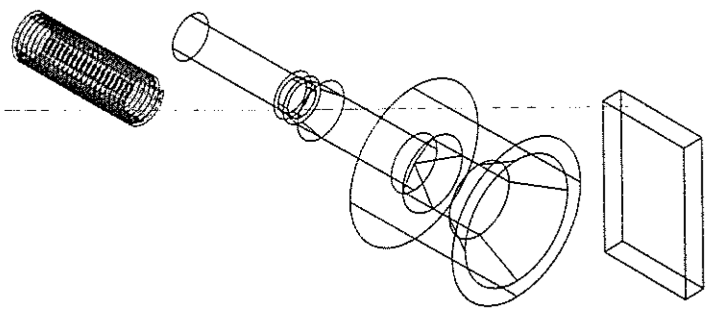


Figure 2.321 Helical sweep surface, revolve solid, and extrude solid exploded apart

Start a new drawing by using the template drawing Surfsol.dwt. As shown in Figure 2.322, construct a series of wires. While constructing the wires, do not include the dimensions. Then convert the wires into a region.

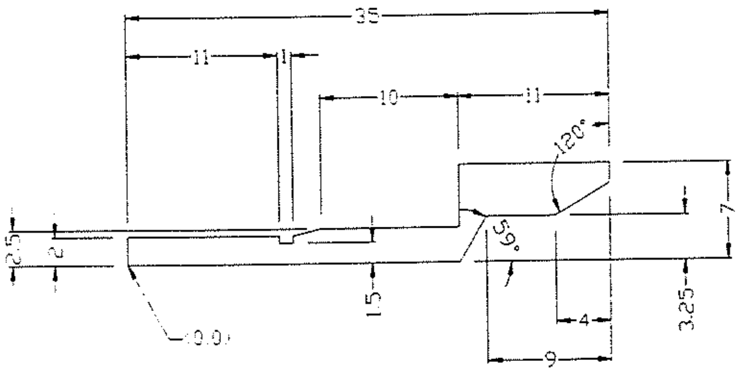
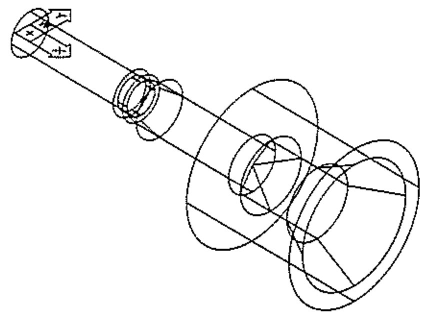


Figure 2.322 Dimensions for the main body

Set the current layer to Solid. Then use the REVOLVE command to revolve the region into a solid of revolution. Use the lower horizontal line as the axis of revolution. Then set the display to an isometric view. (See Figure 2.323.)



2.323 Region constructed and revolved

Construct a rectangle with corners at (26,–1,–7) and (35,1). Then extrude the rectangle to a height of 14 units. (See Figure 2.324.)

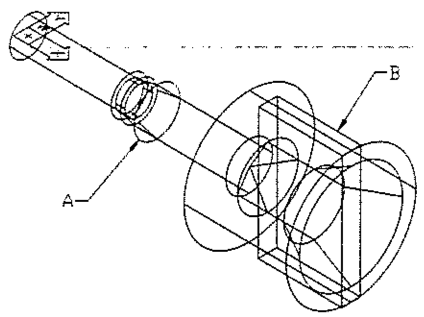


Figure 2.324 Rectangle constructed and extrude

To cut a slot, subtract the extrude solid B (Figure 2.324) from the revolve solid A (Figure 2.324). (See Figure 2.325.)

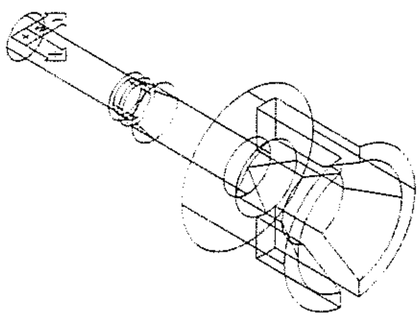


Figure 2.325 Extrude solid subtracted from revolve solid

Set the current layer to Wire and turn off layer Solid. Then use the shortcut key [9] to set the display to the top view. After that, construct a polyline and a line. (See Figure 2.326.)

<Design> <Polyline>

Command: **PLINE**
From point: **-0.48,2.1**
<Endpoint of line>: **-0.31,1.77**
<Endpoint of line>: **A**
<Endpoint of arc>: **-0.19,1.77**
<Endpoint of arc>: **L**
<Endpoint of line>: **-0.02,2.1**
<Endpoint of line>: **[Enter]**

<Design> <Line>

Command: **LINE**
From point: **-0.5,0**
To point: **@12<0**
To point: **[Enter]**

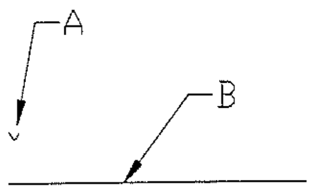


Figure 2.326 Polyline and line constructed

Set the display to a left isometric view. Then fit wire A (Figure 2.326) into a spline. You will use it as a cross section for making a sweep surface.

<View> <Model Views> <Left Isometric>

<Surface> <Edit Wireframe> <Spline Fit...>

Command: **AMFITSPLINE**
Select wires: **[Select A (Figure 2.326).]**
Select wires: **[Enter]**

[OK]

Because the pitch of the screw is 0.5 unit and the line is 12 units in length, the number of thread pitch is 24. If we use 12 control points for each pitch length, the total number of control points should be 288. Therefore, use the AMREFINE3D command to refine wire B (Figure 2.326) into 288 points.

<Surface> <Edit Wireframe> <Refine>

Command: **AMREFINE3D**
Select lines or polylines: **[Select B (Figure 2.326).]**
Select lines or polylines: **[Enter]**

Points/<Tolerance>: **P**
Points: **288**

Use the AMPREFS command to set the default augment vector length to 1.5 units. Then add augmented lines to the refined line.

<Surface> <Preferences...>

Command: **AMPREFS**
[Surfaces
Aug Vector Length: **1.5**
OK]

<Surface> <Edit Wireframe> <Augment Polyline>

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **ADD**
Select lines or polylines: **[Select B (Figure 2.326).]**
Select lines or polylines: **[Enter]**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **[Enter]**

As we have explained, there are 24 pitches. Because each pitch rotates 360°, the total twist angle of the augmented line is 8640°. After twisting the vectors, set the display to a left isometric view. (See Figure 2.327.)

<Surface> <Edit Wireframe> <Twist Vectors>

Command: **AMEDITAUG**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **TWIST**
Total angle: **8640**
Range/<All>: **ALL**
Select augmented line: **[Select B (Figure 2.326).]**
Select augmented line: **[Enter]**
Add vectors/Blend/Copy/Normal length/Rotate/Twist/<eXit>: **[Enter]**

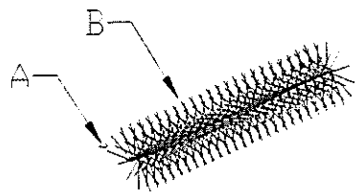


Figure 2.327 Spline fitted, augmented line edited

Now construct a sweep surface by using spline A (Figure 2.327) as the cross section and augmented line B (Figure 2.327) as the rail. (See Figure 2.328.)

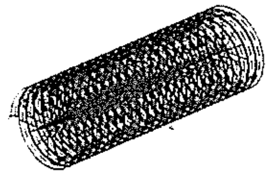


Figure 2.328 Helical sweep surface constructed

Turn on layer Solid and set the current layer to Solid. (See Figure 2.329.)

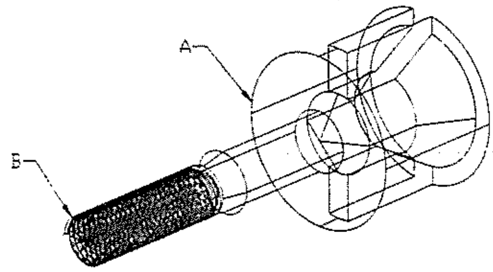


Figure 2.329 Layer Solid turned on

Now use the surface B (Figure 2.329) to cut the solid A (Figure 2.329). Then turn off layers Wire and Surface. (See Figure 2.330.)

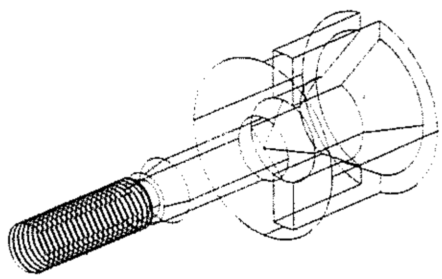


Figure 2.330 Completed model

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Axle.dwg**

Exercise 2.7

As we have said, car bodies involve many surfaces of different kinds. Constructing a model car body will help you learn more about surface modeling. Figure 2.331 shows the rendered image of the scale model car. Before you start to construct the model, take some time to think about what surfaces and wires are needed.

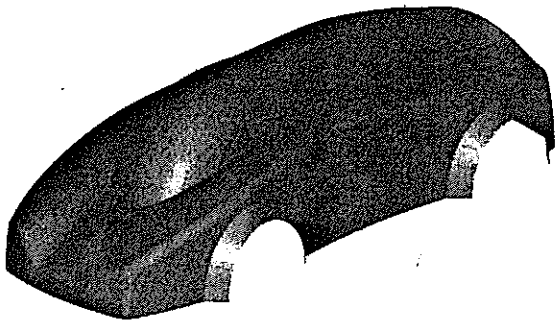


Figure 2.331 Rendered image of the scale model car

This model has three major features: the main body, the greenhouse, and the wheel flarings. The main body consists of a number of sweep, loft uv, fillet, and corner surfaces. The greenhouse is a sweep surface. The wheel flarings are rule surfaces. In this exercise, you will learn how to treat the corners of three intersecting fillet surfaces.

Start a new drawing. Use the template drawing Surf_mdl.dwt. Construct four arcs on layer Wire as shown in Figure 2.332.

<Design> <Arc> <3 Points>

Center/<Start point>	Center/End/<Second point>	End point
−70,−50	40,−88	155,−80
270,−82	355,−95	440,−68
−20,115	−50,0	−20,−115
400,−125	420,0	400,125

Figure 2.332 Arcs constructed

As shown in Figure 2.333, construct a fillet B (Figure 2.333) with a radius of 580 units. Then join arcs A, B, and C (Figure 2.333) to form a spline.

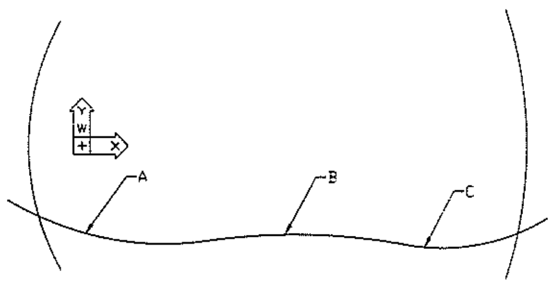


Figure 2.333 Fillet constructed, arcs joined into a spline

Set the display to an isometric view. Then rotate the UCS 90° about the X axis. After that, construct two arcs as shown in Figure 2.334.

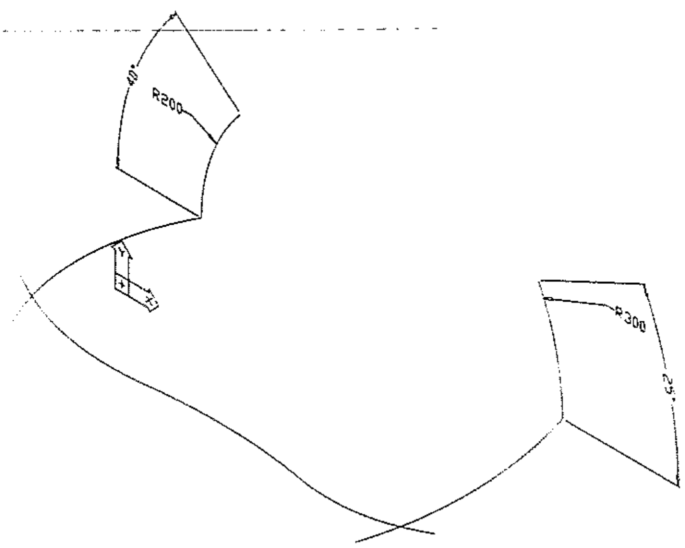


Figure 2.334 UCS rotated, two arcs constructed

Rotate the UCS 90° about the Y axis. Then construct an arc as shown in Figure 2.335.

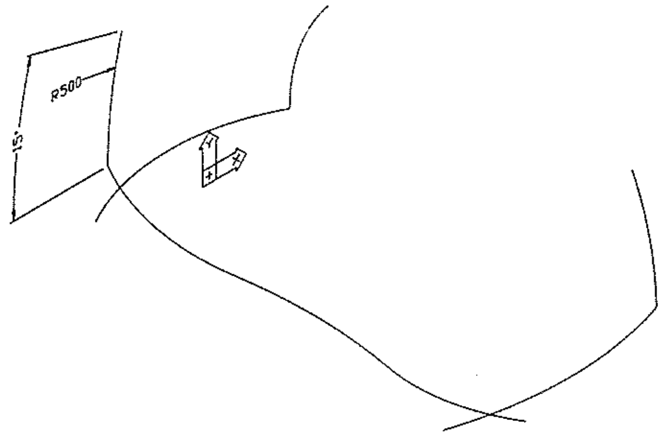


Figure 2.335 UCS rotated, arc constructed

Now the wires for the sides of the main body are complete. Set the current layer to Surface. Then construct three sweep surfaces and mirror a sweep surface. Sweeping orientation is parallel. (See Figure 2.336.)

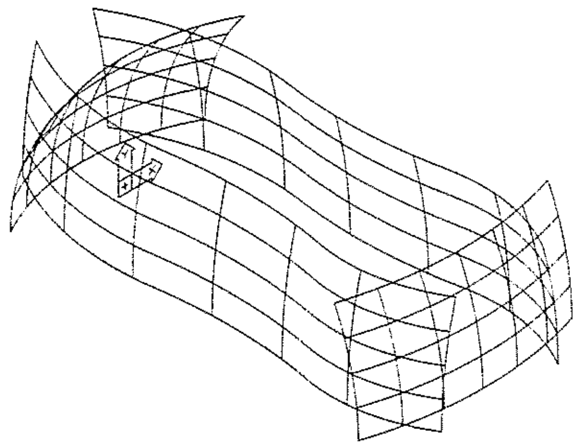


Figure 2.336 Sweep surfaces constructed and mirrored

Construct four constant-radius fillet surfaces as shown in Figure 2.337. The fillet radius is 10 units.

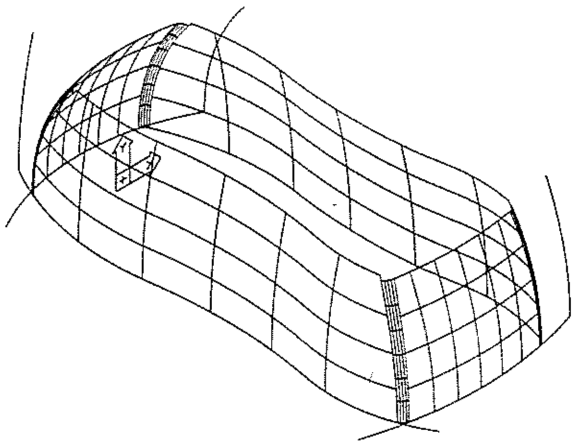


Figure 2.337 Fillet surfaces constructed

Hide all the surfaces and wires. Then set the UCS to World and set the current layer to Wire. After that, construct the following splines. (See Figure 2.338.)

The first spline passes through the following points:

(–60,–110,24), (97,–110,84), (270,–110,82), and (440,–110,56).

The second spline passes through the following points:

(–60,0,13), (94,0,80), (268,0,93), and (440,0,83).

The third spline passes through the following points:

(–60,110,24), (97,110,84), (270,110,82), and (440,110,56).

The fourth spline passes through the following points:

(–60,–110,24), (–60,–40,24), (–60,–25,13), (–60,0,13), (–60,25,13), (–60,40,24), and (–60,110,24).

The fifth spline passes through the following points:

(97,–110,84), (94,0,80), and (97,110,84).

The sixth spline passes through the following points:

(270,–110,82), (268,0,93), and (270,110,82).

The seventh spline passes through the following points:

(440,–110,56), (440,–70,61), (440,–30,78), (440,0,83), (440,30,78), (440,70,61), and (440,110,56).

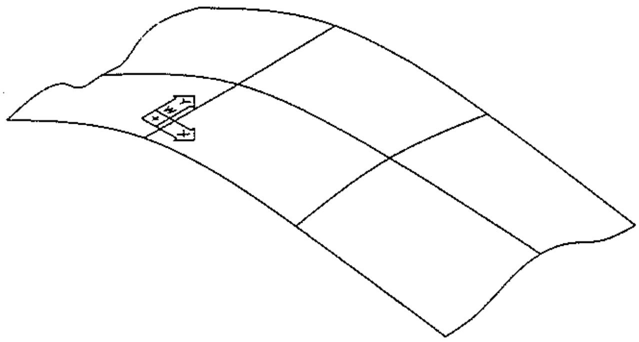


Figure 2.338 Surfaces hidden, splines constructed

Set the current layer to Surface and construct a loft uv surface. Then unhide all the hidden surfaces. (See Figure 2.339.)

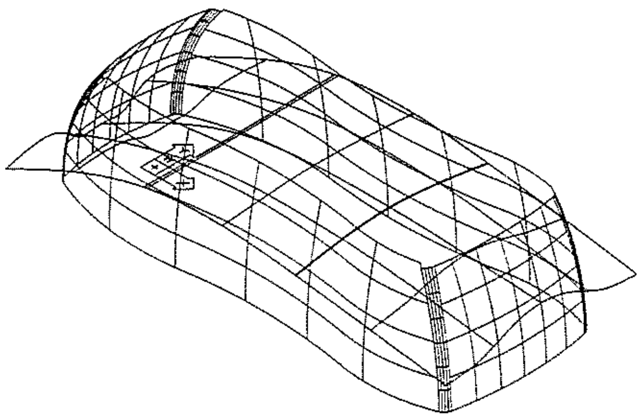


Figure 2.339 Loft uv surface constructed, surfaces unhidden

Construct four fillet surfaces with a constant-fillet radius of 10 units as shown in Figure 2.340. The fillet type is [Extended] and [Base surface].

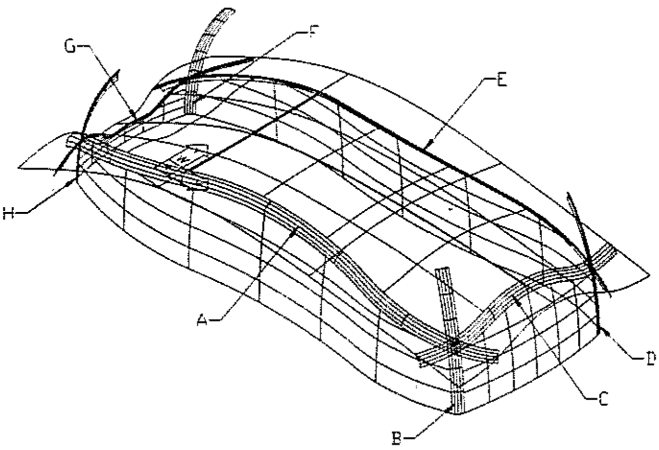


Figure 2.340 Fillet surfaces constructed

Use the AMVISIBLE command to hide the splines. Then edit the intersecting fillet surfaces by using the AMCORNER command. While making the corner fillet surfaces, you may have to change the direction of viewing in order to select the fillet surfaces. (See Figure 2.341.)

<Surface> **<Create Surface>** **<Corner Fillet>**

Command: **AMCORNER**

<Select first fillet surface>:	<Select second fillet surface>:	Select third fillet surface>:
Select A (Figure 2.340).	**Select B (Figure 2.340).**	**Select C (Figure 2.340).**
Select C (Figure 2.340).	**Select D (Figure 2.340).**	**Select E (Figure 2.340).**
Select E (Figure 2.340).	**Select F (Figure 2.340).**	**Select G (Figure 2.340).**
Select G (Figure 2.340).	**Select H (Figure 2.340).**	**Select A (Figure 2.340).**

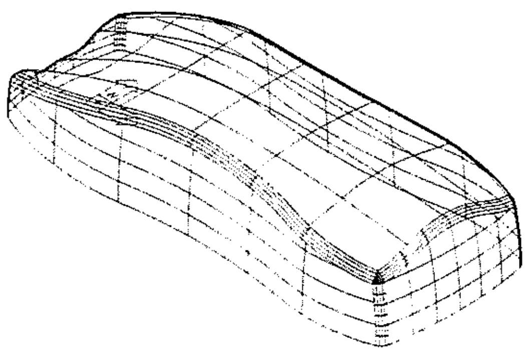


Figure 2.341 Splines hidden, corner fillet surfaces constructed

The main body of the model car is complete. Hide all the surfaces. Now work on the wheel flarings, which are rule surfaces. Rotate the UCS 90° about the X axis. Then set the current layer to Wire and construct two ellipses and a horizontal construction line. (See Figure 2.342.)

<Design> <Ellipse> <Center>

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **80,0**
Axis endpoint: **@55<0**
<Other axis distance>/Rotation: **64**

<Design> <Ellipse> <Center>

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **80,0**
Axis endpoint: **@42<0**
<Other axis distance>/Rotation: **53**

<Design> <Construction Line>

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **H**
Through point: **0,0**
Through point: **[Enter]**

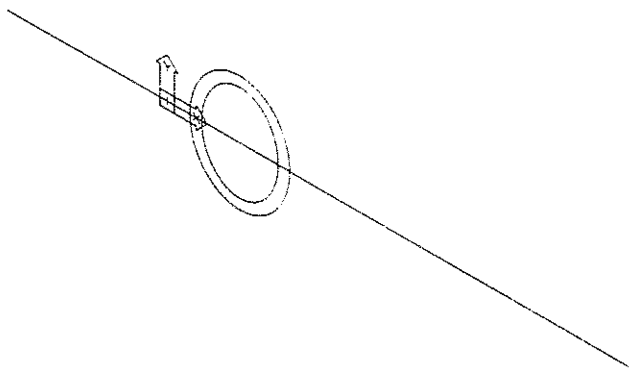


Figure 2.342 Surfaces hidden, ellipses and construction lines constructed

Use the construction line as the cutting boundary and trim the ellipses. Then copy the trimmed ellipses for a distance of 260 units in the X direction. After that, erase the construction line and unhide the two sweep surfaces. (See Figure 2.343.)

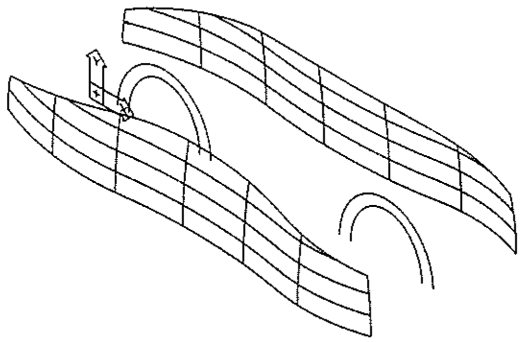


Figure 2.343 Ellipses trimmed and copied, construction line erased, and sweep surfaces unhidden

As shown in Figure 2.344, erase one of the sweep surfaces and project the two smaller elliptical wires on the surface in a direction normal to the current UCS to obtain two wires.

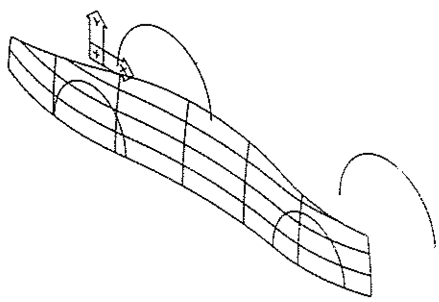


Figure 2.344 Sweep surface erased, elliptical wires projected

Move the two projected wires a distance of 13 units in the Z direction. Then project the two larger elliptical wires in a direction normal to the current UCS to trim the surface. (See Figure 2.345.)

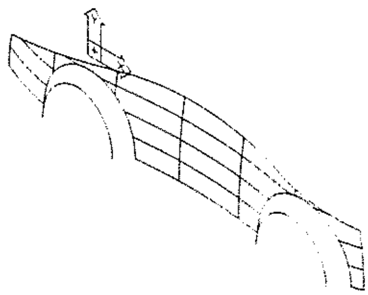


Figure 2.345 Wires moved, surface trimmed by projected elliptical wires

Set the current layer to Surface. Then construct two rule surfaces. (See Figure 2.346.)

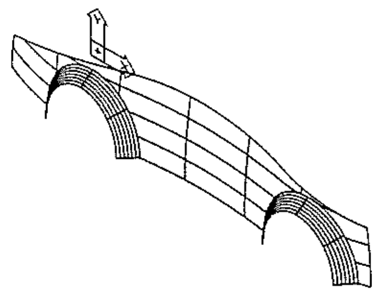


Figure 2.346 Rule surfaces constructed

One side of the car is complete. Use the MIRROR3D command to construct mirror copies of the side surfaces (including the rule surfaces). (See Figure 2.347.)

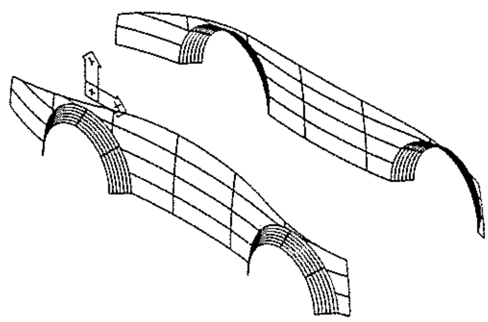


Figure 2.347 Side surfaces mirrored

The main body of the car is complete. Hide all the surfaces and wires. Now work on the greenhouse of the model car. Set the UCS to World. Then set the current layer to Wire. After that, construct an ellipse and a construction line. (See Figure 2.348.)

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: **50,0**
Axis endpoint: **@50<0**
<Other axis distance>/Rotation: **48**

<Design> **<Construction Line>**

Command: **XLINE**
Hor/Ver/Ang/Bisect/Offset/<From point>: **V**
Through point: **50,0**
Through point: **[Enter]**

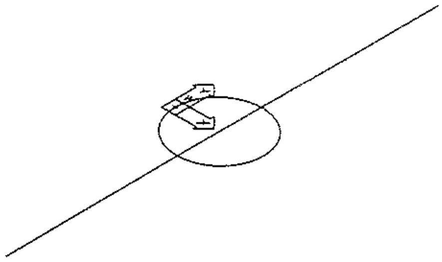


Figure 2.348 Surfaces hidden, ellipse and construction line constructed

Use the construction line as a cutting edge to trim the ellipse. Then erase the construction line and construct a spline to pass through the points (0,0,0), (72,0,113), (210,0,120), and (475,0,20). (See Figure 2.349.)

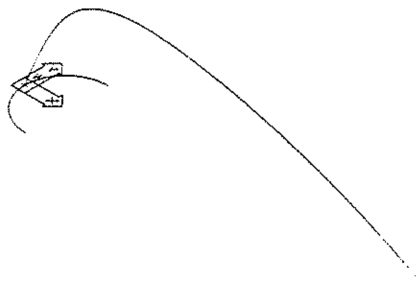


Figure 2.349 Ellipse trimmed, construction line erased, and spline constructed

Set the current layer to Surface. Then construct a sweep surface. Sweeping orientation is normal. Unhide the top surface. (See Figure 2.350.)

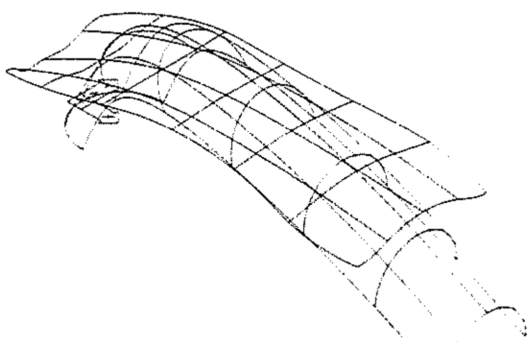


Figure 2.350 Sweep surface constructed, top surface unhidden

Intersect the two surfaces as shown in Figure 2.351. Then hide the wires.

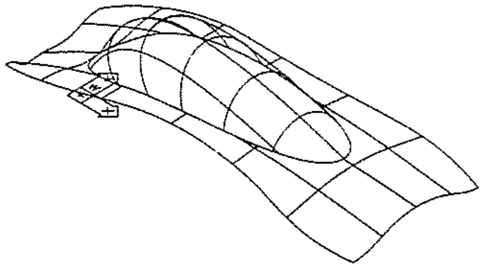


Figure 2.351 Surfaces intersected, wires hidden

Hide all the wires and unhide all the surfaces. (See Figure 2.352.)

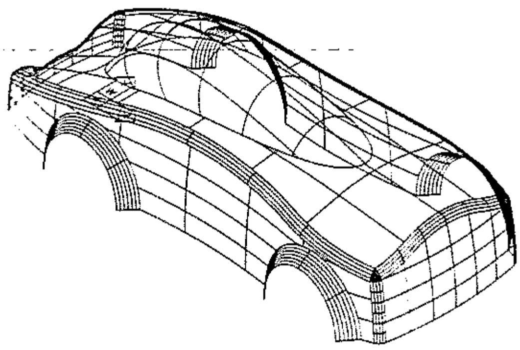


Figure 2.352 Wires hidden, surfaces unhidden

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Sport.dwg**

Exercise 2.8

Now work on another model car body to further enhance your knowledge of construction and editing of surfaces and wires. Figure 2.353 shows the rendered image of the body of a 1/10 scale model car. Now take some time to analyze the model, to think about what surfaces are required and what wires are needed for making the surfaces. (See Figure 2.354.)

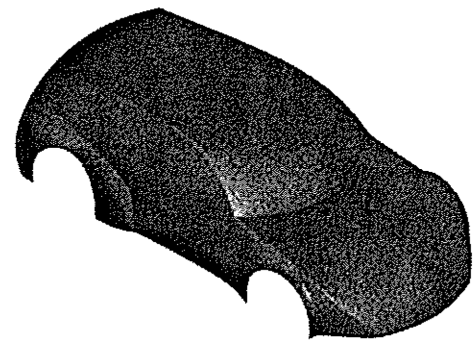


Figure 2.353 Rendered image of the body of a scale model car

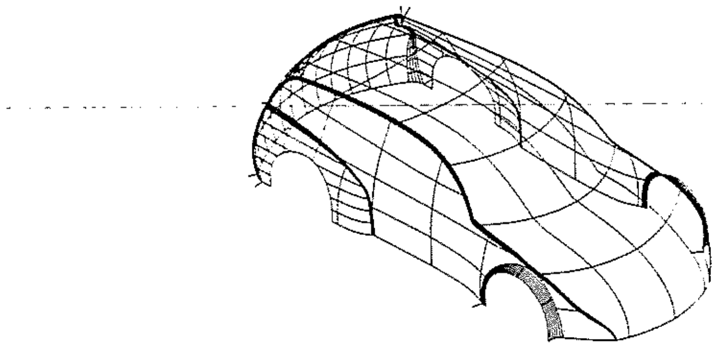


Figure 2.354 Surface model of the model car body

To let you see the various surfaces of the car clearly, the fillet surfaces are removed and the surfaces are exploded apart. (See Figure 2.355.)

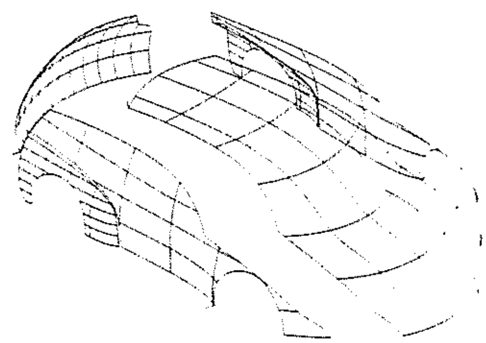


Figure 2.355 Exploded view of the trimmed surfaces

As shown in Figure 2.355, the main surfaces of the model car are trimmed surfaces. Before making the trimmed surfaces, you have to construct the base surfaces. (See Figure 2.356.)

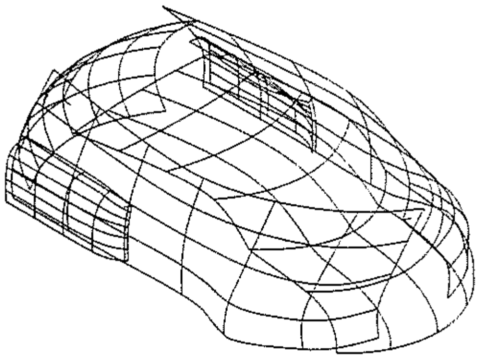


Figure 2.356 Exploded view of the base surfaces

You will start working on the top of the car. It consists of four surfaces joined together. To make this part of the model, construct a set of splines to define the general profile and silhouette. Then construct two loft u surfaces. Next, mirror a loft u surface and join the two surfaces together to form the hood of the car. To produce a smooth transition between the hood and the top of the car, blend the top and the hood surfaces together. Finally, join the hood surface, the blended surface, and the top surface to form a single surface. Figure 2.357 shows the wires, joined surface, and trimmed surface of the top of the car.

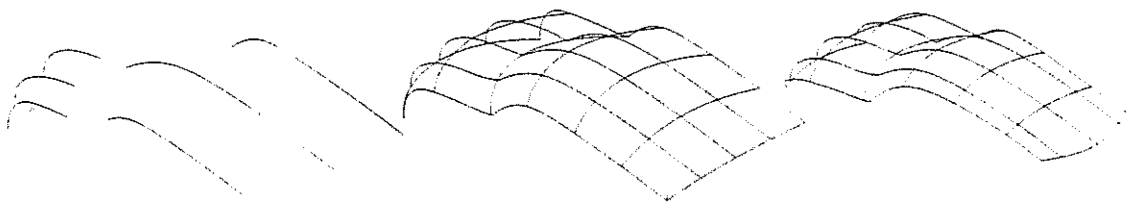


Figure 2.357 Wires, base surface, and trimmed surface of the top of the car

The rear part of the car is a loft u surface that is constructed from three splines. The joint between this part and the top part of the car is a variable fillet surface that starts from a fillet radius of 30 units at one edge and changes through a radius of 15 units at the midpoint to a radius of 30 units at the other edge. Because the AMFILLETSF command does not allow a fillet radius to change in this way, break the loft u surface into two surfaces. Then construct a cubical variable fillet surface of radius 30 units to radius 15 units for one surface and another cubical variable fillet surface of radius 15 units to radius 30 units for the other surface. Figure 2.358 shows the wires, base surfaces, and trimmed surfaces of the rear part of the car.

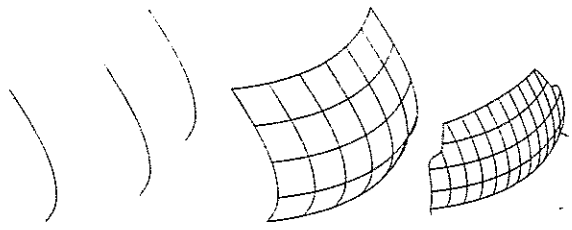


Figure 2.358 Wires, base surface, and trimmed surface of the rear of the car

Because the model car is symmetrical about its longitudinal plane, the two sides of the car are alike, so you will construct one side of the car and then make a mirror copy for the other side. The side of the car has two surfaces. Figure 2.359 shows the wires, base surfaces, and trimmed surfaces of one side of the car.

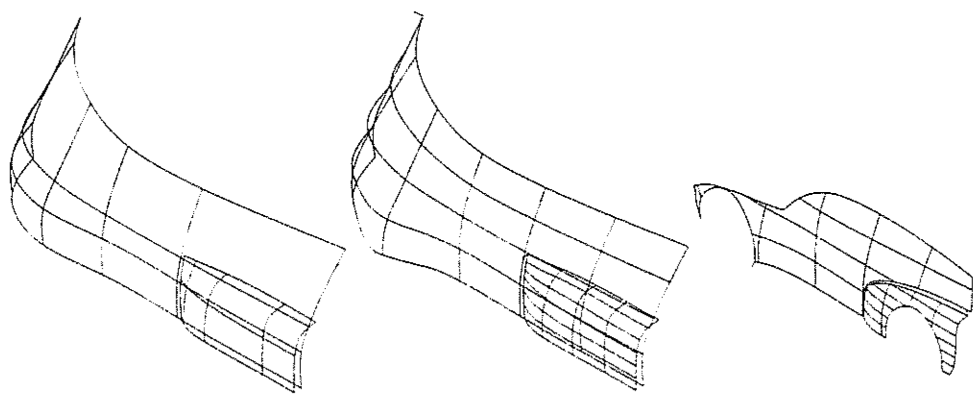


Figure 2.359 Wires, base surfaces, and trimmed surfaces of the side of the car

Start a new drawing and use the template drawing Surf_mdl.dwt. Use the AMPREFS command to set the approximate model size to 3000 mm and the vector length to 10 units.

You will start to work on the top part of the car. Use the UCS command to rotate the UCS 90° about the X axis. Then use the shortcut key [6] to set the display to the plan view of the new UCS. After that, use the SPLINE command to construct five splines for making the hood and top surfaces. (See Figure 2.360.)

<Design> **<Spline>**

Command: **SPLINE**

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
-94,-12	**-92,0**	**-64,32**	**-31,45**	**60,55**	[Enter]	[Enter]	[Enter]
-81,-12	**-80,0**	**-52,32**	**-18,45**	**80,55**	[Enter]	[Enter]	[Enter]
-46,-12	**-46,0**	**-30,32**	**5,45**	**105,55**	[Enter]	[Enter]	[Enter]

First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
10,40	85,90	190,100	340,80	[Enter]	[Enter]	[Enter]
70,40	125,90	190,95	340,75	[Enter]	[Enter]	[Enter]

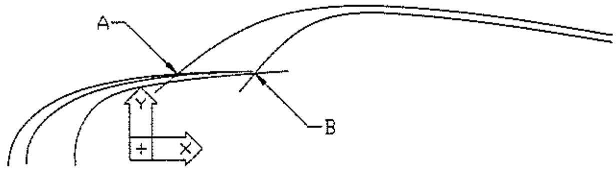


Figure 2.360 UCS rotated about the X axis, display set to front view, and splines constructed

Construct a line from intersection A (Figure 2.360) to intersection B (Figure 2.360). Then lengthen the line at each end by 200 units (a total of 400 units). After that, offset the lengthened line a distance of 3 units on both sides of the line. (See Figure 2.361.)

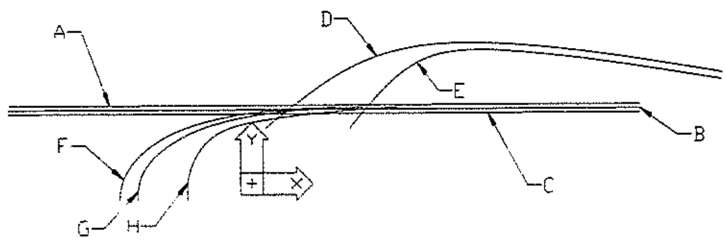


Figure 2.361 Line drawn and lengthened, offset lines constructed

Using A and C (Figure 2.361) as cutting edges, trim the splines D, E, F, G, and H (Figure 2.361). After trimming, erase the lines A, B, and C (Figure 2.361). (See Figure 2.362.)

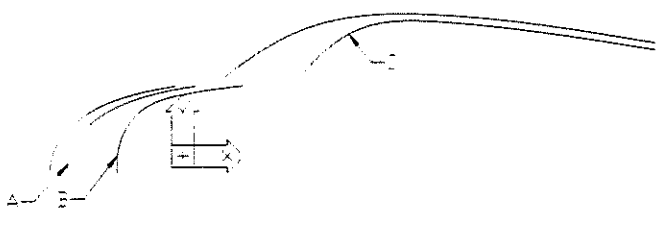


Figure 2.362 Splines trimmed, lines erased

Move splines B and C (Figure 2.362) a distance of 100 units in the Z direction, and move spline A (Figure 2.362) a distance of 55 units in the Z direction. Copy spline C (Figure 2.362) a distance of 200 units in the minus Z direction. Then set the display to an isometric view. (See Figure 2.363.)

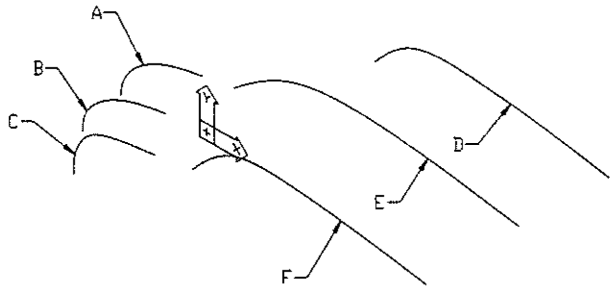


Figure 2.363 Display set to an isometric view, spline copied, and splines moved

Set the current layer to Surface. Then use wires A, B, and C (Figure 2.363) to construct a loft u surface, and use wires D, E, and F (Figure 2.363) to construct another loft u surface. (See Figure 2.364.)

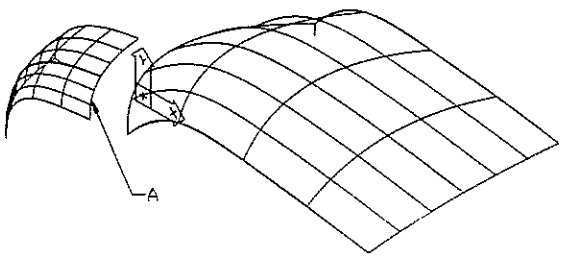


Figure 2.364 Loft u surfaces constructed

Use the MIRROR3D command to make a mirror copy of surface A (Figure 2.364). (See Figure 2.365.)

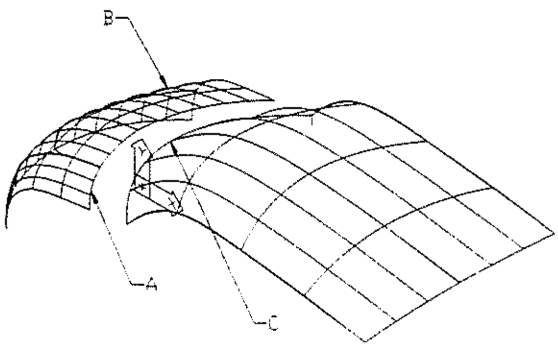


Figure 2.365 Surface mirrored

Hide all the splines. Then join surfaces A and B (Figure 2.365) into a single surface. After that, construct a blend surface between edges A and C (Figure 2.365). (See Figure 2.366.)

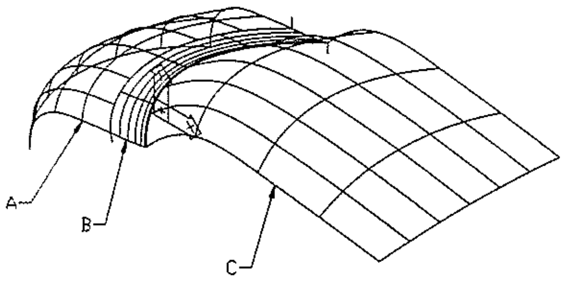


Figure 2.366 Loft u surfaces joined, blended surface constructed

Join surfaces A, B, and C (Figure 2.366) into a single surface. (See Figure 2.367.)

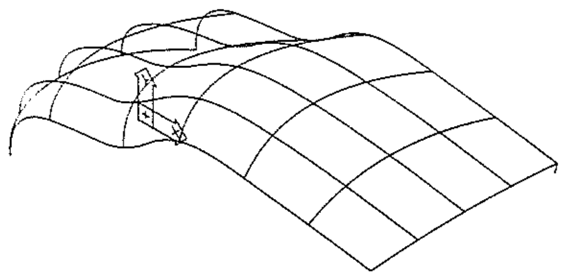


Figure 2.367 Surfaces joined into a single surface

The base surface for the top part of the model car is complete. Now work on the rear part of the car. Later, you will trim the top surface by making fillet surfaces between it and the rear and side surfaces.

Set the current layer to Wire. Then use the SPLINE command to construct a spline, and use the COPY command to make two copies. (See Figure 2.368.)

<Design> **<Spline>**

Command: **SPLINE**
Object/<Enter first point>: **340,-30**
Enter point: **355,0**
Close/Fit Tolerance/<Enter point>: **355,25**
Close/Fit Tolerance/<Enter point>: **290,110**
Close/Fit Tolerance/<Enter point>: [**Enter**]
Enter start tangent: [**Enter**]
Enter end tangent: [**Enter**]

<Construct> **<Copy>**

Command: **COPY**
Select objects: **LAST**
Select objects: [**Enter**]
<Base point or displacement>/Multiple: **-35,0,100**
Second point of displacement: [**Enter**]

Command: **[Enter]**
COPY
Select objects: **LAST**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0,-200**
Second point of displacement: **[Enter]**

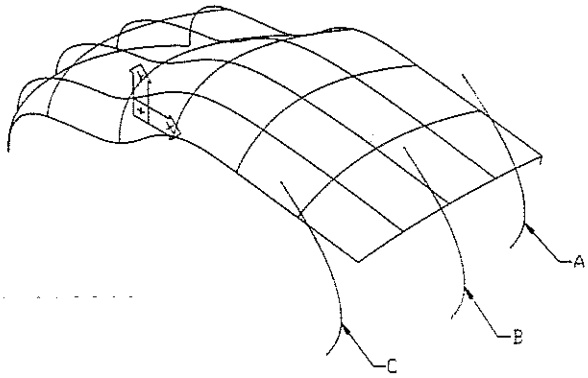


Figure 2.368 Spline constructed and copied

Set the current layer to Surface. Then use splines A, B, and C (Figure 2.368) to construct a loft u surface. (See Figure 2.369.)

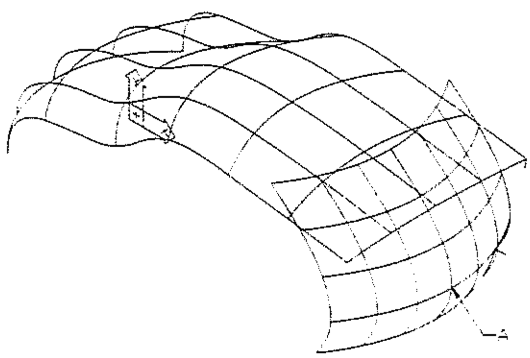


Figure 2.369 Loft u surface constructed

Hide the splines. Then break surface A (Figure 2.369) into two surfaces along vertical flow line A (Figure 2.369). (See Figure 2.370.)

<Surface> **<Edit Surface>** **<Break>**

Command: **AMBREAK**
Select surface: [**Select A (Figure 2.369).**]
(Percent = 50%)
Flip/Reposition/<Break u>: **[Enter]**

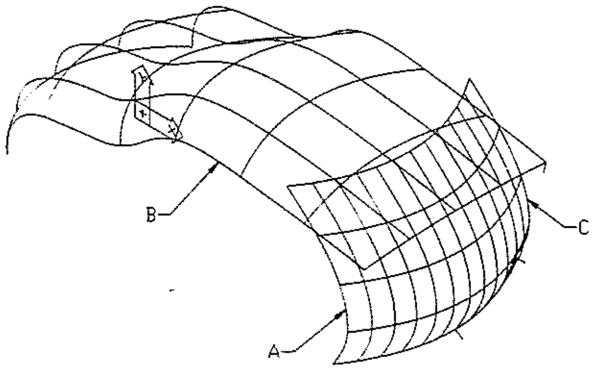


Figure 2.370 Loft u surface broken into two surfaces

Now you have three surfaces: A, B, and C (Figure 2.370). As shown in Figure 2.371, use the AMFILLETSF command to construct two cubical variable fillet surfaces between surfaces A and B (Figure 2.370), and between surfaces C and B (Figure 2.370). The fillet radii start from 30 units and change to 15 units. While filleting, trim surfaces A and C (Figure 2.370) but not surface B (Figure 2.370).

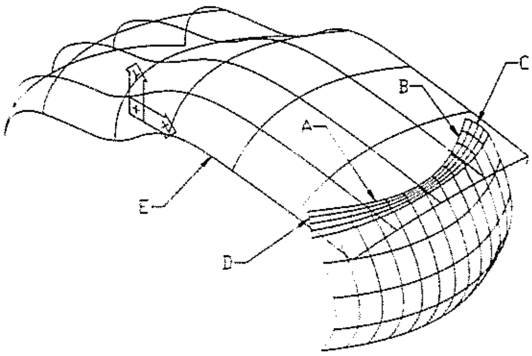


Figure 2.371 Variable fillet surfaces constructed

Now use the AMEDGE command to copy the trimmed edges at A and B (Figure 2.371). After copying, hide the surfaces C and D (Figure 2.371). Then use the copied edges A and B (Figure 2.371) to trim the surface E (Figure 2.371). After trimming, unhide the hidden surfaces C and D (Figure 2.371). (See Figure 2.372.)

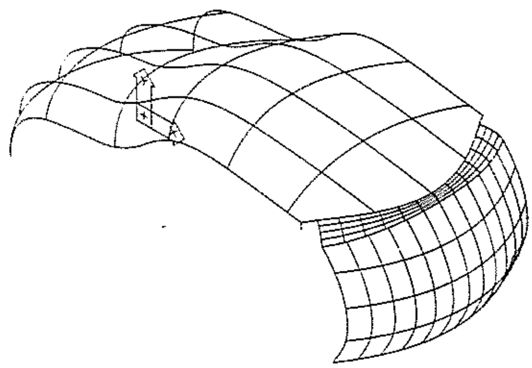


Figure 2.372 Top surface trimmed

Now you will work on the side surfaces. Set the current layer to Wire. Then use the shortcut key [6] to set the display to the front view. After that, set the PDMODE variable to 3 and use the POINT command to construct 24 points.

<Design> **<Point>** **<Single Point>**

Command: **POINT**

	X	Y	Z
A	-108	120	-65
B	-108	55	-15
C	-108	30	0
D	-108	-12	0
E	-82	120	-30
F	-82	55	40
G	-82	30	55
H	-82	-12	55
J	0	120	30
K	0	55	80
L	0	30	95
M	0	-12	95
N	75	120	55
P	75	55	85
Q	75	30	90
R	75	-12	90
S	175	120	55
T	175	55	85
U	175	30	90
V	175	-12	90
W	340	120	30
X	340	55	80
Y	340	30	90
Z	340	-12	90

Set the running object snap mode to Node. Then construct ten splines. (See Figure 2.373.)

<Assist> **<Object Snap Settings...>**

Command: **OSNAP**

[Running Osnap **Node**
OK]

<Design> **<Spline>**

Command: **SPLINE**

Spline	First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
1	**A**	**B**	**C**	**D**	**[Enter]**	**[Enter]**	**[Enter]**
2	**E**	**F**	**G**	**H**	**[Enter]**	**[Enter]**	**[Enter]**
3	**J**	**K**	**L**	**M**	**[Enter]**	**[Enter]**	**[Enter]**
4	**N**	**P**	**Q**	**R**	**[Enter]**	**[Enter]**	**[Enter]**
5	**S**	**T**	**U**	**V**	**[Enter]**	**[Enter]**	**[Enter]**
6	**W**	**X**	**Y**	**Z**	**[Enter]**	**[Enter]**	**[Enter]**

Spline	First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
7	**A**	**E**	**J**	**N**	**S**	**W**	**[Enter]**	**[Enter]**	**[Enter]**
8	**B**	**F**	**K**	**P**	**T**	**X**	**[Enter]**	**[Enter]**	**[Enter]**
9	**C**	**G**	**L**	**Q**	**U**	**Y**	**[Enter]**	**[Enter]**	**[Enter]**
10	**D**	**H**	**M**	**R**	**V**	**Z**	**[Enter]**	**[Enter]**	**[Enter]**

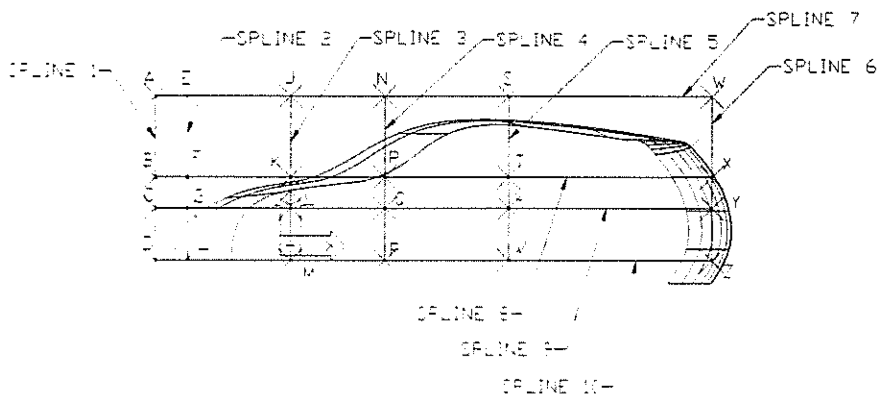


Figure 2.373 Points and splines constructed

Set the current layer to Surface. Then use the AMLOFTUV command to construct a loft uv surface, using splines 1, 2, 3, 4, 5, and 6 (Figure 2.373) as the U-wires and splines 7, 8, 9, and 10 (Figure 2.373) as the V-wires. After that, hide the splines and points, and use the shortcut key [8] to set the display to an isometric view. (See Figure 2.374.)

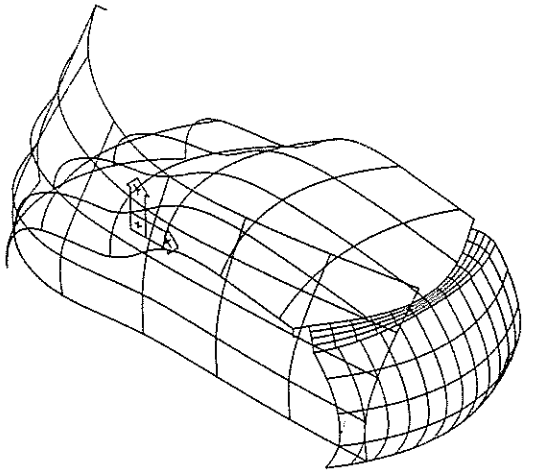


Figure 2.374 Loft uv surface constructed, splines and points hidden, and display set to isometric view

Set the current layer to Wire. Then construct 18 points. After that, construct eight splines. (See Figure 2.375.)

<Design> **<Point>** **<Single Point>**

Command: **POINT**

	X	Y	Z
A	175	55	80
B	175	30	85
C	175	-12	85
D	188	54	77
E	188	30	90
F	188	-12	94
G	222	52	70
H	222	48	84
J	222	30	96
K	222	-12	100
L	298	55	70
M	298	48	93
N	298	30	99
P	298	-12	100
Q	340	55	70
R	340	48	93
S	340	30	99
T	340	-12	100

<Design> **<Spline>**

Command: **SPLINE**

Spline	First Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
1	**A**	**B**	**C**	[Enter]	[Enter]	[Enter]
2	**D**	**E**	**F**	[Enter]	[Enter]	[Enter]

Spline	First Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
3	**G**	**H**	**J**	**K**	[Enter]	[Enter]	[Enter]
4	**L**	**M**	**N**	**P**	[Enter]	[Enter]	[Enter]
5	**Q**	**R**	**S**	**T**	[Enter]	[Enter]	[Enter]

Spline	First Point	Enter Point	Enter Point	Enter Point	Enter Point	Enter Point	Start Tangent	End Tangent
6	**A**	**D**	**G**	**L**	**Q**	[Enter]	[Enter]	[Enter]
7	**B**	**E**	**J**	**N**	**S**	[Enter]	[Enter]	[Enter]
8	**C**	**F**	**K**	**P**	**T**	[Enter]	[Enter]	[Enter]

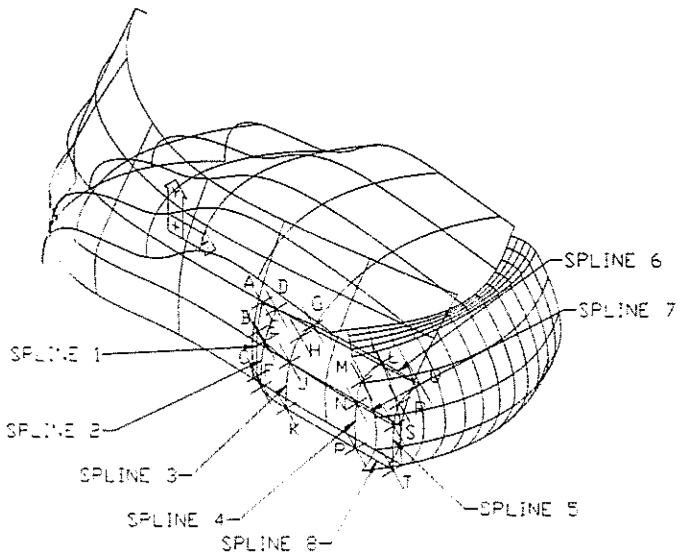


Figure 2.375 Points and splines constructed

Set the current layer to Surface. Then construct a loft uv surface, using splines 1, 2, 3, 4, and 5 (Figure 2.375) as U-wires and splines 6, 7, and 8 (Figure 2.375) as V-wires. After that, hide all the splines and points. (See Figure 2.376.)

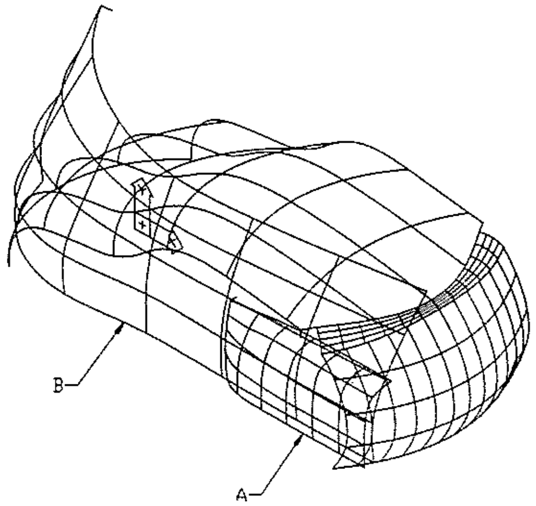


Figure 2.376 Loft uv surface constructed, splines and points hidden

Construct a fillet surface between surfaces A and B (Figure 2.376) with a constant radius of 6 units. (See Figure 2.377.)

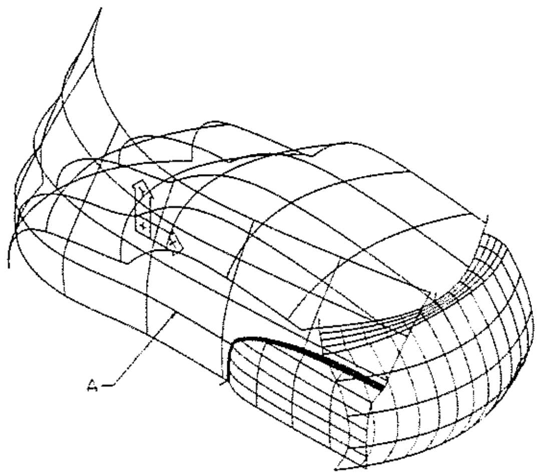


Figure 2.377 Fillet surface constructed

Now use the AMEDITSF command to edit surface A (Figure 2.377) to change its number of U and V grip points and the size of the span. (See Figure 2.378, the Surface Edit dialog box.) Set the number of U grips, the number of V grips, and Span to 20, 2, and 60, respectively. As the names imply, the U and V grips are the grip points along the u and v lines, respectively. When you select a grip point to deform a surface, a circular area of the surface around the selected grip point is affected. Span controls the radius of this circular area. To see how large the affected zone is, select the [Preview] button. (See Figure 2.379.)

To return to the Surface Edit dialog box, press the [Enter] key. Now the numbers of U and V grip points are set and the zone affected by manipulating a grip point is set to a circular area of 60 units radius. Select the surface and then the grip point A (Figure 2.379). Then stretch the surface a distance of 15 units in the Z direction. (See Figure 2.380.)

<Surface> **<Edit Surface>** **<Overall Edit>**

Command: **AMEDITSF**
Select surfaces: **[Select A (Figure 2.377).]**
Select surfaces: **[Enter]**

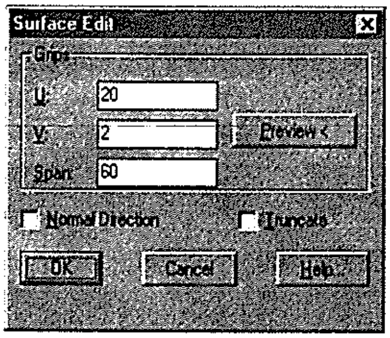


Figure 2.378 Surface Edit dialog box

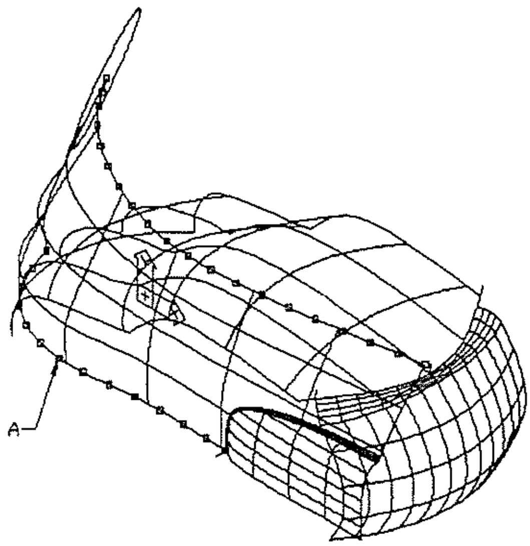


Figure 2.379 Span size and grip points shown

Command:
** STRETCH **
<Stretch to point>/Base point/Copy/Undo/eXit: **@0,0,15**

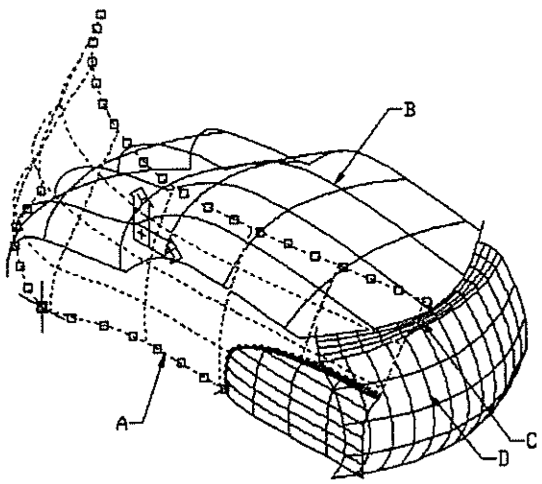


Figure 2.380 Grip point selected and stretched

The side surface is deformed. Now use the AMFILLETSF command to construct fillet surfaces of radius 4 units between surfaces A and B (Figure 2.380), surfaces A and C (Figure 2.380), and surfaces A and D (Figure 2.380). While filleting, trim surfaces A, B, and C (Figure 2.380), but not surface D (Figure 2.380). (See Figure 2.381.)

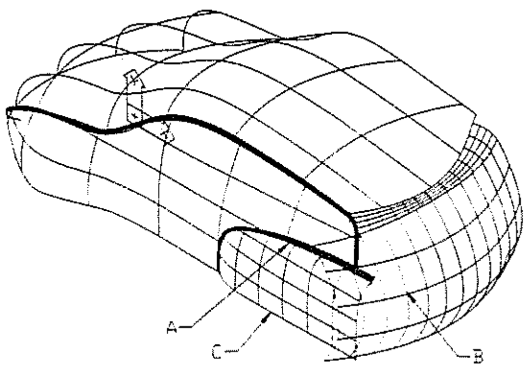


Figure 2.381 Fillet surfaces constructed

As shown in Figure 2.382, construct a constant fillet of radius 4 units between surfaces A and B (Figure 2.381) and a cubical variable fillet of radii 4 units to 12 units between surfaces C and B (Figure 2.381). While filleting, trim surfaces A and C (Figure 2.381), but not surface B (Figure 2.381).

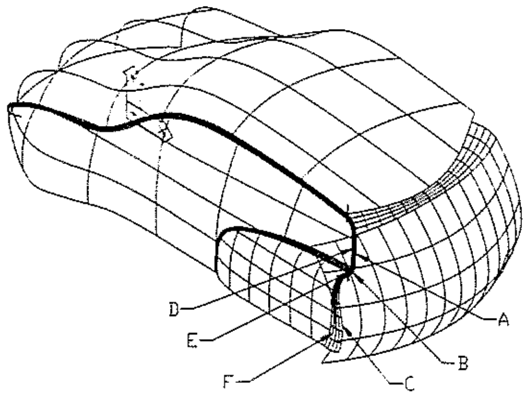


Figure 2.382 Two fillet surfaces constructed

Set the current layer to Wire. Then copy the trimmed edges A, B, and C (Figure 2.382). After copying, hide the fillet surfaces D, E, and F (Figure 2.382). (See Figure 2.383.)

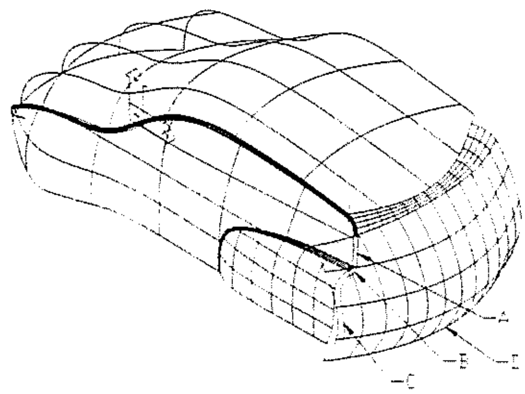


Figure 2.383 Trimmed edges copied, fillet surfaces hidden

Use the copied edges A, B, and C (Figure 2.383) to trim surface D (Figure 2.383). Then unhide the hidden surfaces. (See Figure 2.384.)

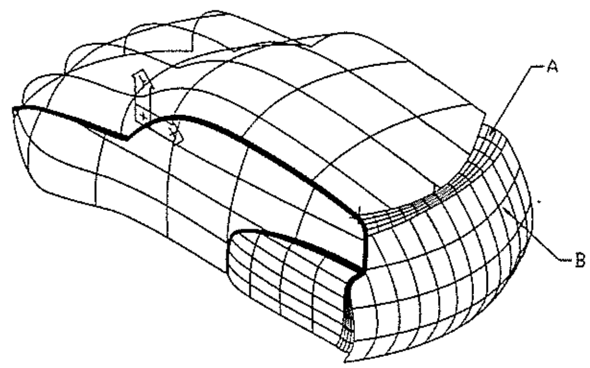


Figure 2.384 Surface trimmed, hidden surfaces unhidden

Erase surfaces A and B (Figure 2.384). Then use the shortcut key [6] to set the display to the front view, and use the LINE command to construct a horizontal line passing through the points (−120,−12) and (400,−12). (See Figure 2.385.)

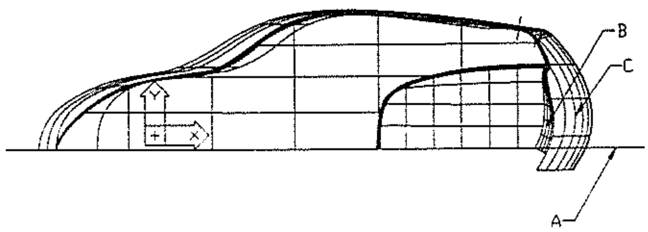


Figure 2.385 Surfaces erased, display set to front view, and horizontal line constructed

Using the horizontal line as a trimming wire, project and trim the rear surface and the variable fillet surface between the side and rear surfaces. (See Figure 2.386.)

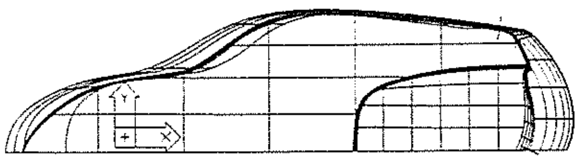


Figure 2.386 Surfaces trimmed by the construction line

Now construct a polyline. (See Figure 2.387.)

<Design> **<Polyline>**

Command: **PLINE**
From point: **38,-12**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<90**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **A**

Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **@76<180**
Angle/CEnter/CLose/Direction/Halfwidth/Line/Radius/Second pt/Undo/Width/
<Endpoint of arc>: **L**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **@12<270**
Arc/Close/Halfwidth/Length/Undo/Width/<Endpoint of line>: **[Enter]**

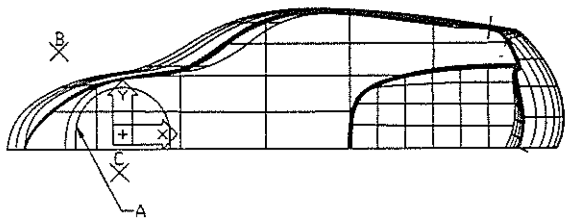


Figure 2.387 Polyline constructed

Use the OFFSET command to offset the polyline twice. (See Figure 2.388.)

<Construct> **<Offset>**

Command: **OFFSET**
Offset distance or Through: **3**
Select object to offset: **[Select A (Figure 2.387).]**
Side to offset? **[Select B (Figure 2.387).]**
Select object to offset: **[Select A (Figure 2.387).]**
Side to offset? **[Select C (Figure 2.387).]**
Select object to offset: **[Enter]**

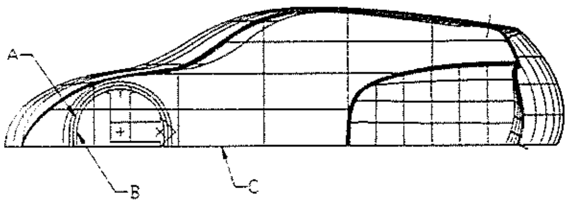


Figure 2.388 Offset lines constructed

Copy wire A (Figure 2.388) a distance of 260 units in the X direction, and project wires A and B (Figure 2.388) onto surface C (Figure 2.388) to obtain two projected wires. After that, set the display to an isometric view. (See Figure 2.389.)

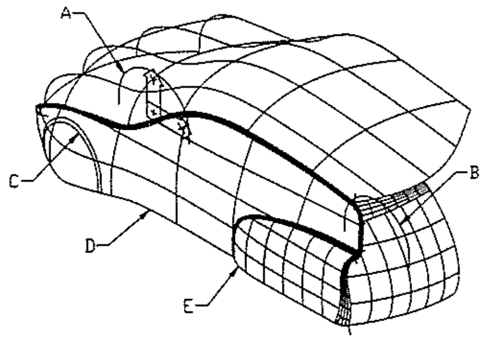


Figure 2.389 Wire copied, wires projected

Move wire C (Figure 2.389) a distance of 4 units in the Z direction. Then project wire A (Figure 2.389) to trim surface D (Figure 2.389), and wire B (Figure 2.389) to trim surface E (Figure 2.389). Projection direction is normal to the UCS. (See Figure 2.390.)

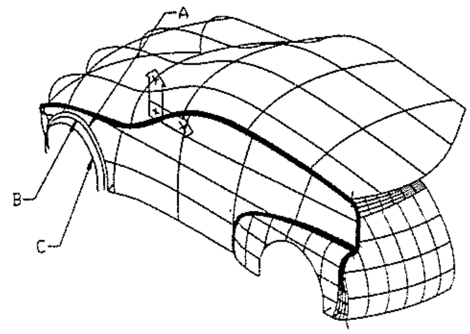


Figure 2.390 Wire moved, surfaces trimmed

Set the current layer to Surface. Then construct a loft u surface, using wires A, B, and C (Figure 2.390). After that, hide all the wires. (See Figure 2.391.)

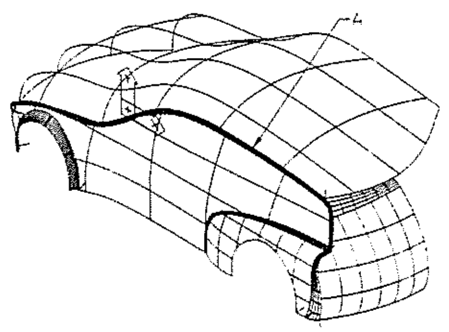


Figure 2.391 Loft u surface constructed

Use the AMEDGE command to copy the trimmed edge A (Figure 2.391). Then use the MIRROR3D command to mirror the copied edge. While mirroring, delete the old object. (See Figure 2.392.)

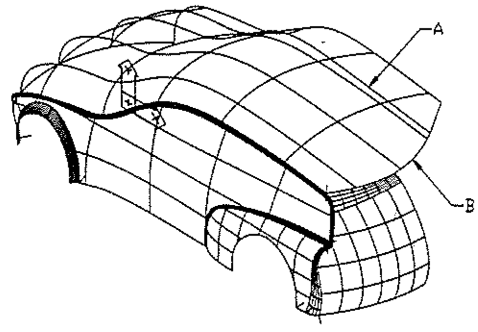


Figure 2.392 Trimmed edge copied and mirrored

Using the mirrored edge A (Figure 2.392) as a cutting wire, project and trim the top surface B (Figure 2.392). Projection direction is normal to the surface. (See Figure 2.393.)

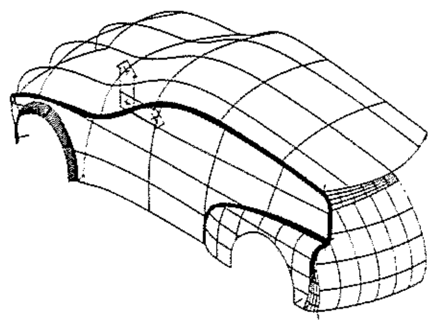


Figure 2.393 Surface trimmed

Complete the model car, as shown in Figure 2.394, by using the MIRROR3D command.

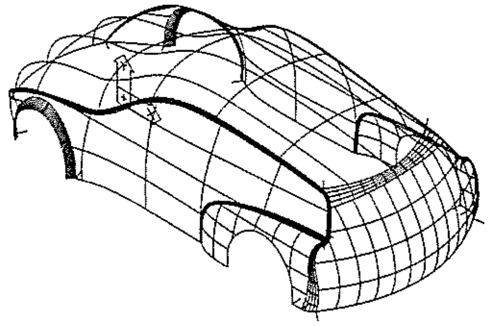


Figure 2.394 Surfaces mirrored

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Coupe.dwg**

Exercise 2.9

Figure 2.395 shows the rendered image of a model of a delivery van. It is modified from one of the design projects in this chapter. Compare Figure 2.395 with Figure 2.396. You can see that the van model is converted from the car model by adding a series of surfaces that form the shape of a box and trimming away some unwanted portions of the surfaces. To make a series of surfaces to form a box quickly, you can use the BOX command to make a solid box, then convert the solid box into a number of surfaces.

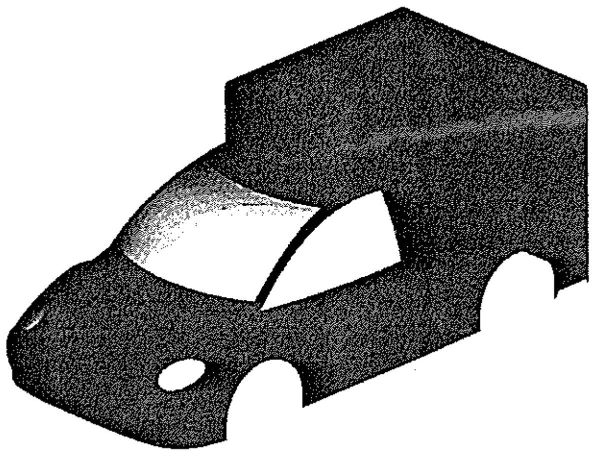


Figure 2.395 Rendered image of the body of a scale model delivery van

Open the drawing Car2.dwg that you saved earlier in this chapter. Then save it under the name Van.dwg. (See Figure 2.396.)

<File> **<Open...>**

File name: **Car2.dwg**

<File> **<Save As...>**

File name: **Van.dwg**

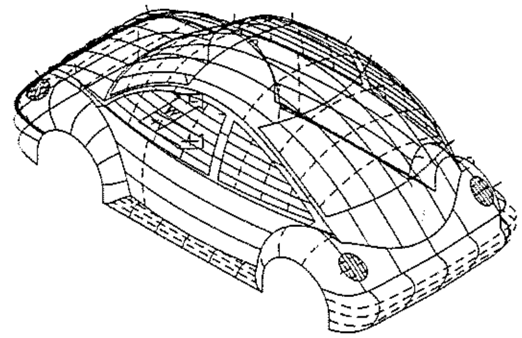


Figure 2.396 Model of the car constructed earlier in this chapter

Use the BOX command to make a solid box with corners at (155,98,–12) and (340,–98,148). (See Figure 2.397.)

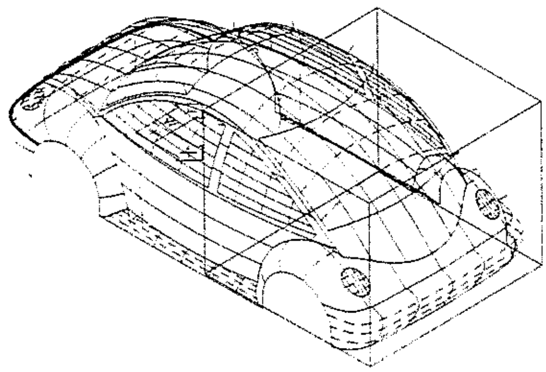


Figure 2.397 Solid box constructed

As shown in Figure 2.398, hide all the surfaces, then round off the edges of the solid box with a fillet radius of 6 units.

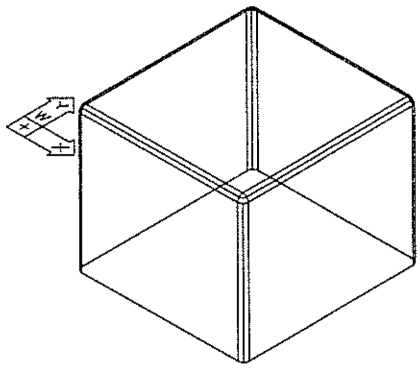


Figure 2.398 Surfaces hidden, edges filleted

Use the AM2SF command to convert the solid box into a collection of surfaces. (See Figure 2.399.)

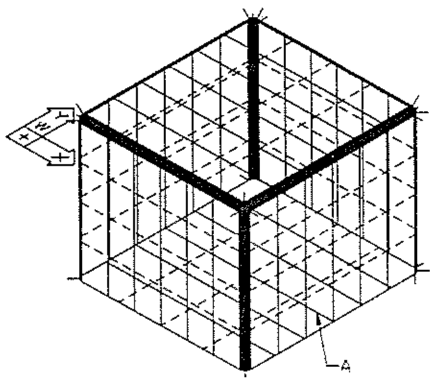


Figure 2.399 Solid box converted into surfaces

The bottom surface A (Figure 2.399) is not needed. Erase it. Then unhide all the hidden surfaces. (See Figure 2.400.)

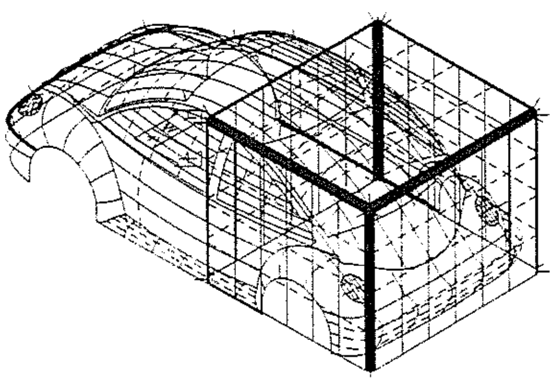


Figure 2.400 Bottom surface erased, hidden surfaces unhidden

As shown in Figure 2.401, trim away the unwanted portions of the surfaces.

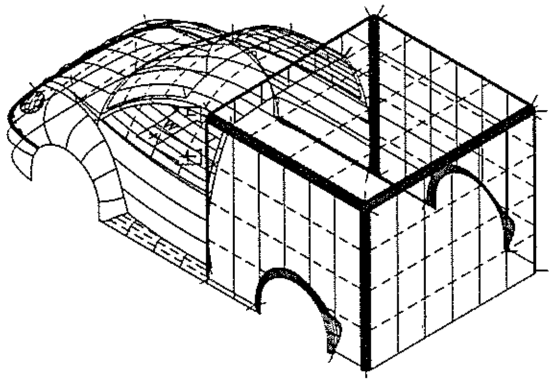


Figure 2.401 Unwanted surfaces trimmed away

The model is complete. Save your drawing.

Chapter 3

Parametric Solid Modeling

Aims and Objectives

The aims of this chapter are to introduce the concepts of parametric solid modeling, to explain ways to construct parametric solids by using the sketching approach and the building-block approach, to delineate the way design variables are manipulated and incorporated, to outline the way free-form surface features can be added to parametric solids, and to let you practice using various solid modeling utilities. After studying this chapter, you should be able to

- Describe the key concepts of parametric solid modeling
- Use the sketching approach and the building-block approach to construct solid parts
- Apply geometric constraints and parametric dimensions in sketching and design
- Employ design variables in engineering design
- Use solid modeling utilities
- Incorporate free-form surfaces into parametric solid parts
- Use Mechanical Desktop as an engineering design tool in making solid parts

Overview

A solid model in the computer is an integrated mathematical representation of a 3D object. There are two kinds of solids: native solids and parametric solids. To construct native solids, you can use AutoCAD solid modeling commands. *(Mastering AutoCAD Release 14 Cheng, R .K. C. Brooks/Cole, 1999.)* To construct parametric solids, you can use Mechanical Desktop. In this chapter, you will learn the concepts of parametric solid modeling, the techniques of constructing parametric solid models, and ways to incorporate 3D surface features into a parametric solid.

Parametric solids differ from AutoCAD R14 native solids in many respects. When constructing native solids, you specify the dimensions explicitly. You start from constructing primitive solids, extrude solids, and revolve solids. Then you combine them into complex solid models by using Boolean operations. After you make the solids, the Boolean operations cannot be reversed. The solids that you constructed in the last chapter are native solids. You can add more features to them, but you cannot remove any combined features afterward.

With parametric solids, you can change the dimensions of a solid feature and you can reverse any Boolean or surface-cutting operations any time you like. Going beyond overcoming these two shortcomings of native solids, parametric solids can be constructed from dimension-driven rough sketches. Instead of making precise wires, you start from freehand sketches. While making such a sketch, you need to concentrate only on form and shape. Then you resolve the sketch and add appropriate geometric constraints and parametric dimensions. With a properly constrained sketch, you construct a parametric solid feature. Not only can you change a sketch before making a solid feature from it, but you can alter the dimensions and shape of the solid feature at a later stage.

In this chapter, you will work on a number of solid modeling projects through which you will learn the techniques of using Mechanical Desktop to construct parametric solid models. In addition, you will learn how to incorporate free-form surfaces into parametric solids. In the next chapter, you will learn how to assemble sets of solid parts that you construct in this chapter.

3.1 Parametric Solid Modeling Concepts

In the early design stage, you have a shape in your mind but you may not have decided on any precise dimensions. To elaborate and formulate your design idea further, you may want to construct a 3D computer model. However, most conventional computer-aided design (CAD) systems require you to input the precise lengths and geometry of the entities from the outset. In the preliminary design stage, such data are not available.

With Mechanical Desktop, you can use the computer as an electronic sketch pad to record your design idea and you can concentrate on forms and shapes rather than on dimensions. In the absence of exact dimensions, you can construct a rough freehand sketch. Figure 3.1 shows a sketch of an L-shaped block.

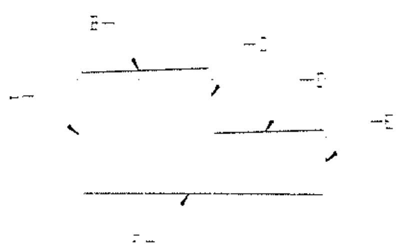


Figure 3.1 Rough freehand sketch

As the name implies, rough sketches are not precise at all. Lengths are approximate. Lines need not be joined at their ends. Horizontal lines need not be absolutely horizontal. Vertical lines need not be perfectly vertical. While sketching, you concentrate only on the form and shape. After you are satisfied with the form and shape of the sketch, you can let Mechanical Desktop resolve it. Figure 3.2 shows the resolved sketch.

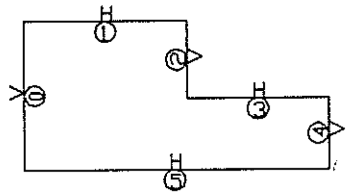


Figure 3.2 Sketch resolved by Mechanical Desktop

When the sketch is resolved, a set of rules is applied. The end points that are close to each other are joined; the lines A, C, and E (Figure 3.1) that are nearly vertical are resolved as vertical; and the lines B, D, and F (Figure 3.1) that are nearly horizontal are resolved as horizontal. A special feature of a resolved sketch is that it can be driven by parametric dimensions. These dimensions do not merely report the size; they drive the change in the size of the sketch. Figure 3.3 shows how the resolved sketch changes with the parametric dimensions added.

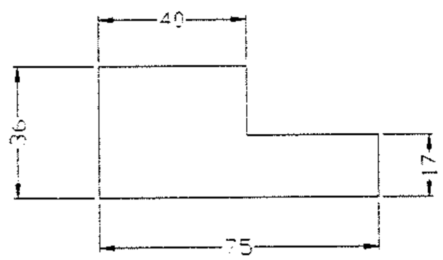


Figure 3.3 Parametric dimensions added

With a properly resolved and dimensioned sketch, you can extrude, loft, revolve, and sweep it to obtain a parametric solid. Figure 3.4 shows the isometric view of a parametric solid extruded from the resolved sketch.

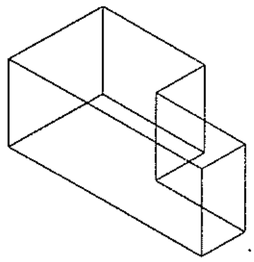


Figure 3.4 Isometric view of the extrude solid constructed from the resolved sketch

Unlike native solids, which are static, parametric solids can be edited. Figure 3.5 shows how the solid changes after new values are assigned to the parametric dimensions.

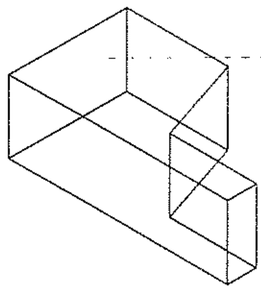


Figure 3.5 Parametric solid changed

Now we will discuss the processes of sketching and resolving, application of geometric constraints and parametric dimensions, and ways to build parametric solid models in more detail.

Sketches

A parametric solid starts from a sketch. There are two kinds of sketches: 3D and 2D. Constructing a 3D sketch is very simple; you just specify its parameters.

To make a 2D sketch, you start from a rough sketch. While you are sketching, dimensions are unimportant, lines need not be absolutely horizontal or vertical, and the drawing entities need not be joined precisely at their ends. You should concentrate only on forms and shapes and then focus on the dimensions at a later design stage. Depending on what subsequent operation you want to perform, you can resolve the sketch into a profile, a path, a cut line, or a split line. (See Figure 3.6.)

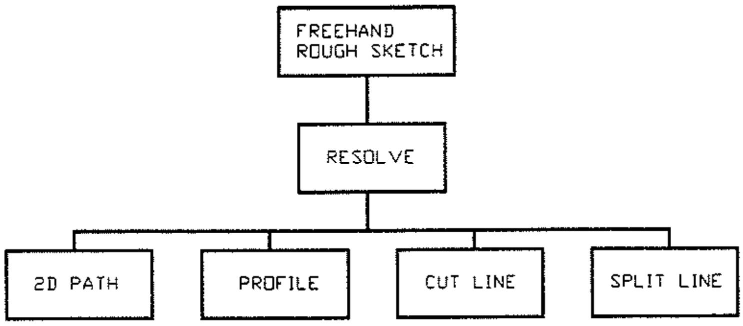


Figure 3.6 Resolution of rough sketches

Resolved Sketches

To construct parametric solids, you need profiles and paths. You can extrude a profile linearly, revolve a profile about an axis, sweep a profile along a 2D path, sweep a profile along a 3D path, and loft along a set of profiles. To construct an offset section while generating a 2D engineering drawing from a solid, you need a cut line. To split a face of a solid into two in order to generate a drafting taper on the face from the split edge, you need a split line. Figure 3.7 outlines the use of resolved sketches and specified 3D sketches in solid modeling and associative drafting. Note that a 3D path is not derived from a freehand sketch.

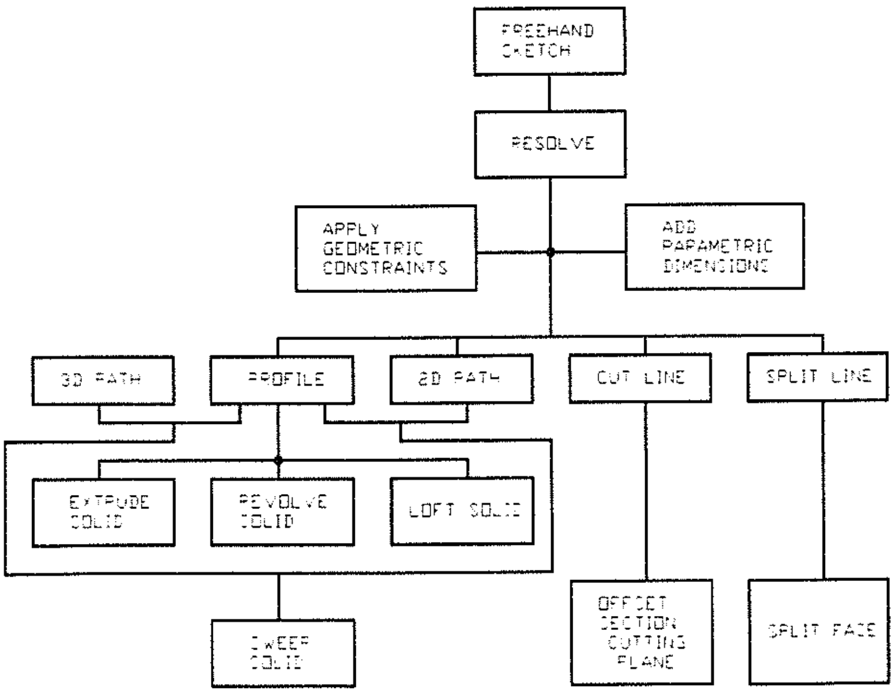


Figure 3.7 Use of resolved sketches and specified 3D paths

Geometric Constraints

During resolving, the following logical geometric constraints are applied to the sketch:

- Lines will become horizontal if they are nearly horizontal.
- Lines will become vertical if they are nearly vertical.
- Lines will become perpendicular if they are nearly perpendicular to each other.
- Lines will become parallel if they are nearly parallel to each other.
- Centers of arcs and circles will align at the same X coordinate if their X values are nearly the same.
- Centers of arcs and circles will align at the same Y coordinate if their Y values are nearly the same.
- Lines will become collinear if they lie nearly on the same straight line.
- Arcs and circles will become concentric if they are nearly concentric.
- A point of an object will be bound to the definition point of another object if they are located at nearly the same position. For example, if the center of an arc and the end point of a line lie nearly at the same location, the two points will be bound together.
- Objects will join together at their end points if they are close enough.
- Arcs and circles will have the same radius if they have nearly the same radius.
- Lines, arcs, and circles will become tangent if they are nearly tangent to each other.
- End points of lines will align at the same X coordinate if their X values are nearly the same.
- End points of lines will align at the same Y coordinate if their Y values are nearly the same.
- Lines will have equal lengths if their lengths are nearly the same.

It is possible that the kinds of geometric constraints that Mechanical Desktop adds automatically to your sketch will not conform to your design intention. For example, you may want a line to be horizontal but the line that you constructed may deviate so much from the horizontal that Mechanical Desktop treats it as an inclined line. Or perhaps you do not want two lines to be collinear but the locations of the lines that you drew are such that Mechanical Desktop treats them as collinear. In the first example, you should add the horizontal constraint to the line. In the second example, you should remove the collinear constraint from the lines. Figure 3.8 outlines the process of constraining a resolved sketch geometrically. You should make necessary adjustments to the geometric constraints that Mechanical Desktop automatically applies to the sketch by deleting those constraints that you do not want and adding those constraints that Mechanical Desktop does not apply for you.

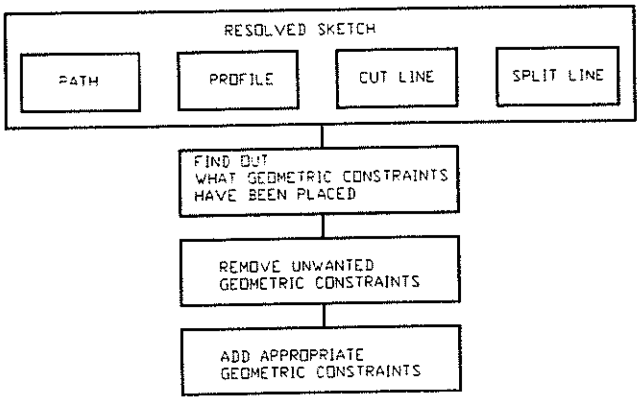


Figure 3.8 Application of geometric constraints

Parametric Dimensions

To constrain a resolved sketch properly, you have to add parametric dimensions as well as geometric constraints. While adding the parametric dimensions, note that the size of the resolved sketch changes instantly — the parametric dimensions drive the sketch.

Each parametric dimension that you add to a resolved sketch has a default value and a parameter name. The default value is the measured size of the dimension. You can change it while it is placed or you can change it later. The parameter name consists of the letter *d* and a suffix number that starts at zero. The parameter name of the first parametric dimension that you add to a resolved sketch is d0. The parameter name of the second dimension is d1, and so forth.

In your display, a parametric dimension is exhibited in one of three formats: as a number, as a parameter, or as an equation. (See Figure 3.9.)

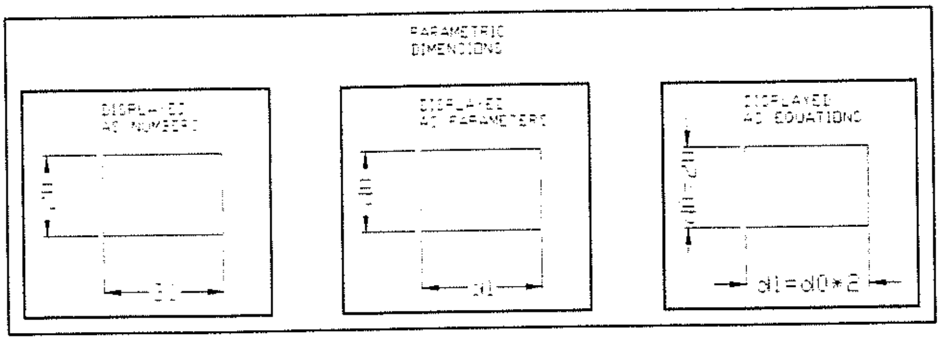


Figure 3.9 Display formats of parametric dimensions

When a parametric dimension is exhibited as a number, the numeric value of the dimension is displayed. When a parametric dimension is exhibited as a parameter, the default parameter name (such as d0 or d1) is displayed. When a parametric dimension is exhibited as an equation, the dimension is stated as the default parameter name together with an equal sign (=) and an expression that defines the dimension. The expression can be a simple number. It can have parameter names, design variables, and mathematical expressions. For example, if the parameter name is d0 and the value of the dimension is 10 units, the parametric dimension is shown as "d0=10." If the parameter name is d2 and the value of the dimension is half of the dimension d0 plus 5 units, the parametric dimension is shown as "d2=d0/2+5."

Using an expression as the parametric dimension value enables you to incorporate design intent into the model. For example, you may want the width of a rectangle to be half the length. If the parameter name of the length of the rectangle is d5, you can set the width of the rectangle to be d5/2. In a later design stage, when you change the length, the width follows the change.

Sketched Features

Features that are derived from the sketches are called sketched features. They are the extrude solid feature, the revolve solid feature, the sweep solid feature from 2D path, the sweep solid feature from 3D path, the loft solid feature, and the split face feature. (See Figures 3.10 through 3.14.)

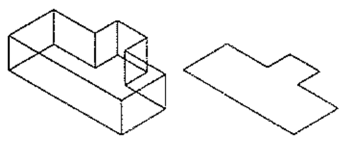


Figure 3.10 Sketched feature — extrude solid

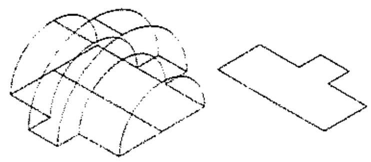


Figure 3.11 Sketched feature — revolve solid

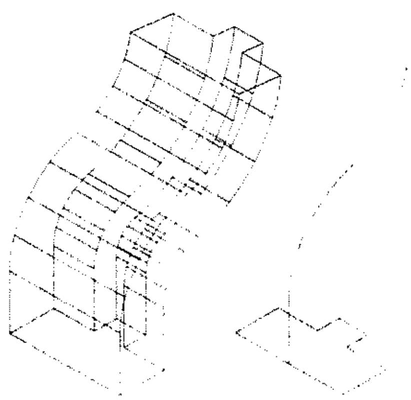


Figure 3.12 Sketched feature — sweep solid from 2D path

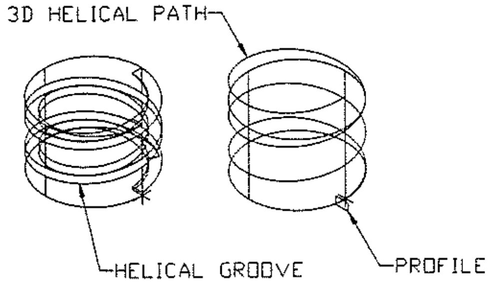


Figure 3.13 Sketched feature — sweep solid from 3D path

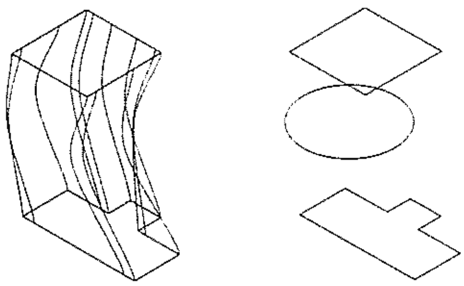


Figure 3.14 Sketched feature — loft solid

A split line is used to construct a split face feature. A split face feature is a face with a split line. With a split line, you can construct face drafting in two directions on a single face. Face drafting is a placed feature and will be explained later. Figure 3.15 shows a split face feature.

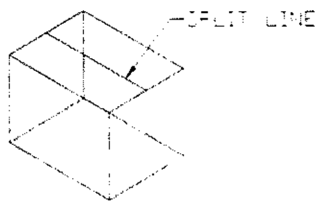


Figure 3.15 Sketched feature — split face

Building-Block Approach

In the real world, 3D objects are seldom as simple as those shown in Figures 3.10 through 3.14. They are usually more complex in shape and form. To build these complex solid models, you have to construct a number of simple solid features and then combine them. Before starting to work, you have to analyze the model to find out what solid features are required and how the features are combined. After making the analysis, you start modeling.

Among the set of solid features that you will construct to make the model, you select one and make it from a resolved sketch or sketches. This first solid feature is called the base feature of the complex solid model. To complete modeling, you add the other features to it by joining, cutting, and intersecting. This is called the building-block approach. (See Figure 3.16.)

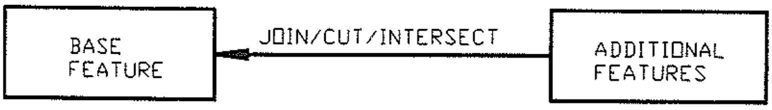


Figure 3.16 Building-block approach

Sketch Planes

While sketching, you need to put your sketch on a sketch plane. For the base feature of a solid model, you can specify the XY plane of any UCS. For subsequently built solid features, you have to set up proper sketch planes in order to maintain proper relationships among the solid features. The most convenient sketch planes are existing faces on a solid model. If you do not have any appropriate faces to work on, you have to establish a work plane and then use it as a sketch plane. A work plane is a kind of work feature.

Work Features

Work axes, work planes, and work points are called work features. They are integral parts of the parametric solid part. Using them, you can define and maintain parametric relationships among the solid features. (See Figures 3.17 through 3.19.)

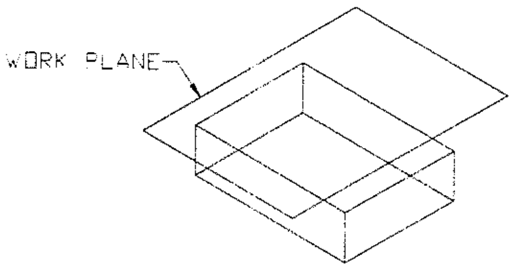


Figure 3.17 Work plane parallel to and offset from an existing face

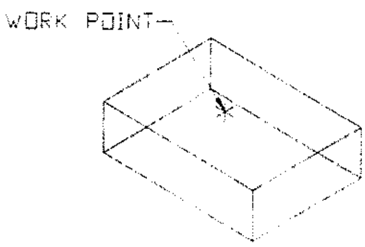


Figure 3.18 Work point placed on a sketch plane

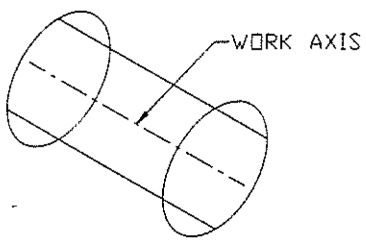


Figure 3.19 Work axis established on a revolve solid

Placed Features

In addition to making solid features from rough sketches, you can construct solid features directly by specifying their parameters. Such features are called placed features. A placed solid feature differs from a sketched solid feature in that it is not derived from a sketch. Instead, it is placed on a solid part. The kinds of features that you can place on a solid part are hole features, fillet features, chamfer features, face draft features, shell features, surface cut features, array features, copy features, combine features, and split part features. Figures 3.20 through 3.27 depict various kinds of placed solid features, Figure 3.28 shows how two solid parts can be combined, and Figure 3.29 shows how a solid is split into two.

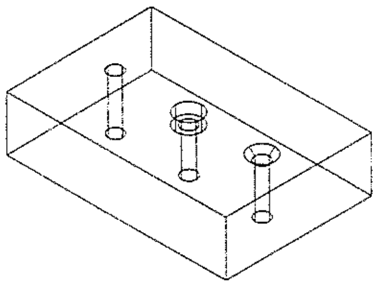


Figure 3.20 Hole features placed in a rectangular solid

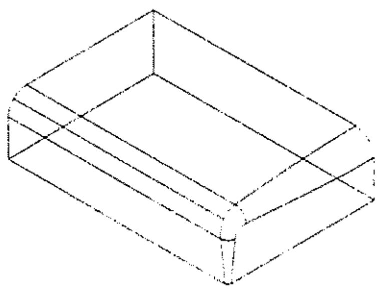


Figure 3.21 Fillet features placed on a rectangular solid

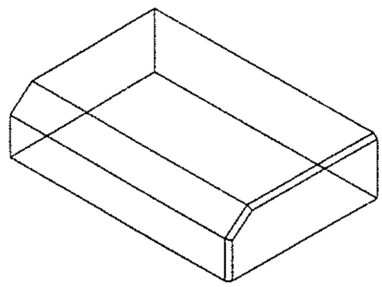


Figure 3.22 Chamfer features placed on a rectangular solid

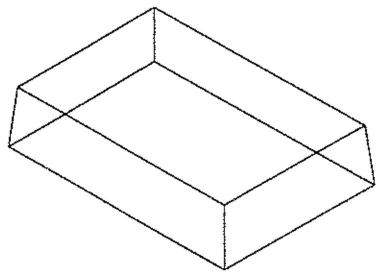


Figure 3.23 Face draft features placed on a rectangular solid

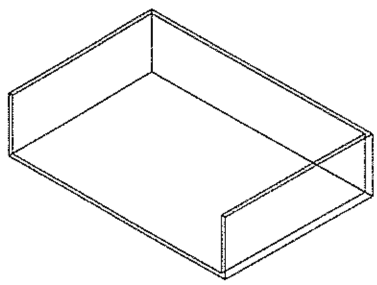


Figure 3.24 Shell feature placed on a rectangular solid

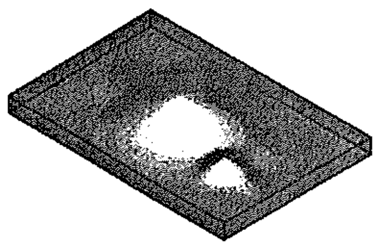


Figure 3.25 Surface cut feature placed on a rectangular solid

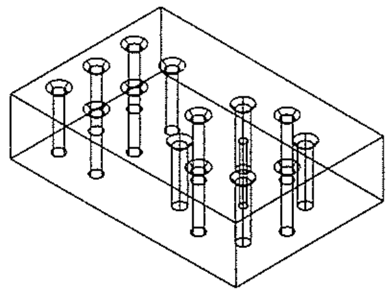


Figure 3.26 Rectangular and polar array features placed on a rectangular solid

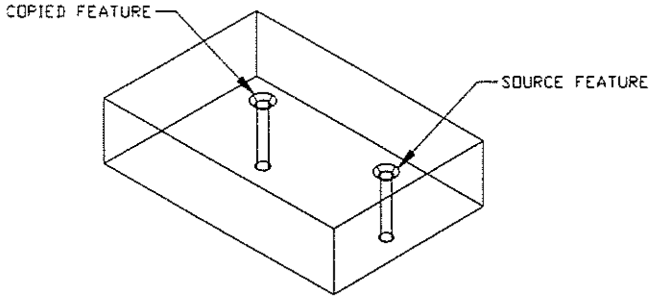


Figure 3.27 Copy feature placed on a rectangular solid

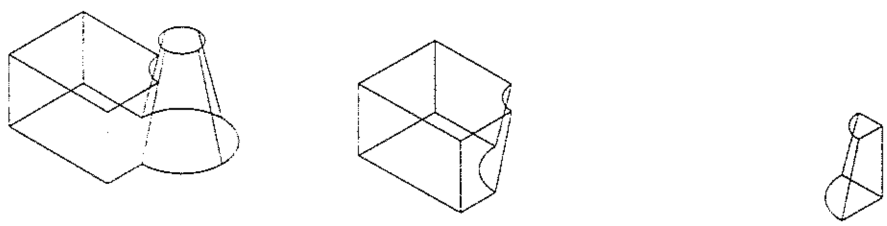


Figure 3.28 Combining two solid parts by joining, cutting, and intersecting

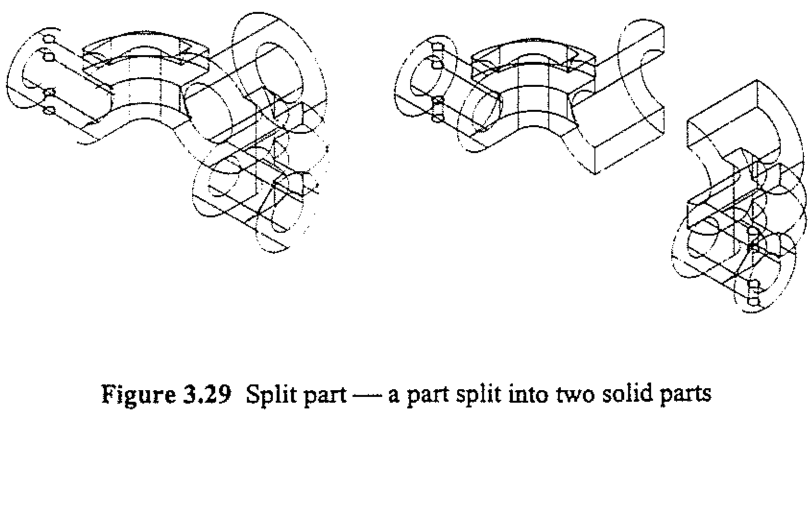


Figure 3.29 Split part — a part split into two solid parts

Features of a Parametric Solid Model

To summarize, a parametric solid model is constructed by first making a sketch from which you construct a solid feature. Then you incorporate additional solid features (either sketched solid features or placed solid features) by joining, cutting, or intersecting. To establish and maintain parametric relationships among the solid features, you need to use work features. Figure 3.30 outlines the three major kinds of features of a parametric solid model: sketched features, placed features, and work features.

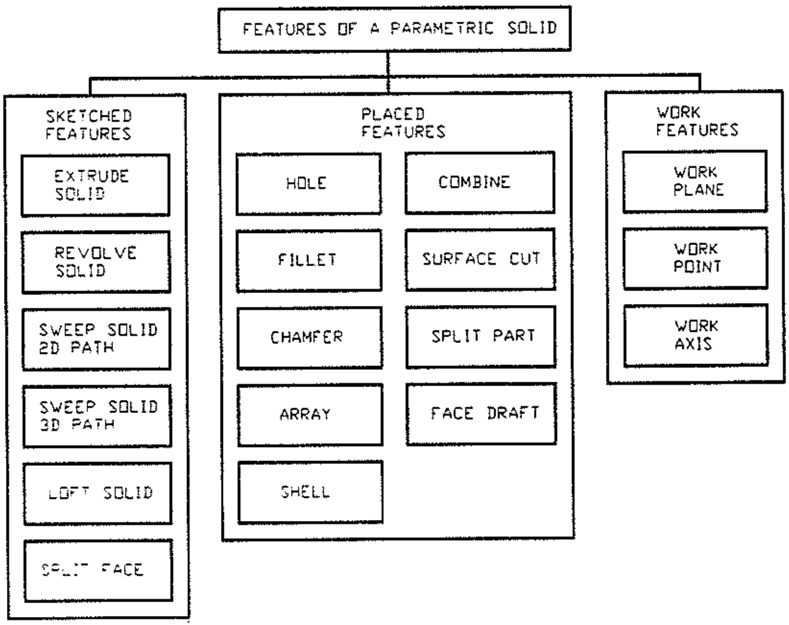


Figure 3.30 Features of a parametric solid model

Edit Feature

Parametric features can be edited. You can modify them by specifying new values for their parameters. (See Figure 3.31 through 3.33.)

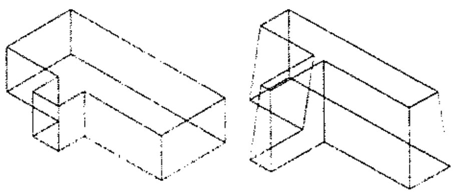


Figure 3.31 Parameters of an extrude solid (a kind of sketched feature) modified

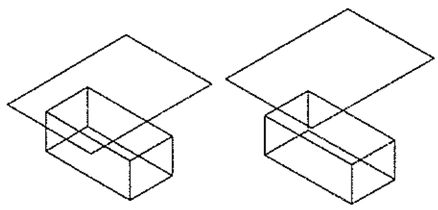


Figure 3.32 Parameters of a work plane (a kind of work feature) modified

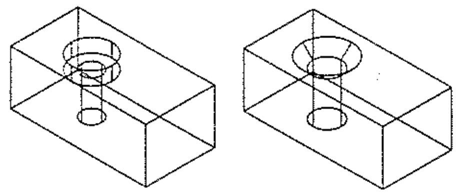


Figure 3.33 Parameters of a hole feature (a kind of placed solid feature) modified

Reorder Feature

Sometimes, the working sequence of constructing a complex solid model affects the modeling outcome. Consider a base feature A and two additional features B and C (Figures 3.34 and 3.35). In Figure 3.34, feature C is joined and then feature B is cut. In Figure 3.35, feature B is cut and then feature C is joined. Note the difference.

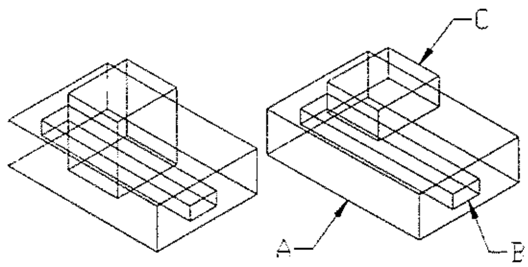


Figure 3.34 Feature C is joined to base feature A, then feature B is cut.

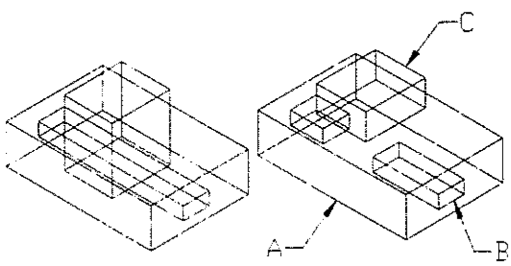


Figure 3.35 Feature B is cut from base feature A, then feature C is joined.

While you are modeling, you may construct the features in the wrong sequence or you may want to add one operation before another that has already been performed. For example, you may want to have the outcome shown in Figure 3.35 but you may be working in the sequence shown in Figure 3.34. To change the outcome, you can reorder the operations.

Delete Feature

Sometimes you want to remove a certain feature from a solid. To do so, you can delete the unwanted feature. Figure 3.36 shows a hole feature deleted.

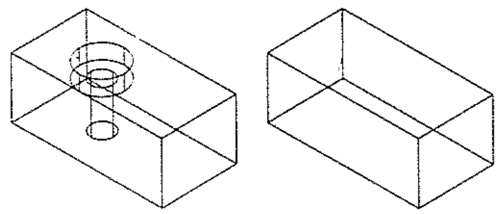


Figure 3.36 Feature deleted

Suppress Feature

While constructing solid models in the computer, we view our 3D models through a 2D display. For very complex solid models, there may be so many wires displayed on the screen that we find it difficult to select a point or an edge. To facilitate working on very complex solids, you can suppress the visibility of some features. After a feature is suppressed, it is disregarded in the solid model as if it had been deleted. You can also use the feature suppression facility to try out various design possibilities.

Compatibility and Interoperation

Mechanical desktop solids and surfaces and AutoCAD native solids are fully compatible with each other. You can interoperate them. You can change a parametric solid to a native solid or convert a native solid to a static base feature. A static base feature is a base solid feature that you cannot modify because there is no parametric history in it. However, any subsequent parametric solid features added to it are parametric and can be edited. Working with surfaces, you can use a NURBS surface as a surface feature to cut a parametric solid or a native solid. You can also convert a parametric solid to a set of NURBS surfaces. (See Figure 3.37.)

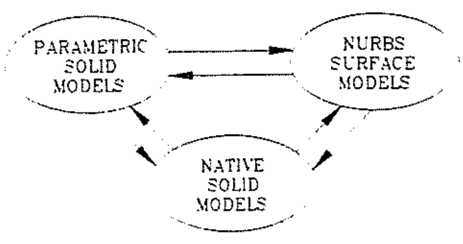


Figure 3.37 Interoperation

3.2 Part Modeling Preferences

Take some time to study the system variables that affect parametric solid modeling. Then save a template drawing for the projects you will work on in this chapter.

Setting Part Modeling Preferences

To set part modeling preferences, select the Preferences... item of the Part pull-down menu to use the Part tab of the Desktop Preferences dialog box. (See Figure 3.38.)

<Part> **<Preferences...>**

Command: **AMPREFS**

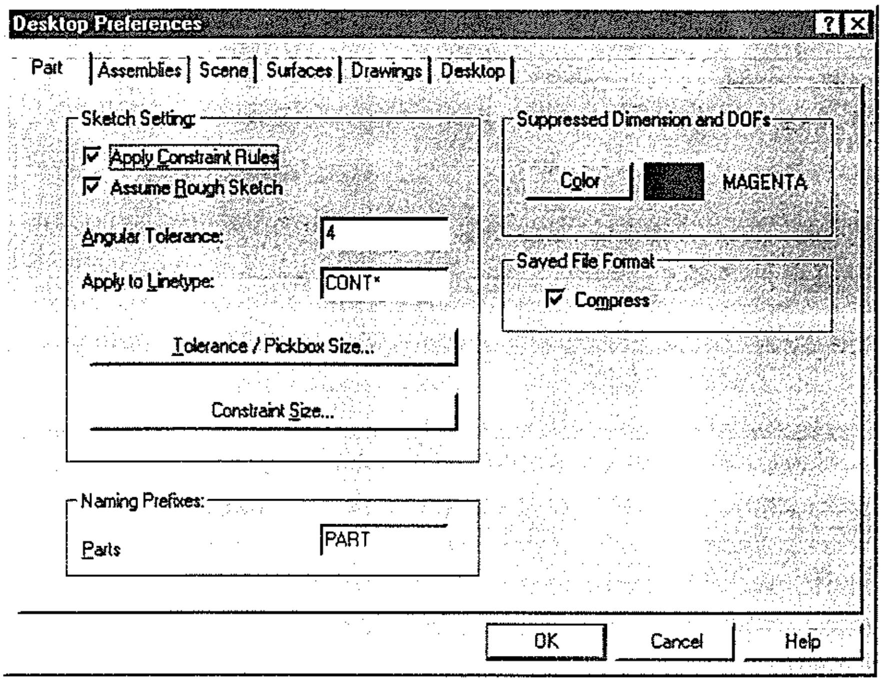


Figure 3.38 Part tab of the Desktop Preferences dialog box

The Part tab of the Desktop Preferences dialog box has four major areas: Sketch Setting, Naming Prefixes, Suppressed Dimension and DOFs, and Saved File Format.

The Sketch Setting area concerns the way a sketch is resolved. There are six items: Apply Constraint Rules, Assume Rough Sketch, Angular Tolerance, Apply to Linetype, Tolerance/Pickbox Size, and Constraint Size.

As we have said, the rough sketch for making parametric solid features need not be precise at all. You do not have to care about the exact orientation and dimensions and you do not have to use the snap object tool to locate the endpoints. Now you may ask how rough a sketch can be, and how Mechanical Desktop interprets your sketch. Of course, there are rules. You may classify the rules into two categories.

The first includes the rules of geometric constraint:

- Lines will become horizontal if they are nearly horizontal.

- Lines will become vertical if they are nearly vertical.
- Lines will become perpendicular if they are nearly perpendicular to each other.
- Lines will become parallel if they are nearly parallel to each other.
- Centers of arcs and circles will align at the same X coordinate if their X values are nearly the same.
- Centers of arcs and circles will align at the same Y coordinate if their Y values are nearly the same.
- Lines will become collinear if they lie nearly on the same straight line.
- Arcs and circles will become concentric if they are nearly concentric.
- Arcs and circles will have the same radius if they have nearly the same radius.
- Lines, arcs, and circles will become tangent if they are nearly tangent to each other.
- End points of lines will align at the same X coordinate if their X values are nearly the same.
- End points of lines will align at the same Y coordinate if their Y values are nearly the same.
- Lines will have equal length if their lengths are nearly the same.

The second set of rules joins objects together or attaches objects to another object:

- Points of an object will be bound to the definition point of another object if they are at nearly the same position. For example, if the center of an arc and the end point of a line lie nearly at the same location, the two points will be bound together.
- Objects will join together at their end points if they are close enough.

There are times when you do not want to apply the first set of geometric constraint rules to your sketch during resolution. You may want to attach only the endpoints. If so, uncheck the Apply Constraint Rules box. To apply the rules, check the box.

For example, you may want to resolve a line as inclined at precisely 4°, despite setting the angular tolerance to 5°. (We will explain angular tolerance later.)

To override angular tolerance and treat the sketch as precise, you can uncheck the Assume Rough Sketch box. To apply angular tolerance to the rough sketch, check the box.

Regarding angular tolerance, you may ask: How close to horizontal must a line be for it to be treated as horizontal? The answer is that you can enter a value in the Angular Tolerance box.

While making a rough sketch, you can include construction lines. Construction lines are used in a resolved sketch to guide, locate, or govern other lines. They are not used in subsequent operations such as extruding, revolving, lofting, sweeping, making a split line, or making a cut line. To show which are construction lines and which are not, you can use different linetypes in a sketch. The Apply to Linetype box specifies the linetype that is used for the sketch itself. Any other linetypes are treated as construction lines.

Regarding the second set of rules we mentioned earlier, you may ask: How close must two endpoints be for them to be treated as a single vertex? The answer is that the PICKBOX variable determines the tolerance. The PICKBOX variable specifies a pickbox size in terms of screen pixel size.

Command: **PICKBOX**
New value for PICKBOX: **5**

If the pickbox size is five pixels, two endpoints will join together if you zoom the screen display such that the gap between them is smaller than five screen pixels. At the other end of the spectrum, if you zoom very far away, a number of vertices that are close together within the specified pickbox size will join together as a single vertex. As a result, you may lose some line segments. Therefore, you should set the pickbox size neither too large nor too small. To set the Pickbox Size graphically, select the [Tolerance/Pickbox Size...] button. (See Figure 3.39.)

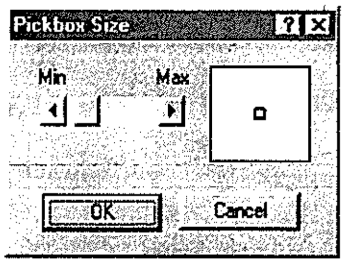


Figure 3.39 Pickbox Size dialog box

After a sketch is resolved and geometric constraints are applied, you can examine it to find out what kind of geometric constraints have been placed on it. To adjust the size of the geometric constraint symbols displayed on the screen, select the [Constraint Size...] button. (See Figure 3.40.)

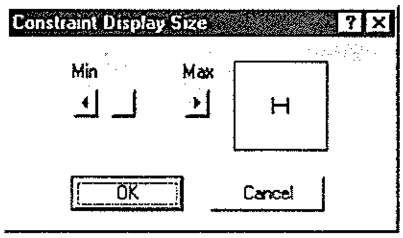


Figure 3.40 Constraint Display Size dialog box

In the Naming Prefixes area of the Part tab, you can set the prefix of the names of the solid parts constructed in the drawing file. The default prefix is Part and the first solid part constructed in a drawing file is named Part1.

In your drawing file, you can suppress dimensions and degrees of freedom (DOFs). An object in the 3D space has six degrees of freedom. In the Suppressed Dimension and DOFs area, you can set the color of the suppressed dimension and DOFs. A parametric solid model "remembers" the history of the process of construction of the parts. Therefore, the file size can become quite large. To reduce drawing file size, you can check the Compress box in the Saved File Format area.

Template Drawing

In this chapter, you will work on a number of solid modeling projects to learn how to construct and modify parametric solids, use solid modeling utilities, and incorporate 3D surfaces into parametric solids. Because you may need similar settings in these drawings, you should set up a template drawing.

Select the File pull-down menu and then the New... item to start a new drawing from scratch. Use metric default settings.

<File> **<New...>**

Command: **NEW**

[Start from Scratch
Metric **OK**]

Use the LAYER command to create two additional layers, Solid and Sketch, with the colors yellow and blue, respectively. You will put the sketches on layer Sketch and the solids on layer Solid. Now set the current layer to Sketch.

<Assist> **<Format>** **<Layer...>**

Command: **LAYER**

[Layer

Name	Color	Linetype
0	White	Continuous
Solid	Red	Continuous
Sketch	Blue	Continuous

Current Layer: **Sketch**

OK]

After resolving a sketch, you need to add geometric constraints and parametric dimensions. To prepare the drawing for dimensioning, use the DDIM command to set the dimension style. (See Figure 3.41.)

<Drawing> **<Annotation Styles>** **<Dimension...>**

Command: **DDIM**

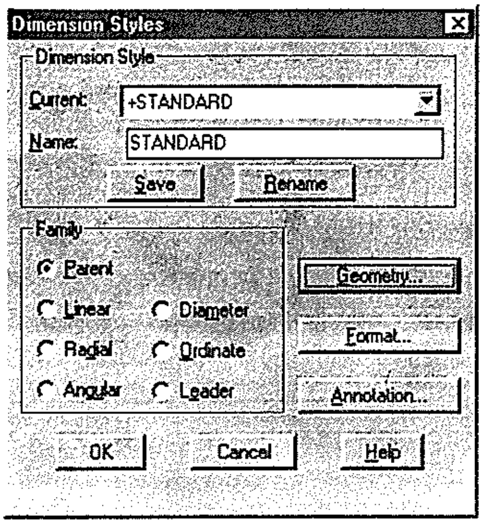


Figure 3.41 Dimension Styles dialog box

Select the [Geometry...] button in the Dimension Styles dialog box to bring up the Geometry dialog box. (See Figure 3.42.)

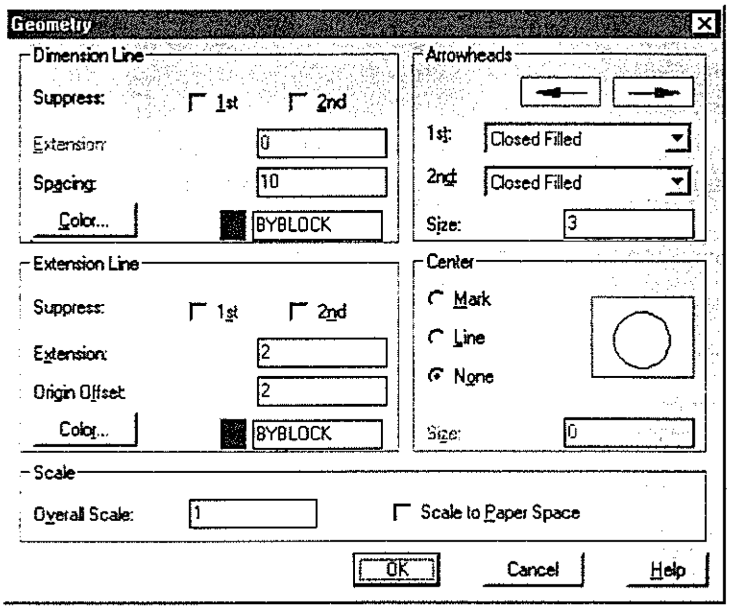


Figure 3.42 Geometry dialog box

Set the values of the parameters in the Geometry dialog box as shown in Figure 3.42. Then select the [OK] button to return to the Dimension Styles dialog box (Figure 3.41). After that, select the [Annotation...] button to bring out the Annotation dialog box. (See Figure 3.43.)

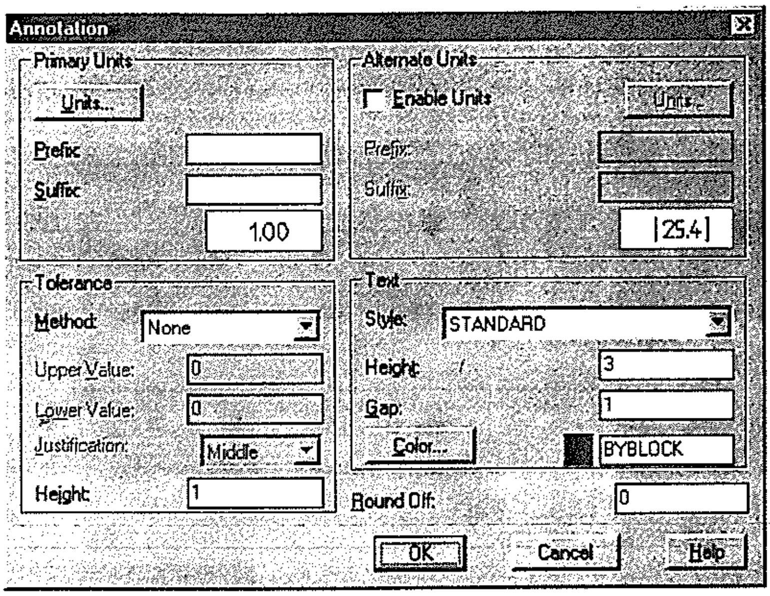


Figure 3.43 Annotation dialog box

In the Annotation dialog box, set the variables as shown in Figure 3.43. Then select the [OK] button to return to the Dimension Styles dialog box. (See Figure 3.41.)

Now select the [Save] button to save the dimension style setting, then select the [OK] button to exit.

While you are working in 3D, it is useful to place the UCS icon at the origin. Run the UCSICON command.

<Assist> **<v Icon at Origin>**

Command: **UCSICON**
ON/OFF/All/Noorigin/ORigin: **OR**

The drawing setup is complete. Save your drawing as a template.

<File> **<Save As...>**

Command: **SAVEAS**

[Save Drawing As
File name: **Solid.dwt**
Save as type: **Drawing Template File (*.dwt)**
Save]

3.3 Multi-Part and Single-Part Drawings

For projects that include more than one solid part, there are three approaches. In the first approach, you construct each individual solid part in a separate drawing file, then attach them to an assembly drawing file. In the second approach, you construct all the solid parts of an assembly in a single drawing file, assemble them together in the same drawing file, and then export the individual parts to separate drawing files.

The third approach combines the first and second. Here you divide the assembly into two sets of solid parts. You construct one set of parts in the assembly drawing file; you construct another set of parts in separate individual drawing files and attach them to the assembly drawing.

Mechanical Desktop provides two ways to start a new drawing: start a new drawing that holds a single solid part and start a new drawing that holds multiple solid parts.

Multi-Part Drawing

To start a new drawing that includes multiple solid parts, you select the New... item from the File pull-down menu. Figure 3.44 shows the user interface for a multi-part drawing.

<File> **<New...>**

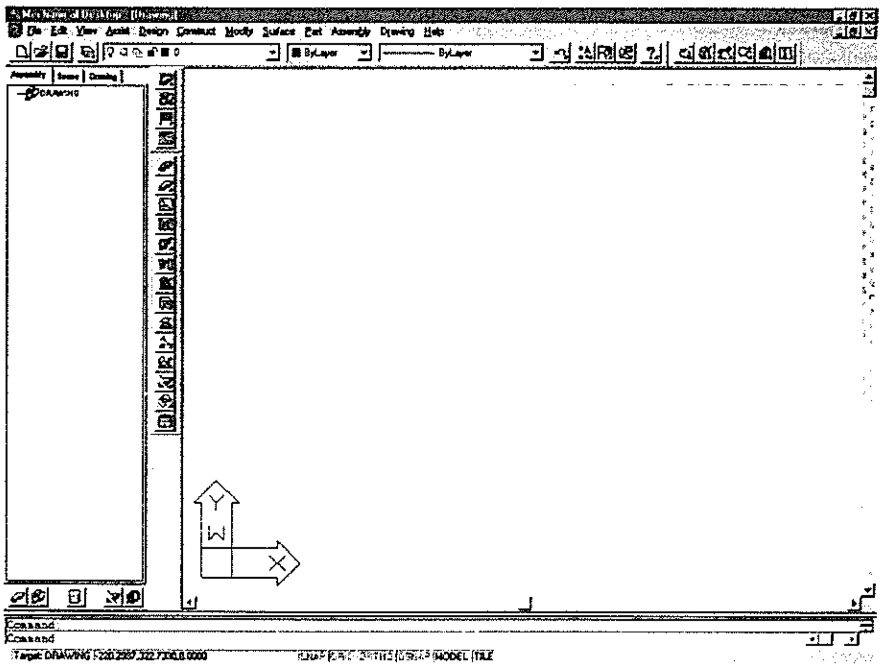


Figure 3.44 User interface of a multi-part drawing

Single-Part Drawing

To start a new drawing that has a single solid part, you select the New Part File item of the pull-down menu. Figure 3.45 shows the user interface for a single-part drawing.

<File> **<New Part File>**

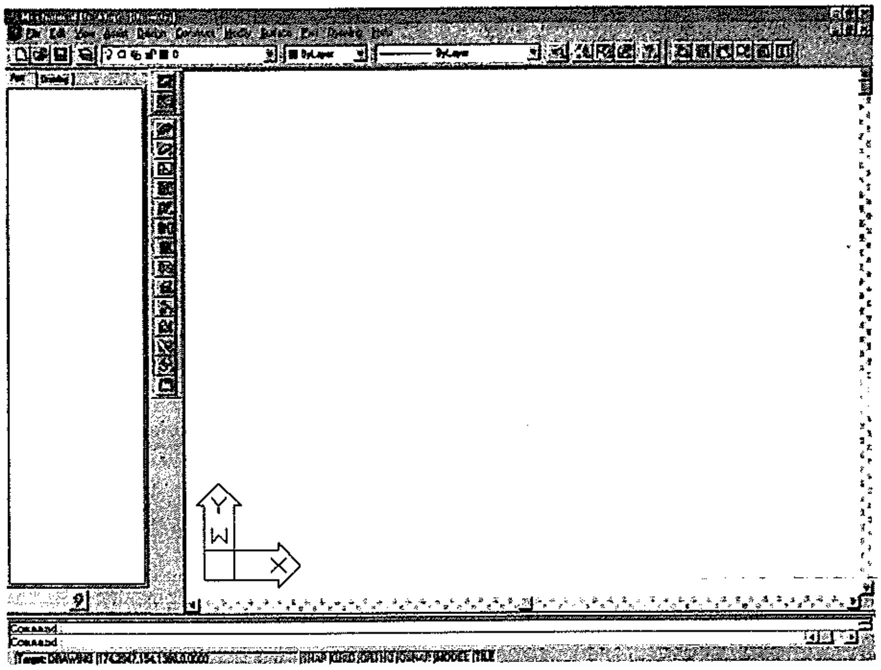


Figure 3.45 User interface of a single-part drawing

Compare Figure 3.45 with Figure 3.44. In Figure 3.44, the Desktop Browser (shown at the left side) has three tabs (Assembly, Scene, and Drawing) and the menu bar has four Mechanical Desktop pull-down menus (Surface, Part, Assembly, and Drawing). In Figure 3.45, the Desktop Browser has only two tabs (Part and Drawing) and the menu bar has three Mechanical Desktop pull-down menus (Surface, Part, and Drawing).

Using the interface shown in Figure 3.44, you can construct multiple solid parts in a single drawing file and assemble solid parts together. Using the interface shown in Figure 3.45, you can construct only a single solid part. Because there is only one solid part in the drawing, assembly is not possible.

Now you have a general understanding of parametric solid modeling and have learned ways to set up a new drawing. In Sections 3.4 through 3.8, you will work on a number of projects to learn the sketching approach to constructing an extrude solid, a revolve solid, a sweep solid (2D path), a 3D helical sweep solid, and a loft solid, and to split faces of a solid.

3.4 Sketching Approach — Extrude

The simplest kind of solid feature is the extrude solid. An extrude solid is constructed by extruding a 2D profile in a direction perpendicular to the plane of the profile.

Mounting Block Project

Figure 3.46 shows the rendered image of a mounting block. It is an extrude solid. To make this model, you first make a rough sketch that resembles the top view of the model. Then you resolve it to a profile. After resolving, you examine the profile to find out what geometric constraints are applied. Then you remove the constraints that you do not want and add constraints that have not been added automatically. When you are satisfied with the shape of the profile, you add parametric dimensions to set the size and fully constrain it. When you have a properly constrained profile, you extrude it and it becomes an extrude solid.

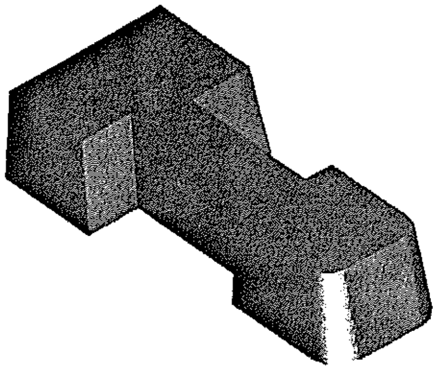


Figure 3.46 Rendered image of a mounting block

Start a new part file and use Solid.dwt as the template.

<File> **<New Part File>**

Template File: **Solid.dwt**

Note that you can construct only a single solid part in a single-part drawing. You should find the Desktop Browser (Figure 3.47) on your screen.

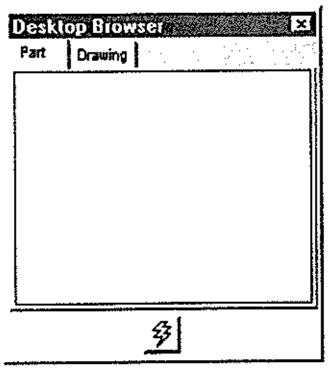


Figure 3.47 Desktop Browser for a single-part drawing

In this browser, there are two tabs: Part and Drawing. (We will explain the use of the Desktop Browser in detail in Section 3.9.) If the Desktop Browser is not displayed on your screen, you can select the Desktop Browser item from the View pull-down menu.

<View> **<Desktop Browser>**

Because this project has only one solid feature (extrude solid), you can use the default XY plane of the WCS as the sketch plane. As shown in Figure 3.48, construct a rough sketch by using the PLINE command. Lines need not be absolutely horizontal, vertical, or tangential, and end points do not have to meet.

<Design> **<Polyline>**

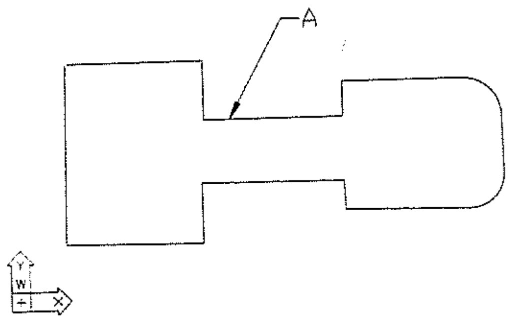


Figure 3.48 Rough sketch

Now resolve the sketch to a profile by selecting the Profile item of the Sketch cascading menu of the Part pull-down menu. Because the rough sketch shown in Figure 3.48 may not be exactly the same as yours, the geometric constraints that Mechanical Desktop automatically applies to your profile may be different. Consequently, the number of geometric constraints or parametric dimensions that you need to add in order to constrain the profile fully may be different from that shown in the illustration below. Here, ten dimensions or constraints are required.

<Part> **<Sketch>** **<Profile>**

Command: **AMPROFILE**
Select objects for sketch:
Select objects: **[Select A (Figure 3.48).]**
Select objects: **[Enter]**
Solved under constrained sketch requiring 10 dimensions or constraints.

If there is only one object on your screen (one polyline or one circle), you can select the Single Profile item to resolve a single sketch without having to select it.

<Part> **<Sketch>** **<Single Profile>**

After you resolve a sketch to a profile, the item Profile appears in the Desktop Browser. (See Figure 3.49.) It denotes the resolved profile (Figure 3.50) of the solid part.

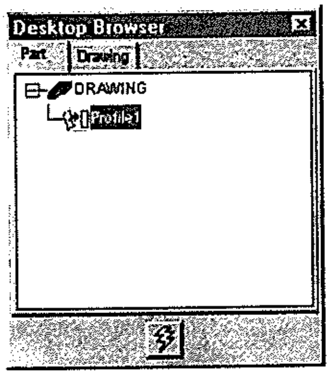


Figure 3.49 Profile in the Desktop Browser

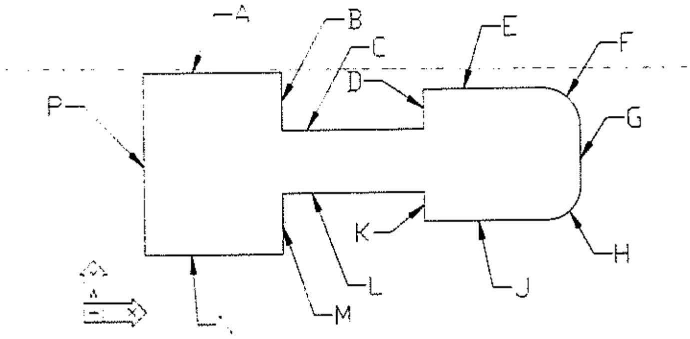


Figure 3.50 Resolved sketch

After resolving, a point on the sketch that is marked with a white box is set as the fixed point. As the name implies, this point will not move. Optionally, you can set another point on the sketch as the fixed point. Now use the AMFIXPT command to set the end point near A (Figure 3.50) as the fixed point.

<Part> **<Sketch>** **<Fix Point>**

Command: **AMFIXPT**
Specify new fixed point for sketch: **[Select A (Figure 3.50).]**

To find out what geometric constraints are applied to your drawing, use the AMSHOWCON command. (See Figure 3.51.) As we have said, the resolved sketch shown here may differ from yours. Therefore, the kinds of constraints applied may be different.

<Part> **<Sketch>** **<Show Constraints>**

Command: **AMSHOWCON**

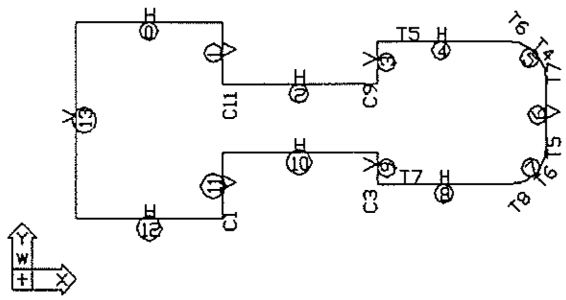


Figure 3.51 Display of geometric constraint symbols

In Figure 3.51, you can see numbers in circles and alphabetic symbols. The numbers denote the entity names. The letters denote the geometric constraints. The following list gives the meanings of all the alphabetic constraint symbols.

H Horizontal constraint
V Vertical constraint
L Perpendicular constraint
P Parallelism constraint
X X-values constraint for center points and end points of lines
Y Y-values constraint for center points and end points of lines
C Collinear constraint
N Concentric constraint
J Projected constraint
R Same-radius constraint
T Tangential constraint
E Equal-length constraint

Let us suppose that we have the following requirements:

Lines B and M (Figure 3.50)	not collinear
Lines D and K (Figure 3.50)	collinear
Lines A, C, E, J, L, and N (Figure 3.50)	horizontal
Lines B, D, G, K, M, and P (Figure 3.50)	vertical
Arcs F and H (Figure 3.50)	equal radii
Arc F and line E (Figure 3.50)	tangent
Arc F and line G (Figure 3.50)	tangent
Arc H and line G (Figure 3.50)	tangent
Arc H and line J (Figure 3.50)	tangent

In accordance with the constraint requirements stated above, remove the constraints that are not required by using the AMDELCON command.

<Part> **<Sketch>** **<Delete Constraints>**

Command: **AMDELCON**
Size/All/<select>: **[Select the geometric constraint symbol that you want to remove. Here, select the collinear constraint of line B or line M (Figure 3.50).]**

To add geometric constraints, use the AMADDCON command by selecting the Add Constraints cascading menu of the Sketch item of the Part pull-down menu. (See Figure 3.52.)

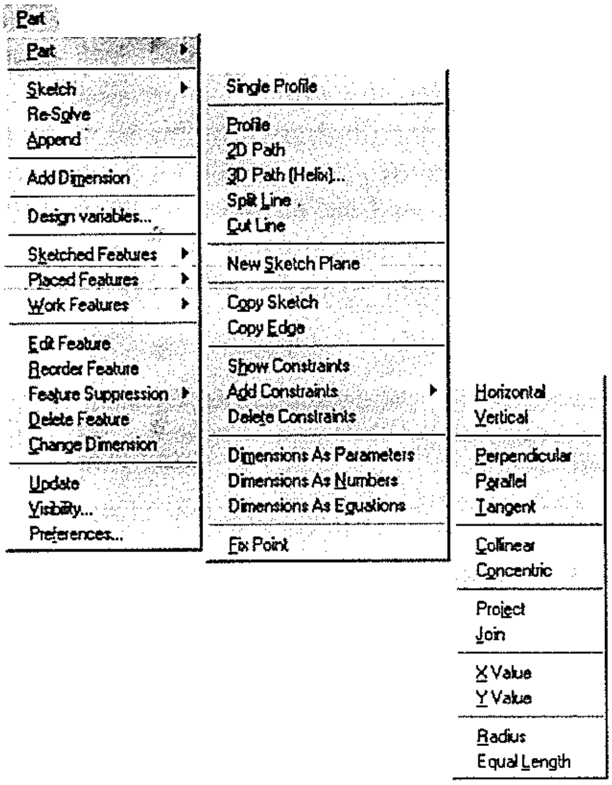


Figure 3.52 Add Constraints cascading menu

To add a horizontal constraint, select the Horizontal item from the Add Constraint cascading menu.

<Part> **<Sketch>** **<Add Constraints>** **<Horizontal>**

Command: **AMADDCON**
Select line: **[Select A (Figure 3.50).]**

Depending on whether the constraint has already been applied, you get one of the following prompts:

This constraint already exists.
Solved under constrained sketch requiring ?? dimensions or constraints.

The "??" in the second prompt is a number that decreases by 1 each time a constraint is added. In accordance with the requirements mentioned earlier, continue to constrain the sketch.

Parametric dimensions, like other AutoCAD dimensions, are governed by the AutoCAD dimension variables. You may directly set the variable values at the command line or use the DDIM command to make the appropriate changes.

After setting the dimension style, you have to decide how the parametric dimensions are displayed. There are three display methods: numeric display, parameter display, and equation display. Numeric display will tell you the exact dimension value. Parameter display will tell you the parameter name assigned to the dimension. The first dimension that you add in a drawing is called d0, the second d1, and so on. Depending on the sequence of dimensioning, the parameter names that are displayed on your screen may not be the same as those shown in the illustration in this book. Therefore, you may want to write down the parameter names of your drawing on a separate piece of paper while you are working, if they are different from those in the illustration. In the third type of display, the equation display, the dimension value is expressed as an equation with the parameter name equal to an expression. Run the AMDIMDSP command and choose the equation display.

<Part> **<Sketch>** **<Dimensions As Equations>**

Command: **AMDIMDSP**
Parameters/Equations/<Numeric>: **E**

Now add dimensions to the profile. Figure 3.53 shows the dimensions that you need to add.

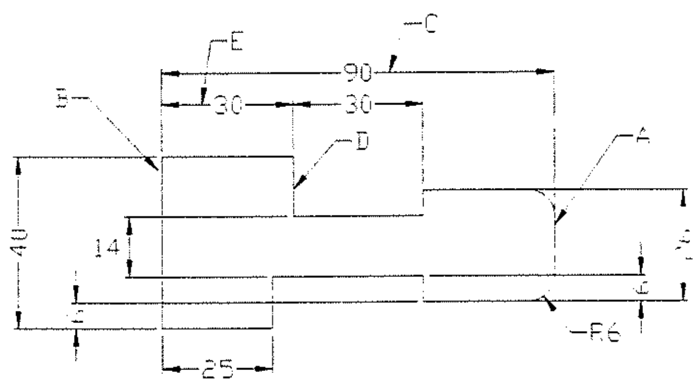


Figure 3.53 Dimensions to be added

Let us start with the outermost dimension. Select the Add Dimension item from the Part pull-down menu to use the AMPARDIM command. After you select the objects and the dimension placement position, the measured dimension value is displayed at the command line. The dimension value "???" shown below is the exact size of the dimension. To change it to 90 units as required (Figure 3.53), you can type 90 at the command line. However, you should first check the measured value against the required value. If the measured value is much larger, say 200, and you type 90, the dimension will shrink from 200 units to 90 units. Because the other dimensions remain unchanged, the profile will be distorted and the result may resemble Figure 3.54. Therefore, do not type the value 90 if the measured value is much larger. Instead, type U to use the undo option.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.53).**]
Select second object or place dimension: [**Select B (Figure 3.53).**]
Specify dimension placement: [**Select C (Figure 3.53).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value <???>:

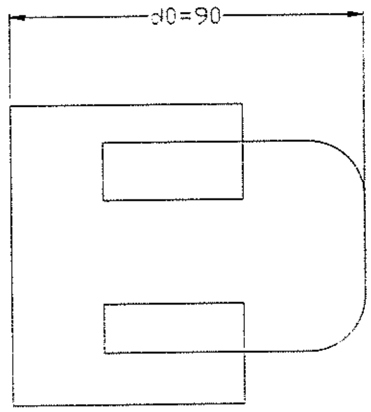


Figure 3.54 Outer dimension shrunk and profile distorted

On the other hand, if the measured dimension is less than 90 and you type in a value of 90, the outer dimension stretches. The result will resemble Figure 3.55. Here the shape is not changed. Therefore, you can continue to add the remaining dimensions.

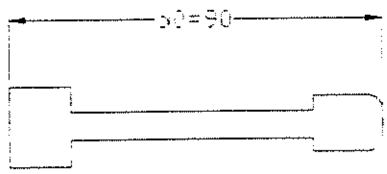


Figure 3.55 Outer dimension stretched, others unchanged

As a rule of thumb, compare the measured dimension that is shown at the command line against the required dimension before inputting the required dimension value. If you start dimensioning the outermost dimension of the profile and the measured value is much larger than the required value, you should undo the command. Otherwise, it may distort the profile and change the shape. To tackle sketches that are constructed much larger than required, you should start dimensioning the smallest dimension as follows:

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select B (Figure 3.53).**]
Select second object or place dimension: [**Select D (Figure 3.53).**]
Specify dimension placement: [**Select E (Figure 3.53).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value <???>:

Continue to add dimensions as shown in Figure 3.53 until you get the prompt "Solved fully constrained sketch." To exit the command, press the [Enter] key. Because your sequence of dimensioning may not be the same, the parameter names of the dimensions may not be the same as those shown in Figure 3.56.

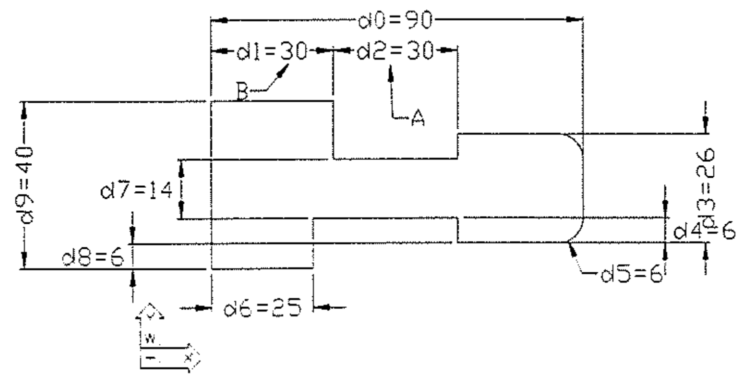


Figure 3.56 Fully constrained profile

As you can see, the dimensions A and B (Figure 3.56) are the same. Instead of specifying the same dimension value, you can set dimension A (Figure 3.56) equal to dimension B (Figure 3.56).

In Figure 3.56, the parameter name for dimension B is d1. Check your drawing. The parameter name of dimension B (Figure 3.56) may not be d1; it can be d?, in which ? is a number that depends on your dimensioning sequence. d0 is the first dimension in a drawing, d1 is the second dimension, and so forth. You should replace d1, below, with the parametric name of dimension B (Figure 3.56) in your drawing.

To set dimension A equal to dimension B, use the AMMODDIM command, select dimension A and enter the parameter name d1. (See Figure 3.57.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 3.56).**]
New value for dimension: **d1**
Select dimension to change: [**Enter**]

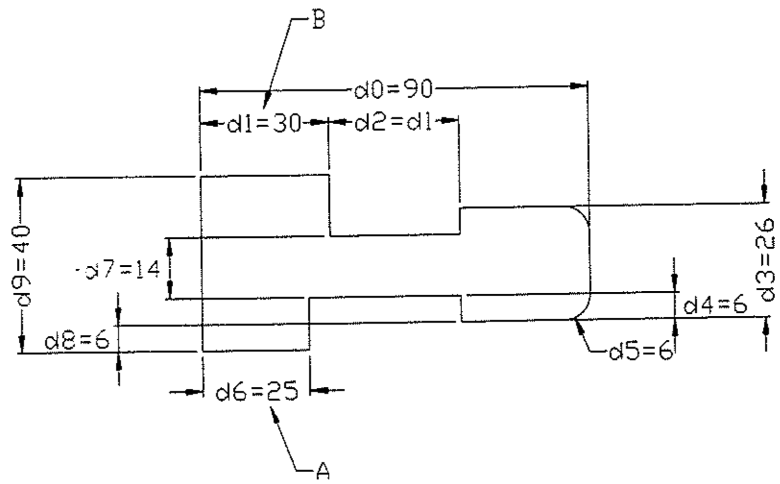


Figure 3.57 Dimension d2 set equal to dimension d1

Now change two dimensions. (See Figure 3.58.) The profile is complete.

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: **[Select A (Figure 3.57).]**
New value for dimension: **20**
Select dimension to change: **[Select B (Figure 3.57).]**
New value for dimension: **35**
Select dimension to change: **[Enter]**

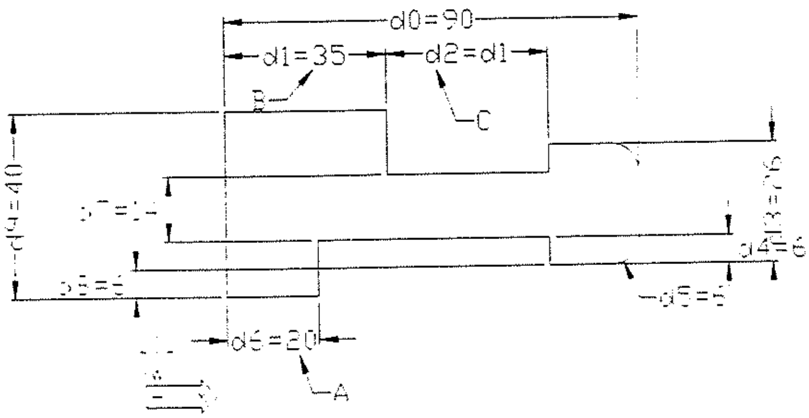


Figure 3.58 Dimensions changed

Use the Mechanical Main toolbar to set the current layer to Solid. Then set the display to an isometric view by using the shortcut key [8]. After that, use the AMEXTRUDE command. (See Figure 3.59, the Extrusion Feature dialog box.) There are five kinds of operations: Base, Cut, Join, Intersect, and Split. As we have explained, complex solid models are composed from sketched solid features by cutting, joining, and intersecting. And you can split up a solid into two separate solids. The first solid feature constructed in a solid part is called the base solid feature. Because it is the first feature and there are no other features in this drawing, only one kind of operation, Base, is available. When extruding, you have to specify how the extrusion operation terminates. For the base feature, you can select only Blind or MidPlane. To extrude Blind, the solid is extruded in one direction perpendicular to the profile. To extrude MidPlane, the solid is extruded in two directions perpendicular to the profile. Select Blind. Then set Distance to 20 and Draft Angle to 0. After that, select the [OK] button. (See Figure 3.60.) The arrow shows the direction of extrusion. To accept, press the [Enter] key. To flip, type [F] and then press the [Enter] key. Now accept the direction. The profile is extruded. (See Figure 3.61.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

Command: **8**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

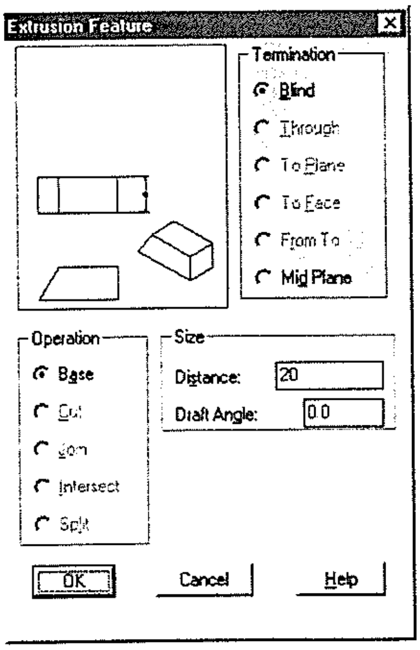


Figure 3.59 Extrusion Feature dialog box

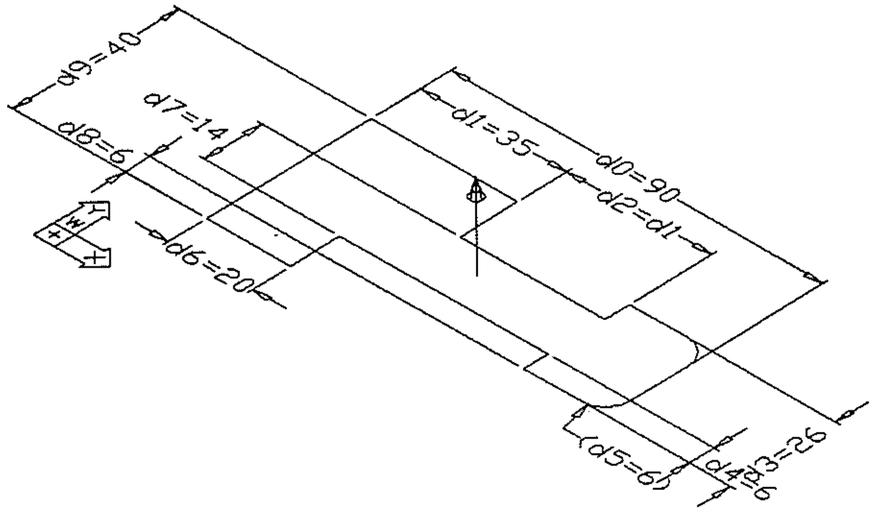


Figure 3.60 Arrow showing the direction of extrusion

Direction Flip/<Accept>: **[Enter to accept if the direction arrow is pointing upward.]**

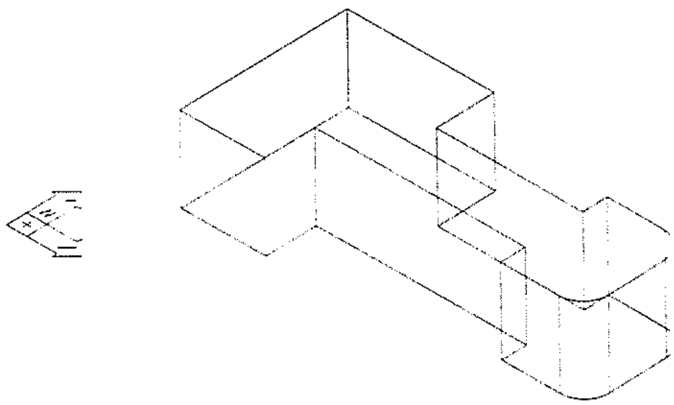


Figure 3.61 Profile extruded

Unlike native solids, which become static after being extruded, a parametric solid can be modified at any time as required. To modify a solid, you can use the AMEDITFEAT command by selecting the Edit Feature item of the Part pull-down menu, or select the feature name in the Desktop Browser and right-click.

Move the mouse cursor within the Desktop Browser. Select the feature named Extrusion Blind 1 and right-click. (See Figure 3.62.) A pop-up menu is displayed. Select the item Edit to run the AMEDITFEAT command. The parametric dimensions appear on your screen. (See Figure 3.63.) Now change the values of the dimensions A, B, and C (Figure 3.63).

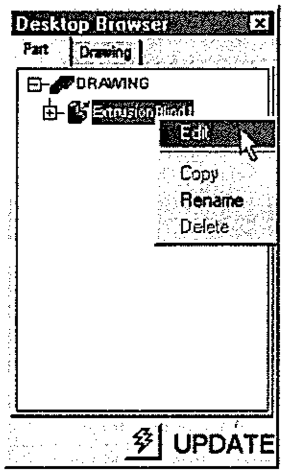


Figure 3.62 Right-click the feature name in the Desktop Browser.

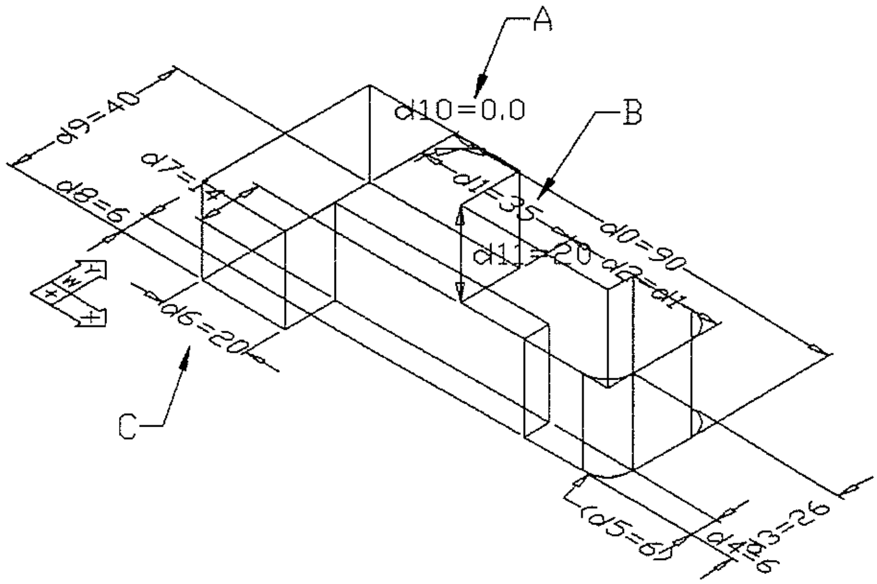


Figure 3.63 Parametric dimensions shown

Command: **AMEDITFEAT**
Select object: **[Select A (Figure 3.63).]**
Enter new value for dimension: **-5**
Select object: **[Select B (Figure 3.63).]**
Enter new value for dimension: **30**
Select object: **[Select C (Figure 3.63).]**
Enter new value for dimension: **25**
Select object: **[Enter]**

Unlike the dimension values in the profile, the solid is not updated automatically because updating may take considerable time. If updating is automatic, you will have to wait many times as you change the dimensions of the solid. To update the solid, you can use the AMUPDATE command. To run this command, you can select the Update item of the Part pull-down menu or select the Update button of the Desktop Browser (Figure 3.62). (See Figure 3.64.)

<Part> **<Update>**

Command: **AMUPDATE**

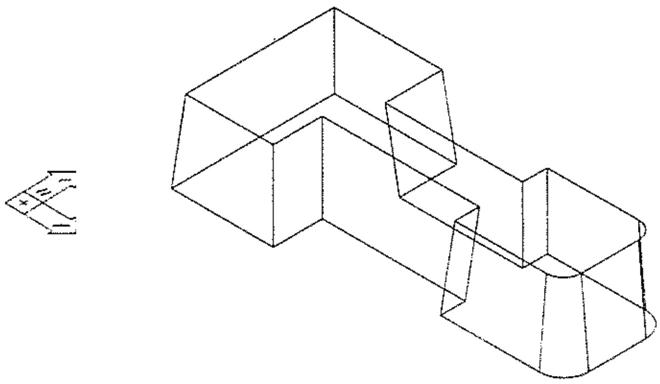


Figure 3.64 Solid updated

The solid model is complete. Save your drawing.

<File> **<Save>**

File name: **Mblock.dwg**

In making this extrude solid, you learned how to make a sketch, resolve the sketch to a profile, delete unwanted geometric constraints, add appropriate constraints, add parametric dimensions, modify parametric dimensions, extrude the profile, and modify the solid after extrusion.

3.5 Sketching Approach — Revolve

The second way to construct a solid feature from a sketch is to revolve a profile about an axis. An axis can be a work axis (a kind of work feature) or a line that is part of the sketch.

Shock Absorber Project

Figure 3.65 shows the rendered image of the cylinder of the shock absorber of a scale model car. You will construct this solid part in two stages. First you will construct the main body of the cylinder. Then, later in this chapter, you will cut a screw thread in it. Study the model carefully. You will find that it is a solid constructed by revolving a profile about an axis. Figure 3.66 shows the dimensions of the profile.

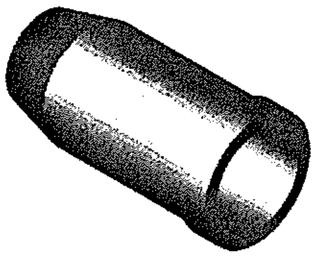


Figure 3.65 Rendered image of the cylinder of the shock absorber

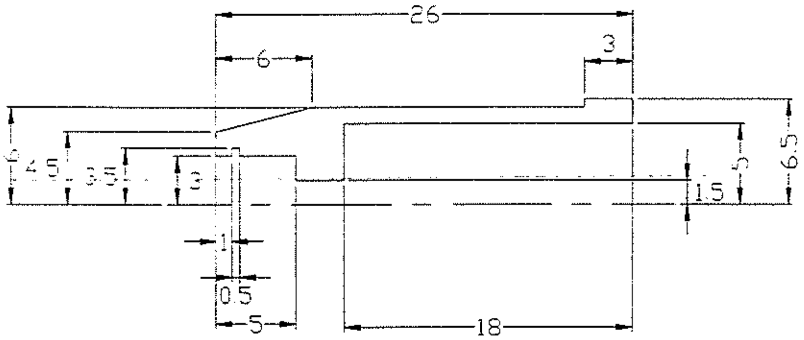


Figure 3.66 Dimensions of the profile for revolving

The shock absorber has a number of solid parts. To construct them in a single drawing file, start a new multi-part drawing, using Solid.dwt as the template. Note that for a multi-part drawing, the Desktop Browser has three tabs (Assembly, Scene, and Drawing). We will discuss them in detail in Section 3.9.

<File> **<New... >**

Template File: **Solid.dwt**

To have better control of the size of the sketch, you can set the drawing limits to an area of 40 units by 30 units and then zoom the display to those limits. This way, the display area approximates 40 units by 30 units.

<Assist> **<Format>** **<Drawing Limits>**

Command: **LIMITS**
ON/OFF/<Lower left corner>: **0,0**
Upper right corner: **40,30**

<View> **<Zoom>**

Zoom Extents

To revolve a profile to create a revolve solid, you need an axis. There are two ways to define an axis: You can make a construction line and include it in the sketch, or you can construct a work axis on the current sketch plane before making the sketch.

Construction Line

Construction lines are part of the resolved sketch. They play a vital part in constraining the profile. You can use one as an axis of revolution, and use it in conjunction with other lines to apply geometric constraints and parametric dimensions to the sketch. Mechanical Desktop will not include the construction lines for subsequent operations.

While you make a sketch, linetypes other than the one specified in the Apply to Linetype box of the Part tab of the Desktop Preferences dialog box are treated as construction lines. As shown in Figure 3.67, use the PLINE command to construct a polyline and the LINE command to construct a line. Then change the linetype of line A (Figure 3.67) to hidden (or any linetype other than Continuous).

<Design> **<Polyline>**

<Design> **<Line>**

<Edit> **<Properties...>**

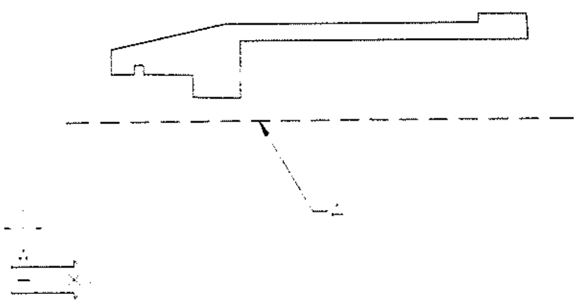


Figure 3.67 Freehand sketch

Work Axis

Another approach is to construct a work axis. Use the AMWORKAXIS command. (See Figure 3.68.) Then make the sketch. (See Figure 3.69.)

<Part> **<Work Features>** **<Work Axis>**

Command: **AMWORKAXIS**
Locate first point: **[Select any point near the lower left corner of your screen.]**
Locate second point: **@40<0**

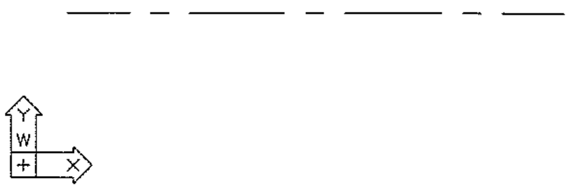


Figure 3.68 Work axis constructed

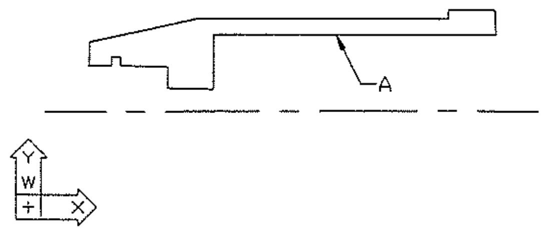


Figure 3.69 Sketch constructed

Resolving

Whether you use a construction line included in the sketch or construct a sketch and a work axis, you need to use the AMPROFILE command to resolve the sketch to a profile. (See Figure 3.70.) Depending on how rough your sketch is, the number of dimensions/constraints required, shown here as "??", is different.

<Part> **<Sketch>** **<Profile>**

Command: **AMPROFILE**
Select objects for sketch:
Select objects: **[Select all the lines (Figure 3.67) or select A (Figure 3.69).]**
Select objects: **[Enter]**
Solved under constrained sketch requiring ?? dimensions or constraints.

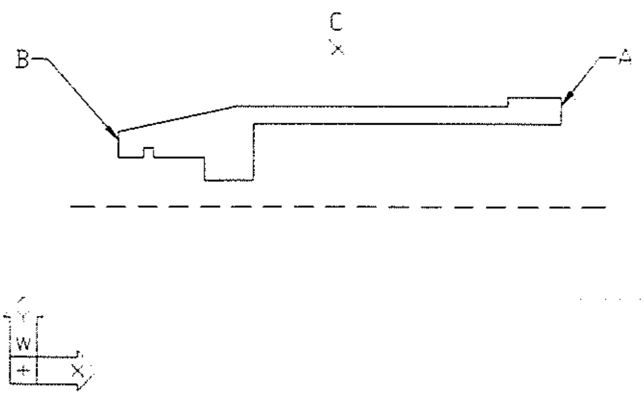


Figure 3.70 Resolved sketch

Now use the AMSHOWCON command to display the geometric constraint symbols. Then use the AMDELCON command to remove the unwanted geometric constraints and use the AMADDCON command to add the required vertical and horizontal geometric constraints. After that, use the AMPARDIM command to add a parametric dimension. (See Figure 3.71.)

<Part> <Sketch> <Show Constraints>

<Part> <Sketch> <Delete Constraints>

<Part> <Sketch> <Add Constraints> <Vertical>

<Part> <Sketch> <Add Constraints> <Horizontal>

<Part> <Add Dimension>

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.70).]**
Select second object or place dimension: **[Select B (Figure 3.70).]**
Specify dimension placement: **[Select C (Figure 3.70).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **26**

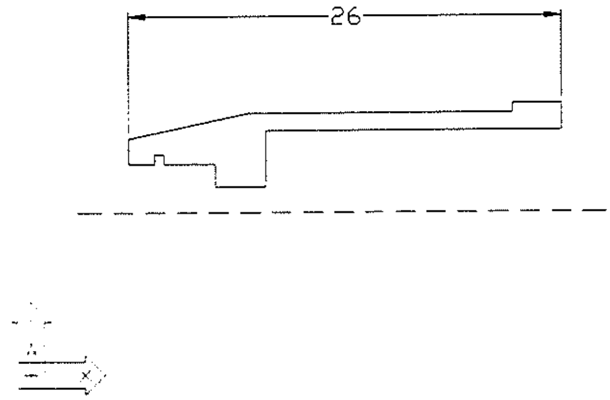


Figure 3.71 A parametric dimension added

To continue, use the AMPARDIM command to complete the parametric dimensions as shown in Figure 3.72. You will get the message "Solved fully constrained sketch" at the command line when the profile is properly constrained. To exit the command, press the [Enter] key. After adding dimensions, set the current layer to Solid.

<Part> <Add Dimension >

[Mechanical Main] [Layer Control]

Current Layer: **Solid**

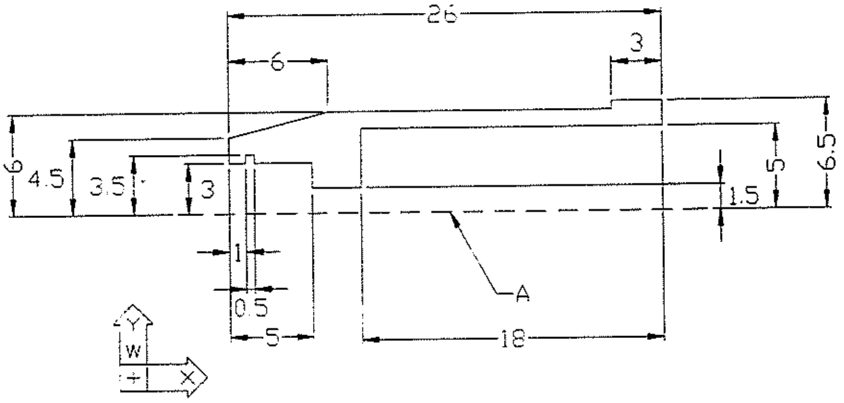


Figure 3.72 Parametric dimensions added

Set the display to an isometric view. Then use the AMREVOLVE command to revolve the profile to create a solid. (See Figure 3.73, the Revolution Feature dialog box.) There are five operations. Here the solid feature is the first sketched solid. There is only one option, Base. For Termination, select Full. Then select the [OK] button. After that, select the revolution axis. (See Figure 3.74.)

Command: 8

<Part> <Sketched Features> <Revolve...>

Command: **AMREVOLVE**

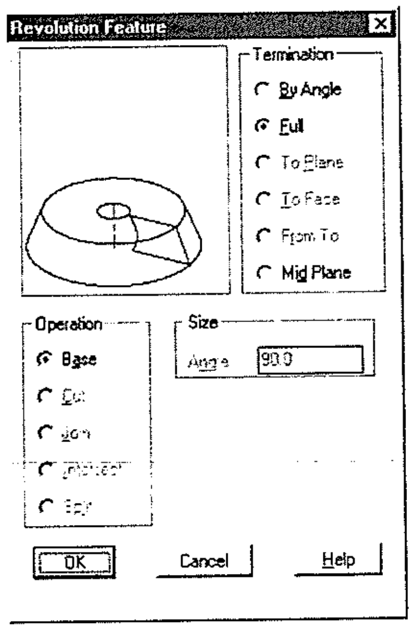


Figure 3.73 Revolution Feature dialog box

Select revolution axis: [**Select A (Figure 3.72).**]

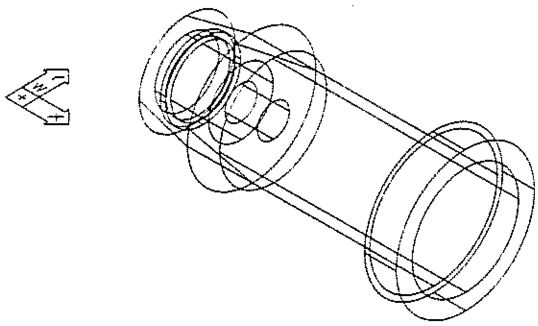


Figure 3.74 Revolve solid

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Shock.dwg**

In making this revolve solid, you learned how to revolve a resolved sketch to create a revolve solid. In addition, you learned how to include a construction line in a sketch, how to use the construction line to constrain the sketch and as an axis, and how to construct a work axis.

3.6 Sketching Approach — Sweep

A sweep solid is a solid created when a profile is swept along a path. There are two kinds of sweep solids. The first is constructed by sweeping a profile along a 2D path. The profile is placed at the start point of the path and is perpendicular to the path. The second kind is constructed by sweeping a profile along a 3D helical path. Here you will work on two projects: a swing arm and a helical spring. The swing arm is a sweep solid constructed by sweeping a profile along a 2D path. The spring is a sweep solid constructed by sweeping a profile along a 3D path.

Swing Arm Project

Figure 3.75 shows the rendered image of a swing arm. You will work on this project in two stages. First you will construct the sweep solid as shown in Figure 3.76. Then you will add more features to it later in this chapter.

To make this solid, you will construct two sketches and resolve one of them as a path and the other one as a profile. After constraining them fully, you will sweep the profile along the path. Figure 3.77 shows the dimensions of the path.

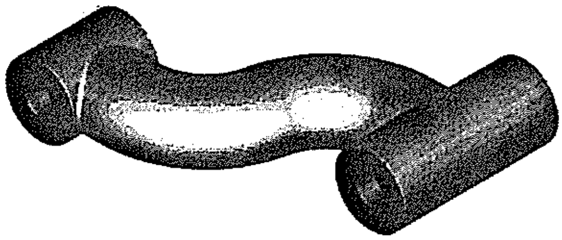


Figure 3.75 Swing arm

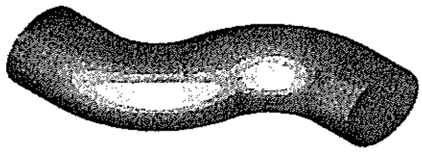
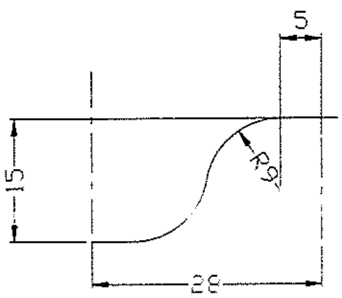


Figure 3.76 Rendered image of a sweep solid



3.77 Dimensions of the path

Start a new single-part drawing. Use the template Solid.dwt.

<File> **<New Part File>**

Template File: **Solid.dwt**

Figure 3.77 shows the dimensions of the path. The approximate size of the path is 30 units by 20 units. To better control the size of the freehand sketch, set the drawing limits to 40 units by 30 units. Then zoom the display to these limits.

<Assist> **<Format>** **<Drawing Limits>**

ON/OFF/<Lower left corner>: **0,0**
Upper right corner: **40,30**

<View> **<Zoom>**

Zoom Extents

As shown in Figure 3.78, construct a polyline A with two line segments and two arc segments. Then use the LINE command to construct two lines B and C. After that, change the linetypes of the lines B and C to hidden.

<Design> **<Polyline>**

<Design> **<Line>**

<Edit> **<Properties...>**

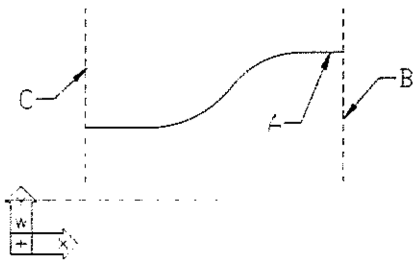


Figure 3.78 Sketch constructed

Now use the AMPATH command to resolve the sketch (including the construction lines) to a path. When resolving, you need to specify a start point for locating a work plane for the sweep profile and to decide whether to construct a work plane. Note that the number of dimensions/constraints required depends on the accuracy of each individual freehand sketch. (See Figure 3.79.)

<Part> **<Sketch>** **<Path>**

Command: **AMPATH**
Select objects: **[Select A, B, and C (Figure 3.78).]**
Select objects: **[Enter]**
Specify start point of path: **[Select A (Figure 3.78).]**
Do you want to create a workplane perpendicular to path? Yes/<No>: **NO**

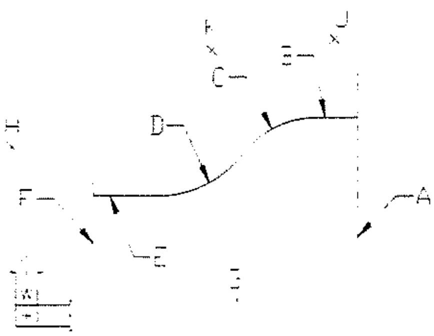


Figure 3.79 Resolved sketch

In accordance with the requirements listed below, add geometric constraints to confine the shape of the path.

Lines A and F (Figure 3.79)	vertical
Lines B and E (Figure 3.79)	horizontal
Arcs C and D (Figure 3.79)	equal radii
Arcs C and D (Figure 3.79)	tangent
Arc C and line B (Figure 3.79)	tangent
Arc D and line E (Figure 3.79)	tangent

After adding the appropriate geometric constraints, use the AMPARDIM command to add parametric dimensions. (See Figure 3.80.)

<Part> <Add Dimension>

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.79).]**
Select second object or place dimension: **[Select F (Figure 3.79).]**
Specify dimension placement: **[Select G (Figure 3.79).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **28**

Select second object or place dimension: **[Select B (Figure 3.79).]**
Specify dimension placement: **[Select J (Figure 3.79).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **5**

Select first object: **[Select B (Figure 3.79).]**
Select second object or place dimension: **[Select E (Figure 3.79).]**
Specify dimension placement: **[Select H (Figure 3.79).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **15**

Select first object: **[Select C (Figure 3.79).]**
Select second object or place dimension: **[Select K (Figure 3.79).]**
Undo/Diameter/Ordinate/Placement point/Enter dimension value: **9**
Select first object: **[Enter]**

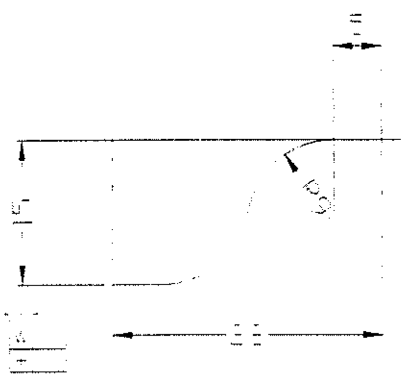


Figure 3.80 Parametric dimensions added

Now set the display to an isometric view. Then use the AMWORKPLN command to construct a work plane at the start point of the path and set the sketch plane to this work plane. To construct a work plane, you can specify one or two modifiers. (See Figure 3.81, the Work Plane Feature dialog box.) Check the Normal to Start box. Then select the Create Sketch Plane box and the [OK] button. When you return to the graphic screen, an icon that resembles a mouse appears at the cursor point. (See Figure 3.82.) While the red mouse-shaped button is blinking, the L-shaped symbol rotates. The purpose of the blinking red button is to tell you that you can change the orientation of the XY plane of the new sketch plane by pressing the left button of your mouse. Now press the left button to rotate and flip the XY plane until the orientation of the UCS icon is the same as that shown in Figure 3.83.

Command: **8**

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

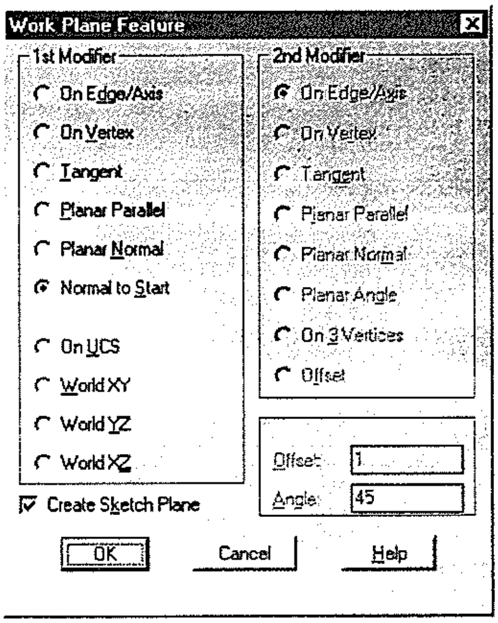


Figure 3.81 Work Plane Feature dialog box

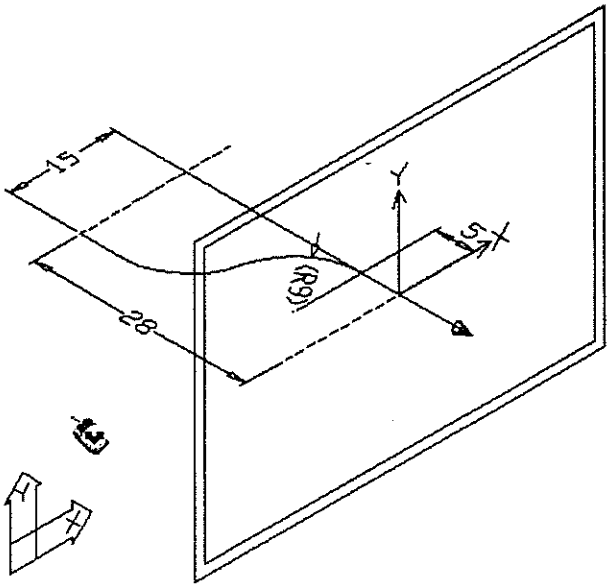


Figure 3.82 Blinking mouse icon

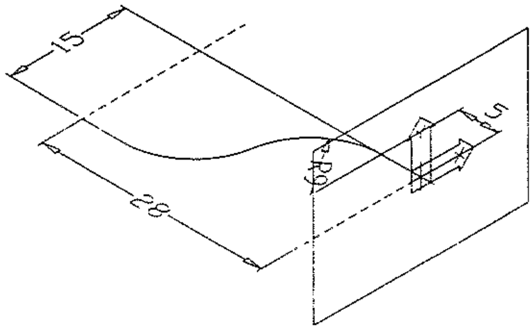


Figure 3.83 Work plane constructed at the start point of the path, sketch plane set

On the new sketch plane, construct an ellipse as shown in Figure 3.84. Then resolve it to a profile.

<Design>	**<Ellipse>**	**<Axis, End>**
<Part>	**<Sketch>**	**<Profile>**

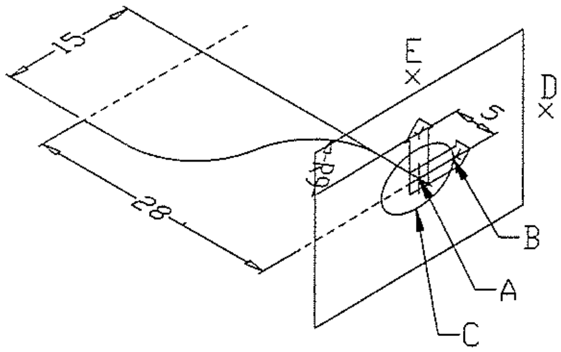


Figure 3.84 Ellipse constructed and resolved

Use the AMPARDIM command to add parametric dimensions to the ellipse. When dimensioning the major and minor axes of an ellipse, you have to observe two points: the object snap tool is needed to select the quadrant positions and the sequence of selection determines whether a major or a minor axis dimension is constructed. (See Figure 3.85.)

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.84).**]
Select second object or place dimension: [**Select C (Figure 3.84).**]
Specify dimension placement: [**Select E (Figure 3.84).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.84).**]
Select second object or place dimension: [**Select C (Figure 3.84).**]
Specify dimension placement: [**Select D (Figure 3.84).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: **QUA** of [**Select B (Figure 3.84).**]
Select second object or place dimension: **QUA** of [**Select C (Figure 3.84).**]
Specify dimension placement: [**Select E (Figure 3.84).**]
Undo/Ordinate/Placement point/Enter dimension value: **4.5**

Select first object: **QUA** of [**Select C (Figure 3.84).**]
Select second object or place dimension: **QUA** of [**Select B (Figure 3.84).**]
Specify dimension placement: [**Select D (Figure 3.84).**]
Undo/Ordinate/Placement point/Enter dimension value: **3.5**
Select first object: [**Enter**]

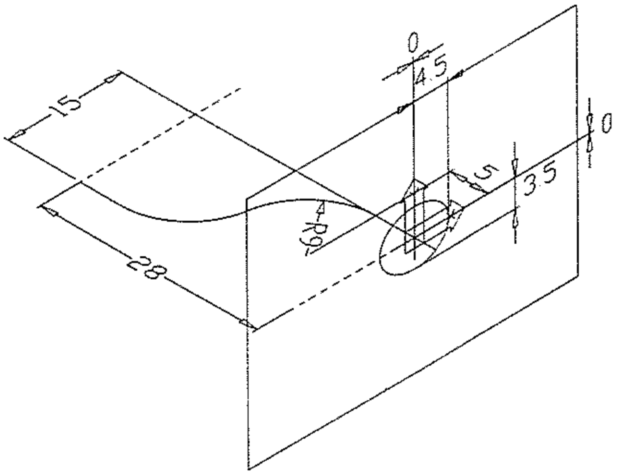


Figure 3.85 Ellipse dimensioned

Set the current layer to Solid. Then use the AMSWEEP command. (See Figure 3.86.) Select Path Only and Normal body type. Then select the [OK] button. (See Figure 3.87.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketched Features>** **<Sweep...>**

Command: **AMSWEEP**

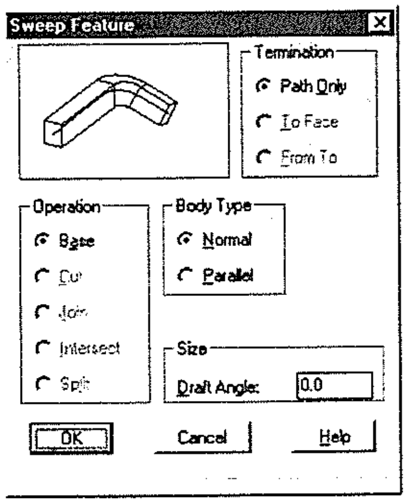


Figure 3.86 Sweep Feature dialog box

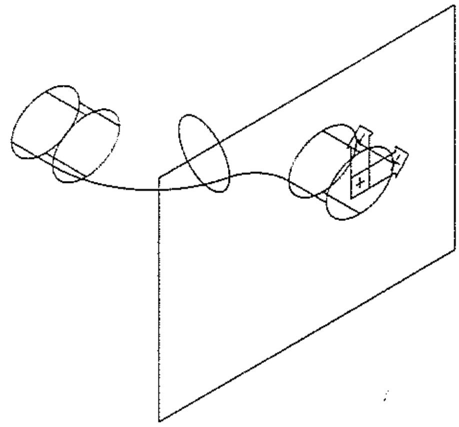


Figure 3.87 Sweep solid constructed

The work plane is not needed. Because it is part of the solid model, you cannot delete it. However, you can hide it. Use the AMVISIBLE command. (See Figure 3.88, the Desktop Visibility dialog box.) Check the Hide and Work Planes boxes. Then select the [OK] button. The work plane is hidden.

<Part> **<Visibility...>**

Command: **AMVISIBLE**

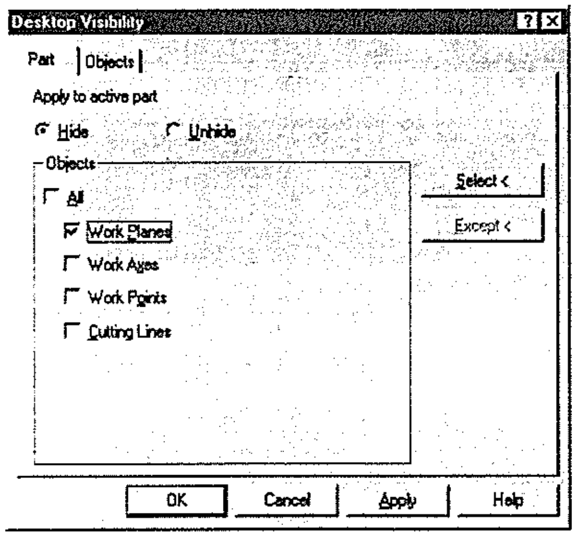


Figure 3.88 Part tab of the Desktop Visibility dialog box

By default, solid models are displayed in wireframe mode without silhouettes. To display a silhouette, select the Preferences... item of the Assist pull-down menu. (See Figure 3.89, the Preferences dialog box.) Select the Performance tab. Then check Show silhouettes in the wireframe box. After that, select the [OK] button.

<Assist> **<Preferences...>**

Command: **PREFERENCES**

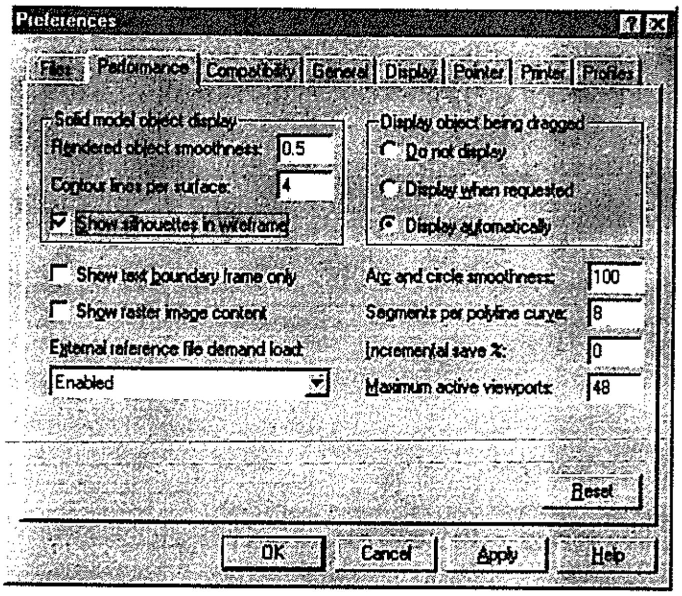


Figure 3.89 Preferences dialog box

For the new display setting to take effect, perform a regeneration. (See Figure 3.90.) The model is complete. Save your drawing.

Command: **REGEN**

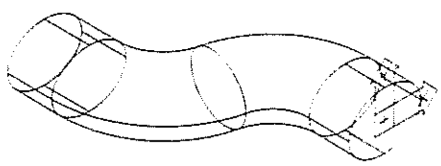


Figure 3.90 Work plane hidden, silhouette displayed

<File> **<Save>**

File name: **Arm_u.dwg**

In making this sweep solid, you learned how to resolve a sketch to a 2D path, select a start point on the path, construct a work plane at the start point of the path, construct a profile on a work plane at the start point, and sweep a profile along a path.

Helical Spring Project

In Chapter 2, you constructed a helical spring by using a helical sweep surface to cut a native solid. Although you can use the same helical surface to cut a parametric solid, you can construct a solid helical spring more directly. Figure 3.91 shows the rendered image of a helical spring that is constructed by sweeping a profile along a specified 3D helical path. Start a new part file and use Solid.dwt as the template.

<File> **<New Part File>**

Template File: **Solid.dwt**

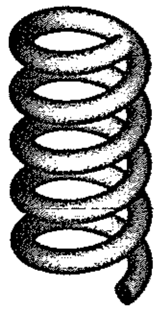


Figure 3.91 Rendered image of a helical spring

To construct a helical spring, you need a 3D helical path. A helical path can be constructed simply by specifying an axis or a circular object. Because this is the first solid feature of the drawing and there is no circular object, you will construct a work axis. To construct a work axis in the Z direction, you need to construct a work plane in the XZ plane. Use the shortcut key [8] to set the display to an isometric view. Then use the AMWORKPLN command to construct a work plane. From the Work Plane Feature dialog box, check the boxes World XZ and Create Sketch Plane. Then select the [OK] button. (See Figure 3.92.)

Command: **8**

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **World XZ**
Create Sketch Plane
OK]

worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**

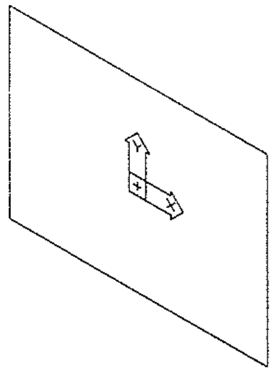


Figure 3.92 Work plane constructed on the XZ plane

On the new work plane, use the AMWORKAXIS command to construct a work axis. (See Figure 3.93.) Note that the length 40 in the following illustration is an arbitrary number and is unimportant. It simply specifies a nonzero length.

<Part> **<Work Features>** **<Work Axis>**

Command: **AMWORKAXIS**
Locate first point: **0,0**
Locate second point: **@40<90**

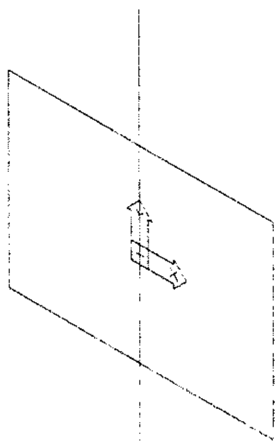


Figure 3.93 Work axis constructed

After constructing a work axis, use the AMWORKPLN command to construct a work plane on the XY plane of WCS. (See Figure 3.94.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **World XY**

Create Sketch Plane
OK]

Z-flip/Rotate/<Select edge to align X axis>: **[Enter]**

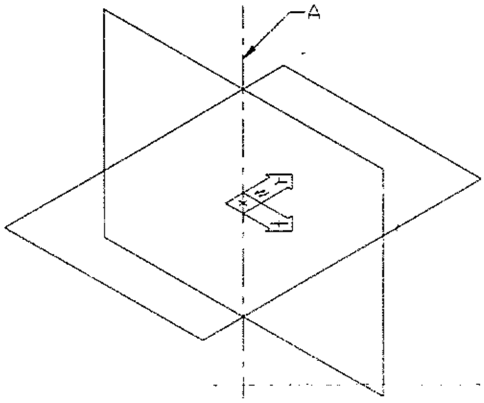


Figure 3.94 Work plane constructed on XY plane of WCS

Now use the AM3DPATH command to construct a 3D helical path. (See Figure 3.95, the Helix dialog box.) In the Type box, specify the Pitch and Revolution in the pull-down list, set the number of revolutions to 5, and set the pitch to 7 units. In the Shape box, specify a circle in the pull-down list to construct a circular helical path, and set the diameter to 13.5 units. Select the [OK] button. (See Figure 3.96.) Note that there is a work point at the start of the 3D helical path.

<Part> **<Sketch>** **<3D Path (Helix)...>**

Command: **AM3DPATH**
Select work axis, circular edge, or circular face for helical center: **[Select A (Figure 3.94).]**

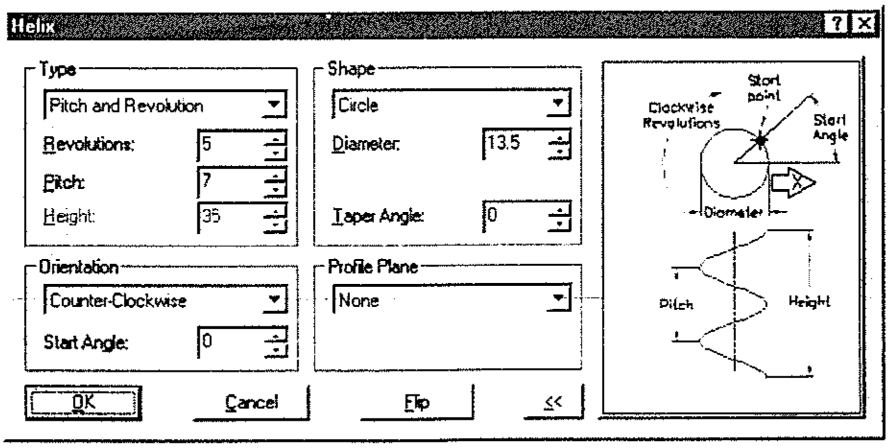


Figure 3.95 Helix dialog box

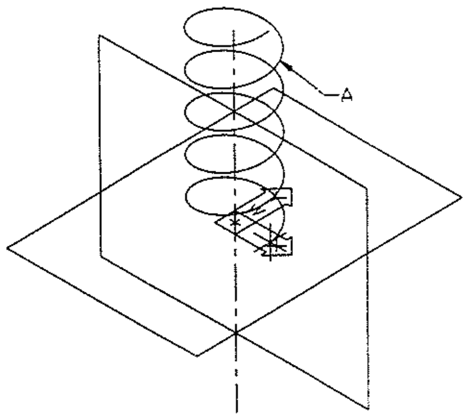


Figure 3.96 3D helical path constructed

Use the AMWORKPLN command to construct a work plane at the start point of the 3D path and set the sketch plane to that work plane. Then construct a circle as shown in Figure 3.97.

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Normal to Start**
Create Sketch Plane
OK]

Select path or sketched work axis: **[Select A (Figure 3.96).]**
Z-flip/Rotate/<Select edge to align X axis>: **[Accept if the UCS orientation is the same as that shown in Figure 3.97.]**

<Design> **<Circle>** **<Center, Radius>**

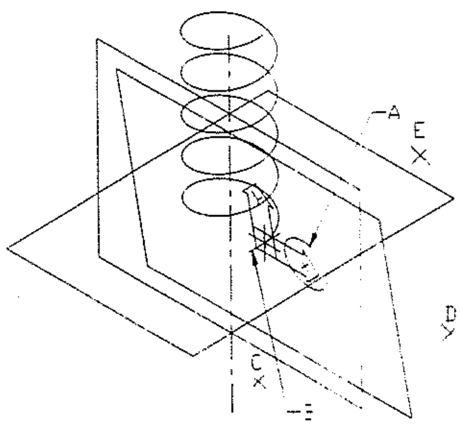


Figure 3.97 Sketch plane set, circle constructed

Use the AMPROFILE command to resolve the circle. Then use the AMPARDIM command to add parametric dimensions to constrain the profile. (See Figure 3.98.) Note that point B (Figure 3.97) is the work point at the start point of the 3D helical path.

<Part> **<Sketch>** **<Profile>**

Command: **AMPROFILE**
Select objects for sketch:
Select objects: [**Select A (Figure 3.97).**]
Select objects: [**Enter**]

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.97).**]
Select second object or place dimension: [**Select B (Figure 3.97).**]
Specify dimension placement: [**Select C (Figure 3.97).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.97).**]
Select second object or place dimension: [**Select B (Figure 3.97).**]
Specify dimension placement: [**Select D (Figure 3.97).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.97).**]
Select second object or place dimension: [**Select E (Figure 3.97).**]
Undo/Radius/Ordinate/Placement point/Enter dimension value: **3**
Solved fully constrained sketch.
Select first object: [**Enter**]

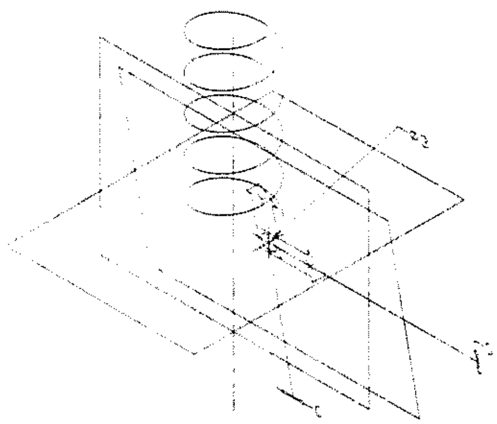


Figure 3.98 Circle resolved and constrained

Now you have a 3D helical path and a 2D profile constructed at the start point of the path. To construct the helical solid feature, set the current layer to Solid. Then use the AMSWEEP command. (See Figure 3.99.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketched Features>** **<Sweep...>**

Command: **AMSWEEP**

[Operation **Base** Termination **Path Only**
OK]

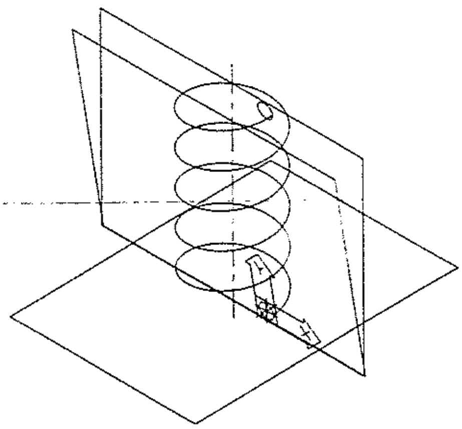


Figure 3.99 Helical sweep solid constructed

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Spring.dwg**

In making this sweep solid, you learned how to construct a work plane and construct a work axis on it, specify a 3D helical path about a work axis, construct a work plane normal to the start point of the helical path, construct a profile at the start of the helical path, and sweep the profile along the helical path.

The result of this project is quite similar to that of the helical spring project in Chapter 2. If you are dealing with helical solids, you should use this approach. If you are dealing with an object that has a free-form 3D path, you can construct a sweep surface and use the sweep surface to cut a solid.

3.7 Sketching Approach — Loft

A common characteristic of extrude, revolve, and sweep solids is that they have regular cross sections. To make a solid that has a changing cross section, you can define a series of profiles and loft them to a solid. A loft solid resembles a free-form solid.

Induction Manifold Project

Figure 3.100 shows the rendered image of the main body of an induction manifold. You will construct a loft solid here and complete the project later in this chapter. This solid is constructed by lofting along three cross sections, each of which is defined by a profile. To construct this solid model, you have to construct three profiles. (See Figure 3.101.) To construct these profiles, you need to construct a series of work planes.

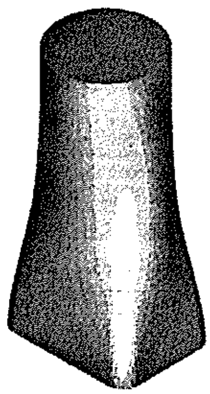


Figure 3.100 Rendered image of the outer body of an induction manifold

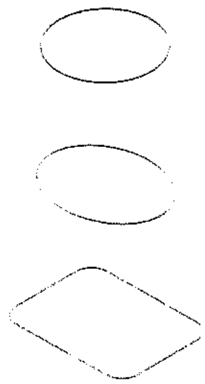


Figure 3.101 Profiles of the loft solid

Start a new part file and use Solid.dwt as the template.

<File> **<New Part File>**

Template File: **Solid.dwt**

Use the AMWOKRPLN command to construct three work planes on the World YZ plane, World XZ plane, and World XY plane. (See Figure 3.102.)

<Part> <Work Features> **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **World YZ**
OK]

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **World XZ**
OK]

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **World XY**
Create Sketch Plane
OK]

Z-flip/Rotate/<Select edge to align X axis>: [**Accept if the XY plane is the same as those shown in Figure 3.102.**]

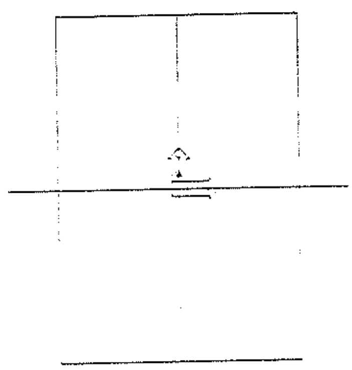


Figure 3.102 Three work planes constructed

Now you have three work planes that are all perpendicular to each other. You will use one work plane for making the first sketch and use the other two as references. On the World XY work plane, construct a rectangle with rounded corners. (See Figure 3.103.)

<Design> **<Rectangle>**

Command: **RECTANG**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **F**
Fillet radius for rectangles: **5**
Chamfer/Elevation/Fillet/Thickness/Width/<First corner>: **-20,-15**
Other corner: **@40,30**

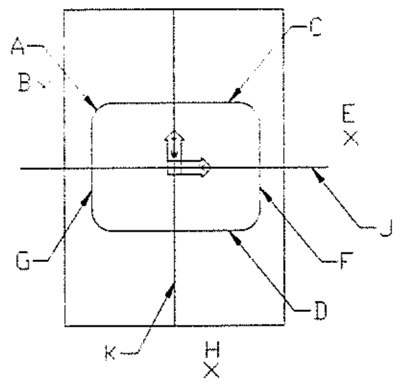


Figure 3.103 Sketch of the first profile

Resolve the sketch to a profile. Then add parametric dimensions. (See Figure 3.104.)

<Part> **<Sketch>** **<Single Profile>**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.103).**]
Select second object or place dimension: [**Select B (Figure 3.103).**]
Undo/Diameter/Ordinate/Placement point/Enter dimension value: **5**

Select first object: [**Select C (Figure 3.103).**]
Select second object or place dimension: [**Select D (Figure 3.103).**]
Specify dimension placement: [**Select E (Figure 3.103).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **30**

Select first object: [**Select F (Figure 3.103).**]
Select second object or place dimension: [**Select G (Figure 3.103).**]
Specify dimension placement: [**Select H (Figure 3.103).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **40**

The following two dimensions are used to set the location of the profile relative to the two perpendicular work planes:

Select first object: [**Select C (Figure 3.103).**]
Select second object or place dimension: [**Select J (Figure 3.103).**]
Specify dimension placement: [**Select E (Figure 3.103).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **V**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **15**

Select first object: [**Select F (Figure 3.103).**]
Select second object or place dimension: [**Select K (Figure 3.103).**]
Specify dimension placement: [**Select H (Figure 3.103).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **H**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **20**

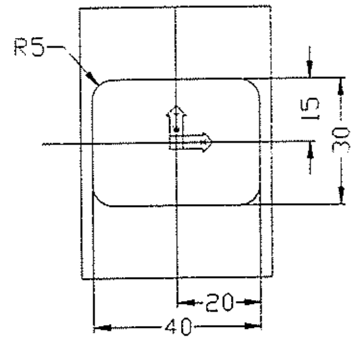


Figure 3.104 First profile constrained

Set the display to an isometric view by using the shortcut key [8].

Command: **8**

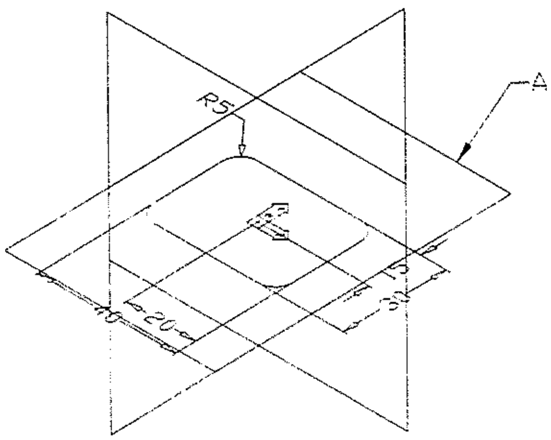


Figure 3.105 Display set to an isometric view

The second profile is offset 50 units from the first profile. Because the second profile also needs a sketch plane, use the AMWORKPLN command to construct an offset work plane and set it as the sketch plane. (See Figure 3.106.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset** Offset **40**
Create Sketch Plane
OK]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: [**Select A (Figure 3.105).**]
Flip/<Accept>: [**Accept if the arrow is pointing upward.**]

Z-flip/Rotate/<Select edge to align X axis>: **[Enter to accept if the UCS orientation is the same as that shown in Figure 3.106.]**

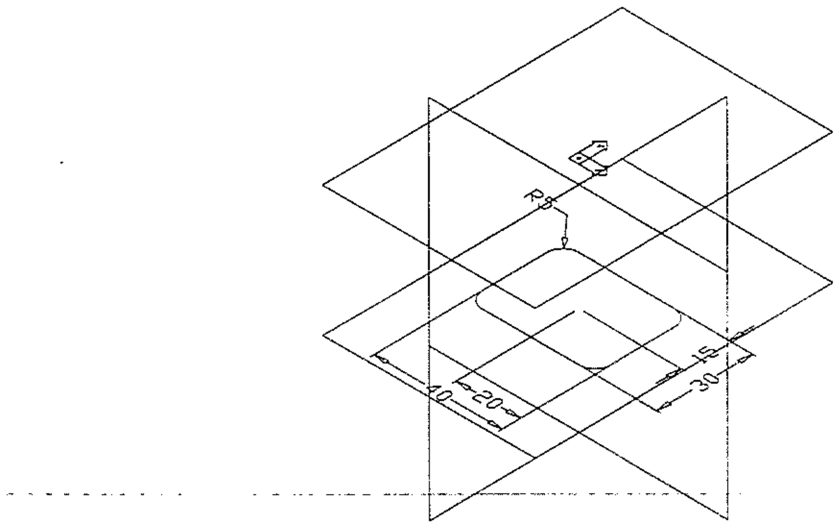


Figure 3.106 Offset work plane constructed

Set the display to the top view by using the shortcut key [9]. Then construct an ellipse as shown in Figure 3.107. After that, resolve it.

Command: **9**

<Design>	**<Ellipse>**	**<Axis, End>**
<Part>	**<Sketch>**	**<Profile>**

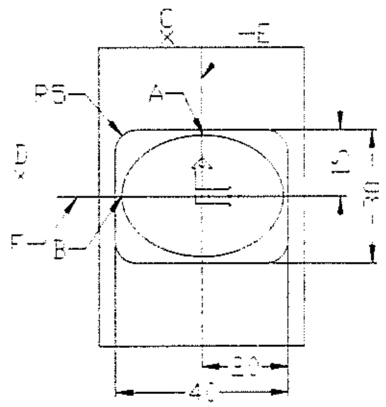


Figure 3.107 Sketch constructed on offset work plane and resolved

Use the AMPARDIM command to add parametric dimensions to constrain the size of the profile and to establish relationships with the two perpendicular work planes.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **QUA** of **[Select A (Figure 3.107).]**
Select second object or place dimension: **QUA** of **[Select B (Figure 3.107).]**
Specify dimension placement: **[Select D (Figure 3.107).]**
Undo/Ordinate/Placement point/Enter dimension value: **14**

Select first object: **QUA** of **[Select B (Figure 3.107).]**
Select second object or place dimension: **QUA** of **[Select A (Figure 3.107).]**
Specify dimension placement: **[Select C (Figure 3.107).]**
Undo/Ordinate/Placement point/Enter dimension value: **18**

The following two dimensions relate the profile to the two vertical work planes:

Select first object: **[Select A (Figure 3.107).]**
Select second object or place dimension: **[Select E (Figure 3.107).]**
Specify dimension placement: **[Select C (Figure 3.107).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: **[Select B (Figure 3.107).]**
Select second object or place dimension: **[Select F (Figure 3.107).]**
Specify dimension placement: **[Select D (Figure 3.107).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Now set the display to an isometric view. (See Figure 3.108.)

Command: **8**

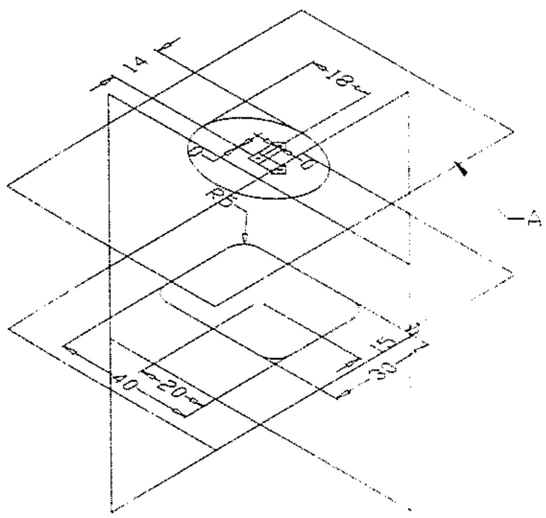


Figure 3.108 Second resolved sketch constrained, display set to an isometric view

Construct another offset work plane and set the sketch plane to the new work plane. (See Figure 3.109.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset**
Offset **40**
Create Sketch Plane
OK]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.108).]**
Flip/<Accept>: **[Accept if the arrow is pointing upward.]**
Z-flip/Rotate/<Select edge to align X axis>: **[Enter to accept if the UCS orientation is the same as that shown in Figure 3.109.]**

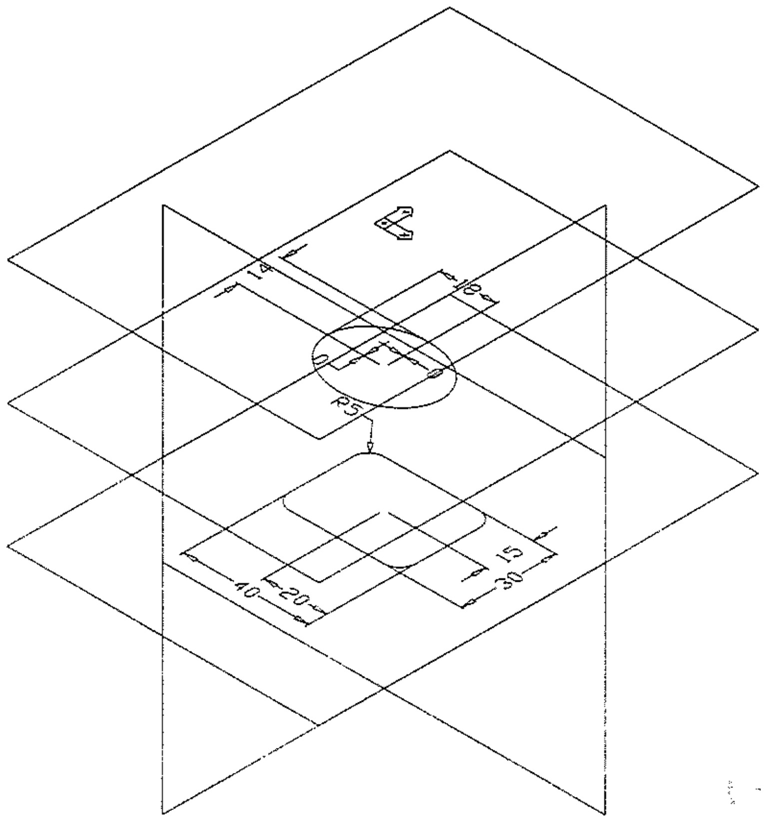


Figure 3.109 Offset work plane constructed

Set the display to the top view by using the shortcut key [9]. Then construct a circle on the new work plane and resolve the sketch. (See Figure 3.110.)

Command: **9**

<Design> **<Circle>** **<Center, Radius>**

<Part> **<Sketch>** **<Profile>**

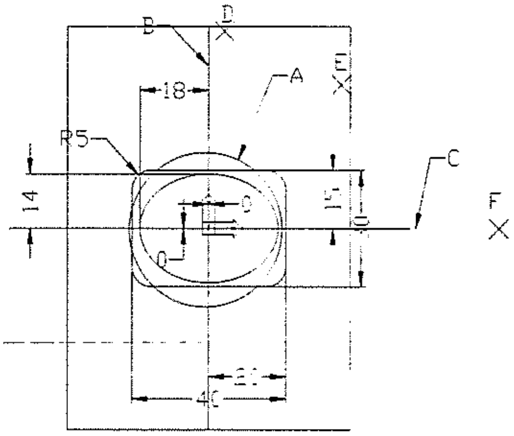


Figure 3.110 Third sketch constructed and resolved

Add parametric dimensions to constrain the shape and size of the profile and to set the relationships with the two vertical work planes. (See Figure 3.111.)

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.110).**]
Select second object or place dimension: [**Select E (Figure 3.110).**]
Undo/Radius/Ordinate/Placement point/Enter dimension value: **30**

Select first object: [**Select A (Figure 3.110).**]
Select second object or place dimension: [**Select B (Figure 3.110).**]
Specify dimension placement: [**Select D (Figure 3.110).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.110).**]
Select second object or place dimension: [**Select C (Figure 3.110).**]
Specify dimension placement: [**Select F (Figure 3.110).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

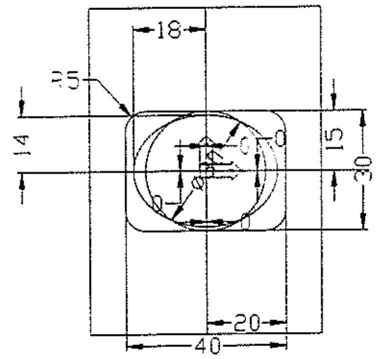


Figure 3.111 Third profile completed

Three profiles that represent the three cross sections of the loft solid are complete. Now use the AMVISIBLE command to hide the work planes. Then set the display to an isometric view. (See Figure 3.112.)

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[Hide · Work Planes OK]

Command: **8**

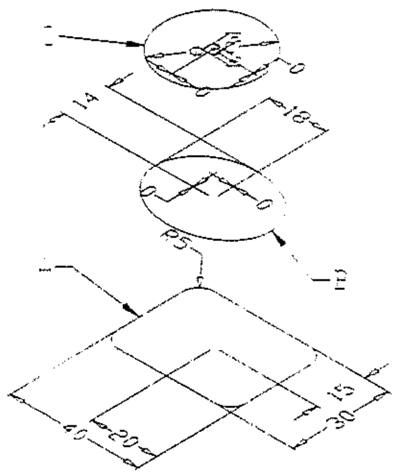


Figure 3.112 Work planes hidden, display set to an isometric view

Set the current layer to Solid. Then use the AMLOFT command to construct a loft solid. (See Figure 3.113, the Loft dialog box.) Set the Angle and Weight of the Start Section and End Section to 90 and 2, respectively. Then select the [Define<] button. After that, select the profiles one by one. On returning to the Loft dialog box, select the [OK] button. (See Figure 3.114.)

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketch>** **<Loft...>**

Command: **AMLOFT**

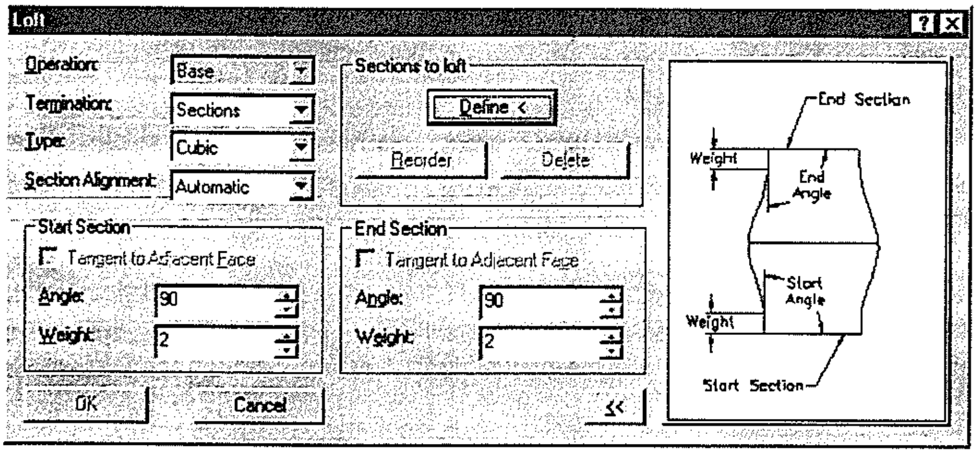


Figure 3.113 Loft dialog box

Select profiles or planar faces to loft: **[Select A (Figure 3.112).]**
Select profiles or planar faces to loft: **[Select B (Figure 3.112).]**
Redefine/<Select profiles or planar faces to loft>: **[Select C (Figure 3.112).]**
Redefine/<Select profiles or planar faces to loft>: **[Enter]**

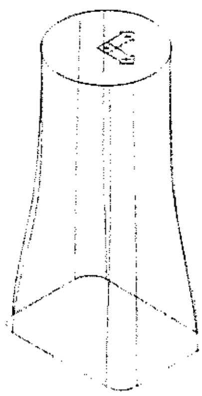


Figure 3.114 Loft solid constructed

The model is complete. Save your drawing.

<File> <Save>

File name: **Manifold.dwg**

In making this loft solid, you learned how to construct a number of work planes on which you constructed a number of sketches and to construct a loft solid from a series of profiles. Together with extrude, revolve, and sweep solids, loft solids provide a way to construct a large repertoire of 3D objects.

3.8 Sketching Approach — Split Face

It is common engineering practice to add face drafts to side faces of objects that are to be manufactured by molding or forging. Adding face drafts facilitates removal of the object from the mold.

Forged Link Project

Figure 3.115 shows the rendered image of a link bar. This object is manufactured by forging a piece of hot metal between a pair of forging dies. To facilitate removal of the object from the forging die, the side faces are drafted at an angle. To make this solid model, you will construct a sketch, extrude the sketch, split the side faces into two parts, and apply face drafts to the side faces. You will work on this solid model in two stages. First you will make the model and split the side face. Then, later in this chapter, you will apply face drafts along the split line.

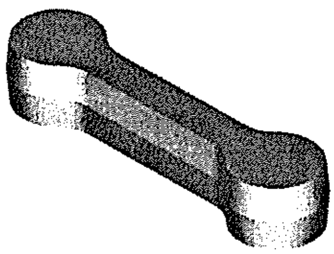


Figure 3.115 Rendered image of a link bar

Start a new multi-part drawing and use Solid.dwt as the template.

<File> <New... >

Template File: **Solid.dwt**

Set the drawing limits to 120 units by 100 units. Then zoom the display to these limits. After that, construct a sketch like the one in Figure 3.116.

<Design> <Polyline>

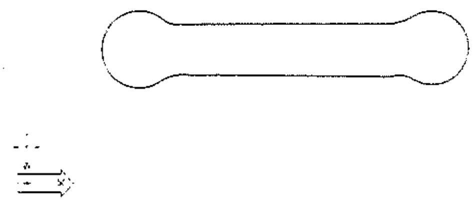


Figure 3.116 Sketch for the link bar

Use the AMPROFILE command to resolve the sketch to a profile. (See Figure 3.117.)

<Part> **<Sketch>** **<Single Profile>**

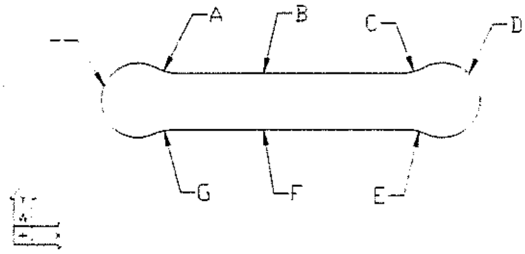


Figure 3.117 Resolved sketch

After resolving the sketch, use the AMSHOWCON command to display the geometric constraints applied. Then remove inappropriate geometric constraints by using the AMDELCON command and add appropriate geometric constraints by using the AMADDCON command in accordance with the following table.

Lines B and F (Figure 3.117)	horizontal
Arcs A, C, E, and G (Figure 3.117)	equal radius
Arcs H and D (Figure 3.117)	equal radius
Arcs H and D (Figure 3.117)	same Y value
Lines B and F (Figure 3.117)	equal length
Arc A and line B (Figure 3.117)	tangent
Line B and arc C (Figure 3.117)	tangent
Arcs C and D (Figure 3.117)	tangent
Arcs D and E (Figure 3.117)	tangent
Arc E and line F (Figure 3.117)	tangent
Line F and arc G (Figure 3.117)	tangent
Arcs G and H (Figure 3.117)	tangent
Arcs H and A (Figure 3.117)	tangent

<Part> **<Sketch>** **<Add Constraints>** **<... >**

After adding geometric constraints, use the AMPARDIM command to add parametric dimensions as shown in Figure 3.118.

<Part> <Add Dimension>

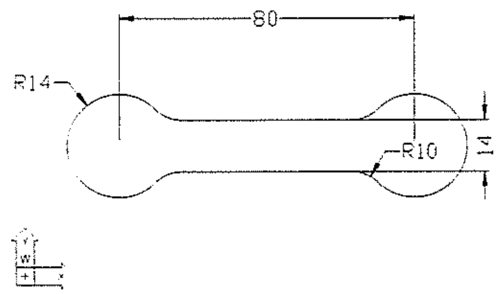


Figure 3.118 Parametric dimensions added

Set the display to an isometric view by using the shortcut key [8]. Then use the AMEXTRUDE command to extrude the profile. The extrusion height is 20 units and its termination is mid-plane. (See Figure 3.119.)

Command: **8**

<Part> <Sketched Features> <Extrude...>

Command: **AMEXTRUDE**

[Operation **Base** Termination **Mid Plane**
Size Distance: **20** Draft Angle: **0**
OK]

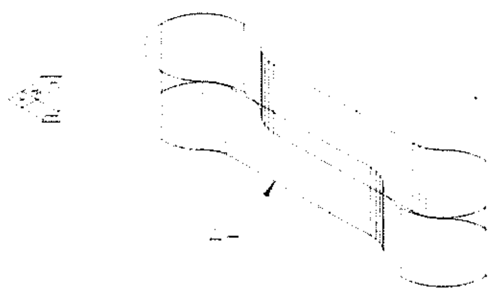


Figure 3.119 Extrude solid

While extruding from midplane, you can add a draft angle to the extrude solid. However, there are two limitations: The drafts have to start from midplane and the draft angles in two directions have to be equal. Now the profile is extruded. You will construct a split line to split one face into two.

Set the sketch plane to the side face A (Figure 3.119). (See Figure 3.120.)

<Part> **<Sketch>** **<New Sketch Plane>**

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: [**Select A (Figure 3.119).**]
Next/<Accept>: [**Accept if the vertical face A (Figure 3.119) is highlighted.**]
Z-flip/Rotate/<Select edge to align X axis>: [**Press the left button of your mouse to toggle the UCS icon until it resembles that in Figure 3.120. Then press Enter.**]

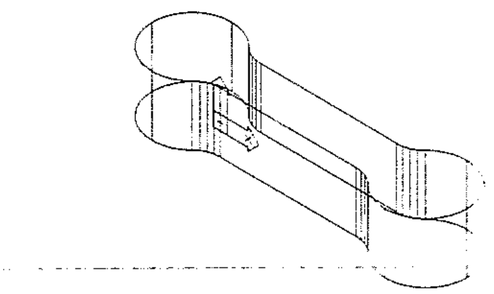


Figure 3.120 Sketch plane set to one side of the extrude solid

Set the display to the front view by using the shortcut key [9]. Then construct a horizontal line. (See Figure 3.121.) Although the exact length of the line is unimportant, it has to go beyond both ends of the solid.

Command: **9**

<Design> **<Line>**

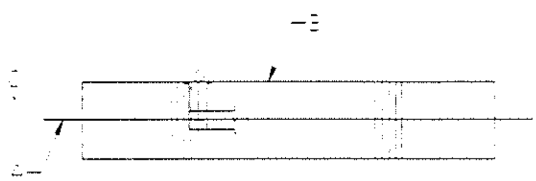


Figure 3.121 Display set, line constructed

Use the AMSPLITLINE command to resolve the horizontal line to a split line. Then add a parametric dimension. After that, set the display to an isometric view. (See Figure 3.122.)

<Part> **<Sketch>** **<Split Line>**

Command: **AMSPLITLINE**
Select objects for sketch:
Select objects: [**Select A (Figure 3.121).**]
Select objects: [**Enter**]
Select edge to include in splitline or press Return to accept: [**Enter**]

<Part> <Add Dimension>

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.121).**]
Select second object or place dimension: [**Select B (Figure 3.121).**]
Specify dimension placement: [**Select C (Figure 3.121).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **V**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **10**
Solved underconstrained sketch requiring 2 dimensions or constraints.

Command: **8**

Note that the prompt indicates that two more dimensions/constraints must be added to the split line. Because the length is unimportant, you do not have to add these dimensions.

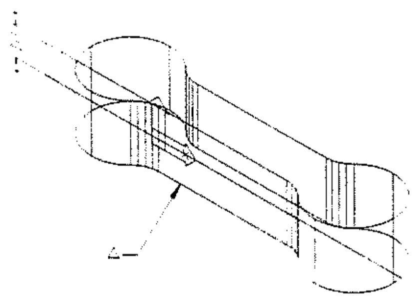


Figure 3.122 Line resolved to become a split line and constrained

Now use the AMFACESPLIT command to split the side faces. (See Figure 3.123.)

<Part> <Sketched Features> <Face Split >

Command: **AMFACESPLIT**
Planar/<pRoject>: **R**
All/<Select faces to split>: [**Select A (Figure 3.122).**]
Next/<Accept>: [**Accept if all the vertical faces are highlighted.**]
All/Remove/<Select more faces to split>: [**Enter**]

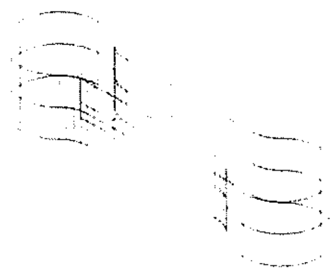


Figure 3.123 Side faces split

Now the side faces are split. Save the drawing.

<File> **<Save>**

File name: **Forge.dwg**

In making this solid model, you learned how to resolve a sketch into a split line and use the split line to split the faces of a solid. Later in this chapter, you will work on placing features and applying face drafts along the split line.

Summary

There are four kinds of sketches. You can resolve a rough sketch into a profile, a 2D path, a cut line, or a split line. In addition, you can specify a 3D helical path.

To construct a solid, you can extrude a profile in a direction perpendicular to the plane of the profile, revolve the profile about an axis, sweep the profile along a 2D path, and sweep the profile along a 3D helical path. A split line is used to split a face of a solid part into two faces along which you can apply face drafts in two directions. A cut line is used for defining an offset section plane in associative drafting.

A solid model has three kinds of features: sketched features, placed features, and work features.

A sketched feature is a solid feature that you construct from a sketch. To make a solid part, you begin by making a sketched solid feature. The first solid feature you construct for a solid part is called the base solid feature. It can be an extrude solid, a revolve solid, a sweep solid, or a 3D helical sweep solid.

While working on the preceding projects in this chapter, you learned various sketching approaches, including the use of construction lines and the use of work planes and a work axis. After learning how to use the Desktop Browser in the next section, you will learn the building-block approach in modeling and learn how to use the three kinds of features in modeling. In Chapter 5, you will work on using a cut line to construct an offset section.

3.9 The Desktop Browser

Desktop browsers are graphic interfaces that allow command shortcuts and provide controls on the construction of parts, features, assemblies, and drawings. By selecting different tabs, you can switch among different working modes. Applicable commands are displayed and inappropriate commands are greyed out. In addition, desktop browsers activate appropriate toolbars.

There are two kinds of desktop browsers: the single-part drawing browser and the multi-part drawing browser. In a single-part drawing, you can construct only one solid part. Therefore, the browser has only two tabs: Part and Drawing. (See Figure 3.124.) The Part tab provides control for parts and features construction, and the Drawing tab provides control for associative drafting creation.

In a multi-part drawing, you can construct and manipulate multiple solid parts. Therefore, the browser has three tabs: Assembly, Scene, and Drawing. (See Figure

3.125.) Here the Assembly tab serves two purposes. First, it provides control for construction of parts and features. Second, it provides control for assembling two or more solid parts. The Scene tab is for setting up scenes in an assembly. Finally, the Drawing tab of the multi-part drawing is similar to that of a single-part drawing; it provides control for the creation of drawings.

Here we will give you an overview of the browsers, and outline what you can do with the Desktop Browser in solid modeling. The Assembly tab and the Scene tab will be discussed in Chapter 4. The Drawing tab will be discussed in Chapter 5.

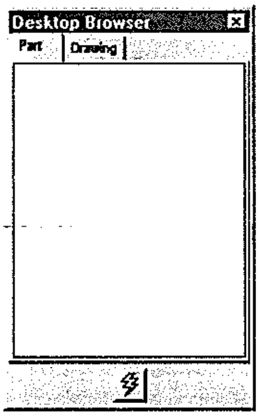


Figure 3.124 Desktop Browser: single-part drawing

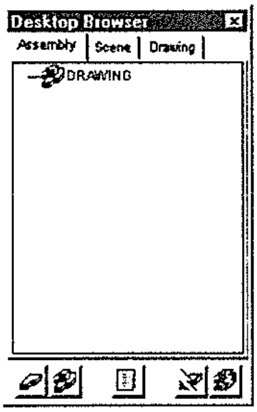


Figure 3.125 Desktop Browser: multi-part drawing

The Part tab of the single-part drawing (Figure 3.124) has one button at the bottom. It is the [Update Part] button. You can use this button to activate the AMUPDATE command. The Assembly tab of the multi-part drawing (Figure 3.125) has five buttons at the bottom:

1. The [Part Filter ON/OFF] toggle button controls the display of parts and features in the browser.
2. The [Assembly Filter ON/OFF] toggle button controls the display of assembly and constraints.

3. The [Catalog] button activates the AMCATALOG command.
4. The [Update Part] button activates the AMUPDATE command.
5. The [Update Assembly] button activates the AMASSEMBLE command.

In a new single-part drawing, move the cursor onto the background of the browser and press the right mouse button. A pop-up menu appears. (See Figure 3.126.) It has eight menu items: New, New Sketch Plane, Solve, Placed Features, Work Features, Feature Suppression, Design Variables, and Browser Filter. Because this is a new drawing and some menu items are not applicable, they are greyed out.

When you select the New item, a cascading menu displays. Here you can activate a new solid part or convert a native solid to a static base solid feature.

When you select the Solve item, the Solve cascading menu displays. It provides shortcuts for resolving sketches and specifying helical paths. (See Figure 3.127.)

When you select the Work Features item, the Work Features cascading menu displays. It provides shortcuts for construction of work features. (See Figure 3.128.)

When you select the Browser Filter item, a Desktop Browser Filter dialog box appears. (See Figure 3.129.)

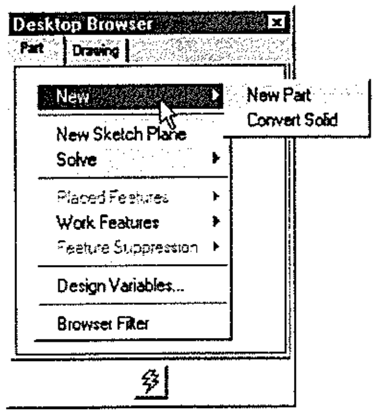


Figure 3.126 Pop-up menu for a new single-part drawing

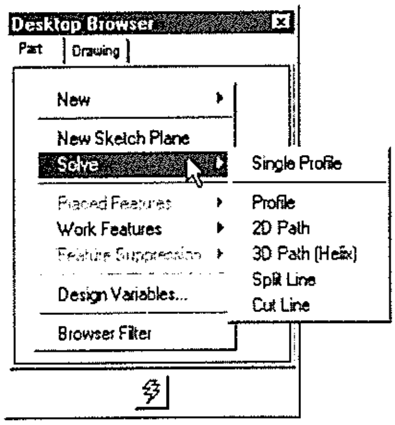


Figure 3.127 Solve cascading menu

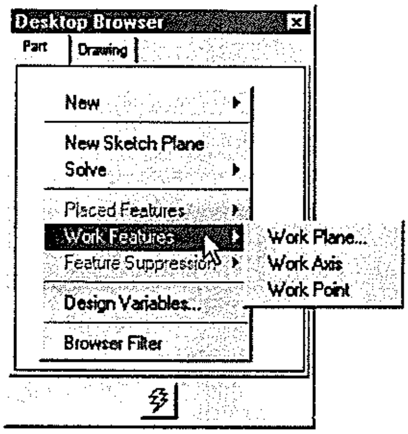


Figure 3.128 Work Features cascading menu

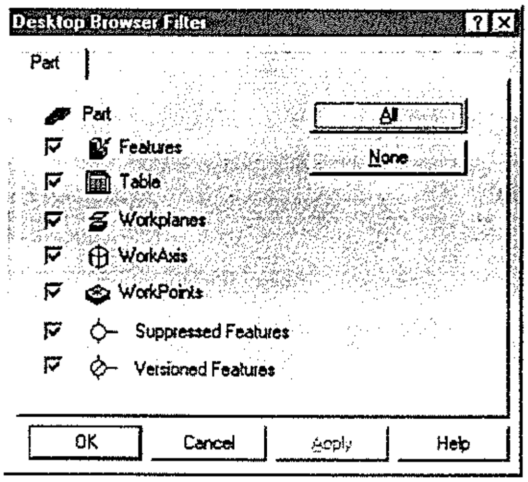


Figure 3.129 Desktop Browser Filter dialog box

As shown in Figure 3.129, this dialog box has a number of check boxes. They control the kinds of objects to be displayed in the browser. For example, unchecking the Workplanes box suppresses the display of work planes in the browser. Now check all the boxes and then select the [OK] button to exit.

Now open the drawing Manifold.dwg. In the browser, you will see an object called Manifold, which is the file name of this part drawing. If you have saved the drawing under a different name, the name of the object in the browser will be your saved file name. Select the object and press the right mouse button. A pop-up menu is displayed. (See Figure 3.130.) There are five items, of which the List Attributes... item is greyed out. The BOM Attributes... item concerns attributes of this solid part in a bill of materials in an assembly. The Design Variables item concerns the use of design variables. The Delete item deletes the solid part. The Feature Suppression item, of course, concerns suppression of features.

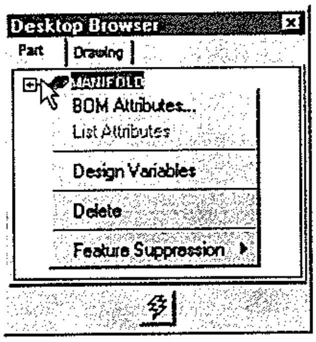


Figure 3.130 Desktop Browser, showing a solid part

Now select the background of the browser. Then press the right mouse button. After that, select the Placed Features item. A Placed Features cascading menu is displayed. It provides shortcuts for constructing placed features. (See Figure 3.131.)

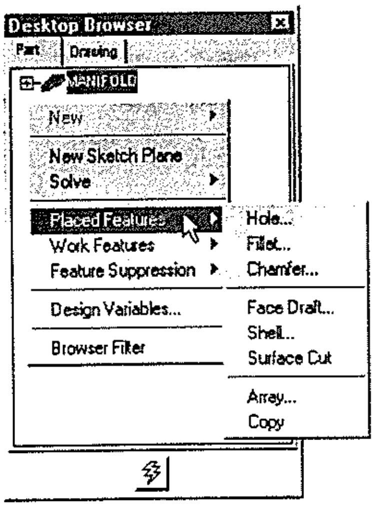


Figure 3.131 Placed Features cascading menu

When you select the Feature Suppression item, a cascading menu is displayed. (See Figure 3.132.) This menu enables you to suppress selected objects.

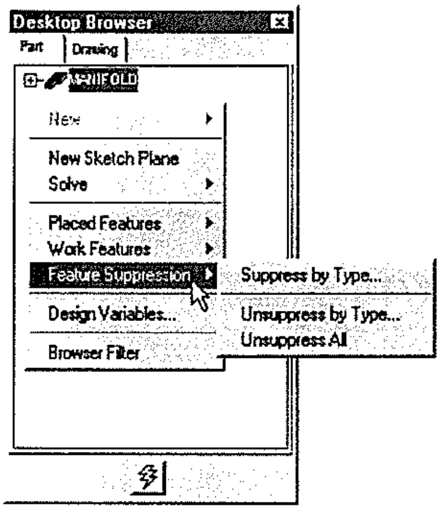


Figure 3.132 Feature Suppression cascading menu

Display the graphic tree. Then select a work feature in the graphic tree and press the right mouse button. (See Figure 3.133.)

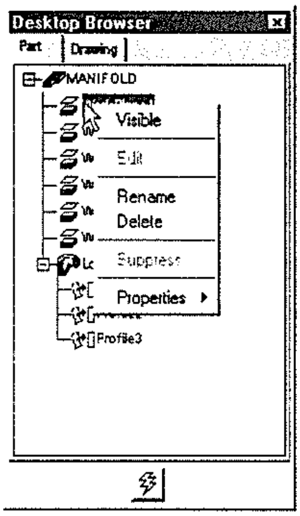


Figure 3.133 Selecting a work feature and pressing the right mouse button

Now open the drawing Forge.dwg. This is a multi-part drawing. Place the cursor on the background and press the right mouse button. (See Figure 3.134.)

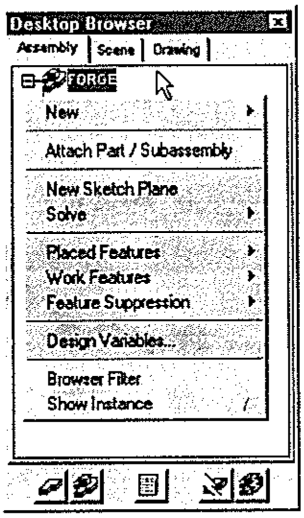


Figure 3.134 Pop-up menu when the background of the browser is selected for a multi-part drawing

With the Desktop Browser in action, some working space on your screen is occupied. To maximize the working space, you can either close or collapse the browser. To close the browser, you can select the [x] sign at the upper right corner of the browser. To collapse the browser, you can drag the browser to the central part of your screen, place the cursor on the frame of the browser, and double-click. (See Figure 3.135.) When the browser is collapsed, double-clicking while the cursor is on the browser frame expands the browser.

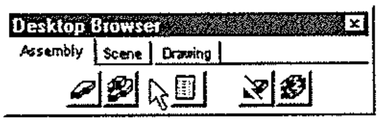


Figure 3.135 Browser collapsed

To sum up, the Desktop Browser is a graphic interface that provides command shortcuts. To bring up appropriate pop-up menus, you can press the right mouse button.

3.10 Building-Block Approach

A solid has three kinds of features: sketched features, placed features, and work features. Sketched and placed features are solid features that you use to compose a solid model. The distinction between a sketched solid feature and a placed solid feature is that a sketched solid feature starts from a rough sketch, whereas a placed solid feature is added to the solid part directly. Work features are construction aids with which you combine solid features parametrically. Given a solid model to construct, you should take some time to analyze its geometric components. This analysis process will help you to break down the model into a number of simple solid features, which can be either sketched solid features or placed solid features.

Base Solid Feature

From among the sketched solid features, you select the first feature to construct — the base solid feature. Sometimes deciding which sketched solid feature will be the base solid feature is critical because you will add other sketched solid features to it by cutting, joining, or intersecting. Choosing the inappropriate sketched solid feature as the base solid feature may cost you extra effort in composing the remaining sketched solid features or may keep you from adding other solid features to the solid part.

There is no rule as to which solid feature you should choose as the base solid feature. Experience plays a vital part. By working on the projects and exercises in this chapter, you can gain some insight into how this choice should be made.

Sketch and Sketch Plane

After selecting a base solid feature, you construct a sketch. When you construct a sketch, you put it on a sketch plane. In Mechanical Desktop terms, a sketch plane is equivalent to the XY construction plane of AutoCAD on which you make your 2D sketches. The default sketch plane of a new drawing is the XY plane of the world coordinate system. If you want to put the sketch on other planes, you can make a work plane and use it as the sketch plane. Then you construct the first sketch and subsequently make the base solid feature.

The sketch of the second sketched feature of a solid model, like the sketch of the base solid feature, has to be placed on a sketch plane. To maintain parametric relationships among the solid features, the sketch plane needs to be placed on an existing plane of the solid model. If such a plane is not available on the existing solid, you must construct a parametric work plane so you can place the sketch plane.

Work Features

To reiterate, you need to select a plane on the solid model or construct a work plane as the sketch plane for subsequent sketches of the solid model. When constructing the work plane, you can make use of other work planes and work features. In all, there are three kinds of work features: work planes, work points, and work axes.

Placed Features

As their name implies, placed features are placed on existing solid features and sketching is not required. To maintain parametric relationships among them and the sketched solid features, you may need to use work features as well.

Projects

Now you will use the building-block approach to work on a number of projects. In these projects, you will compose a solid model by constructing sketched solid features and placed solid features. In addition, you will construct work features to locate them and to maintain parametric relationships among them.

Mounting Block Project (Continued)

Figure 3.136 shows two rendered images of the model of the completed mounting block.

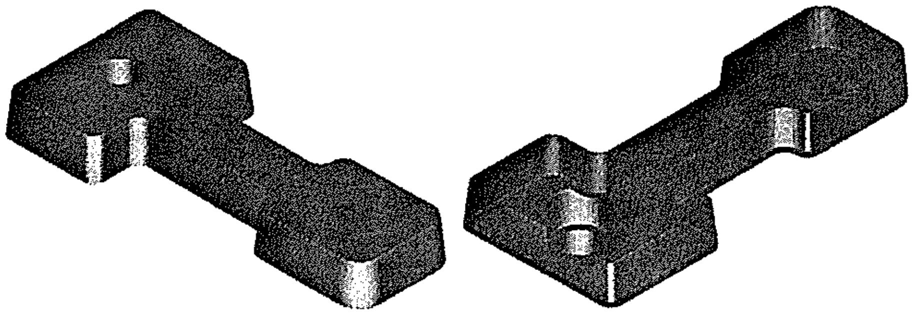


Figure 3.136 Rendered images of the mounting block

Open the drawing that you saved in Section 3.4. (See Figure 3.64.)

<File> **<Open...>**

File name: **Mblock.dwg**

In Section 3.4, you constructed the base solid feature of this solid part. Now you will perform the following tasks to modify and incorporate additional features:

1. Change the height of the block.
2. Add chamfer edges.
3. Remove the chamfer edges (as if you had changed your mind).
4. Add fillet edges.
5. Cut a shell.
6. Cut a hole.
7. Reorder the cutting of the hole.

In the Desktop Browser, expand the graphic tree. Then select the Extrusion Blind item to highlight it, press the right mouse button, and select the Edit item from the pop-up menu. After that, select the d11 (Figure 3.137) dimension (note that the parametric name of this dimension may not be the same as the one in your drawing) and change its value to 10.

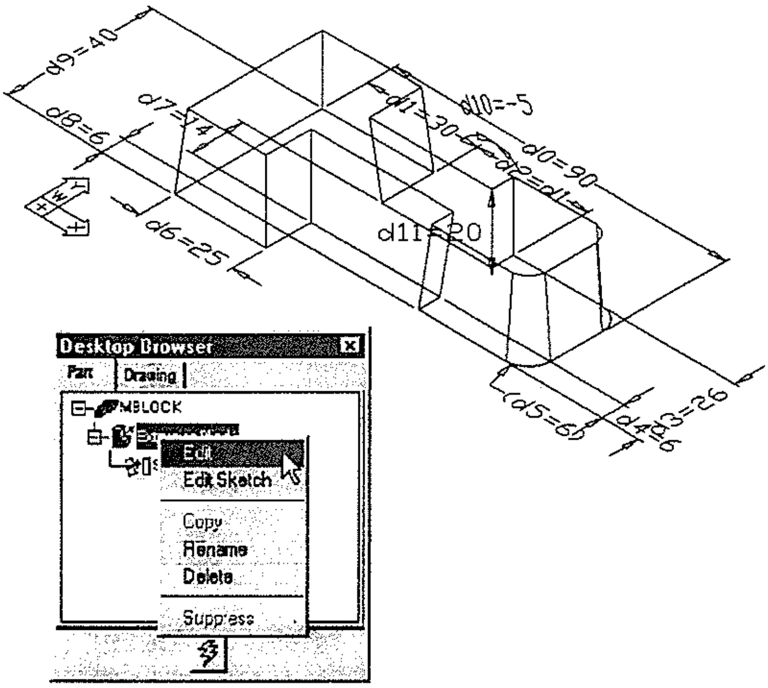


Figure 3.137 Using the Desktop Browser shortcut to edit the extrude solid

Enter new value for dimension: **10**
Select object: **[Enter]**

After editing, select the [Update Part] button in the Desktop Browser to update the change. (See Figure 3.138.)

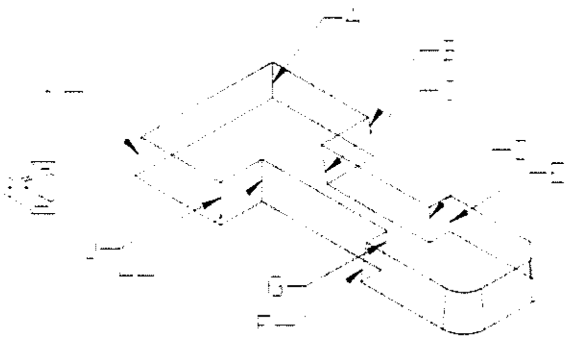


Figure 3.138 Change updated

Now add two chamfer edges. As we have said, chamfer edges are placed solid features. Use the AMCHAMFER command. To run this command, you can use the pull-down menu or the browser shortcuts (Figure 3.131). (See Figure 3.139, the Chamfer Feature dialog box.) There are three kinds of chamfer operations. Select the Equal Distance operation and set Distance to 4 units. Then select the [OK] button. After that, select the edges to chamfer.

<Part> **<Placed Features>** **<Chamfer...>**

Command: **AMCHAMFER**

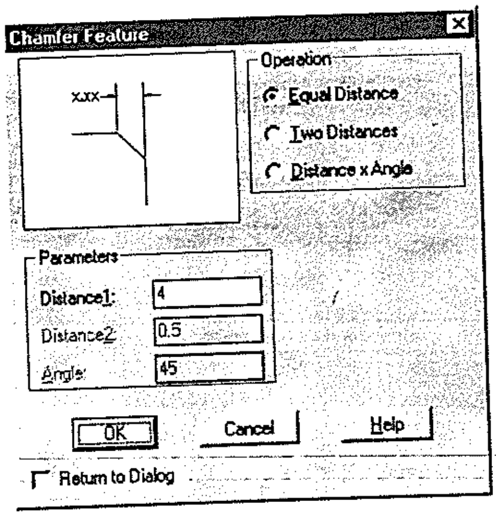


Figure 3.139 Chamfer Feature dialog box

Pick the edge to chamfer: **[Select A and K (Figure 3.138).]**
Pick the edge to chamfer: **[Enter]**

Now suppose you change your mind and want to remove the chamfer edges. Use the AMDELFEAT command by selecting Delete Feature from the Part pull-down menu or use the shortcut in the browser. (Figure 3.140 shows the pop-up menu in the browser.) Because the two chamfer edges are constructed in a single operation, deletion will remove both of them at the same time.

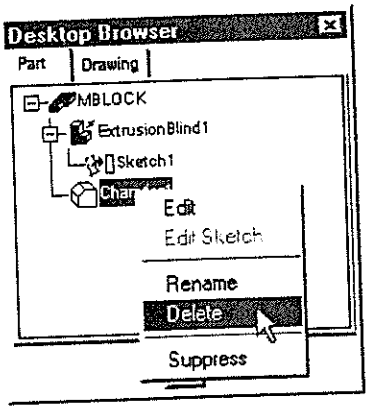


Figure 3.140 Using the shortcut to delete chamfer edges

Command: **AMDELFEAT**
Highlighted features will be deleted. Continue ? No/<Yes>: **[Enter]**

Now use the AMFILLET command to add fillet edges. (See Figure 3.141, the Fillet dialog box.) There are four kinds of fillet features. Select Constant and specify a radius of

3 units. Then select the [Apply] button. After that, select the edges to fillet. (See Figure 3.142.)

<Part> **<Placed Features>** **<Fillet...>**

Command: **AMFILLET**

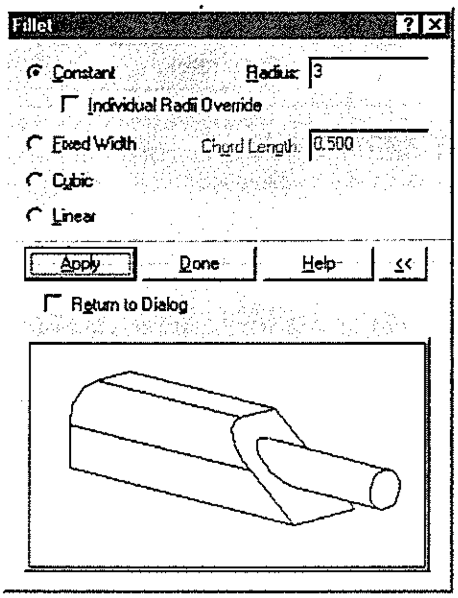


Figure 3.141 Fillet dialog box

Select edges: **[Select A, B, C, D, E, F, G, H, J, and K (Figure 3.138).]**
Select edges: **[Enter]**

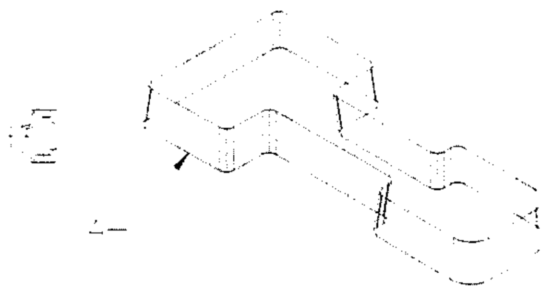


Figure 3.142 Fillet edges placed

Use the AMSHELL command to place a shell feature. While shelling, exclude the bottom face. In Figure 3.143, the Shell Feature dialog box, there are three kinds of shells. Specify an inside thickness of 2 units. Then select the [Add<] button. After that, select the bottom face of the solid model. On returning to the Shell Feature dialog box, select the [OK] button. (See Figure 3.144.)

<Part> **<Placed Features>** **<Shell...>**

Command: **AMSHELL**

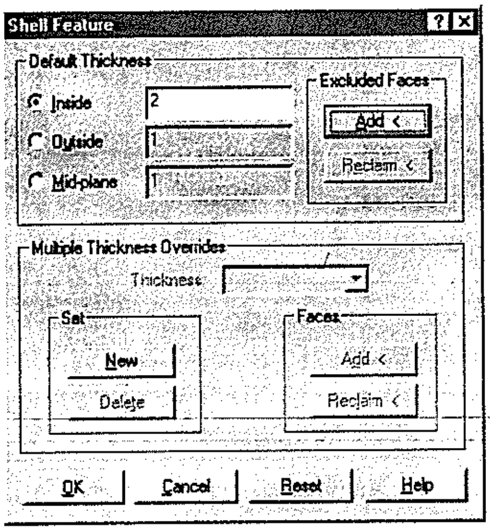


Figure 3.143 Shell Feature dialog box

Select faces to exclude: [**Select A (Figure 3.142).**]
Next/<Accept>: [**Accept if the bottom face is highlighted.**]
Select faces to exclude: [**Enter**]

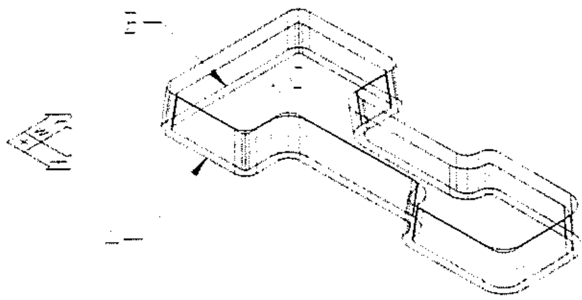


Figure 3.144 Shell feature placed

Features constructed on a base solid feature can be reordered, depending on validity after reordering. To see how reordering can be done, select the Shell1 item in the browser (Figure 3.145) and drag it to just above the Fillet1 item. (See Figure 3.146.)

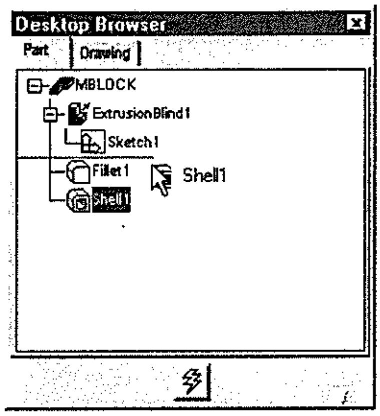


Figure 3.145 Select and drag to reorder feature

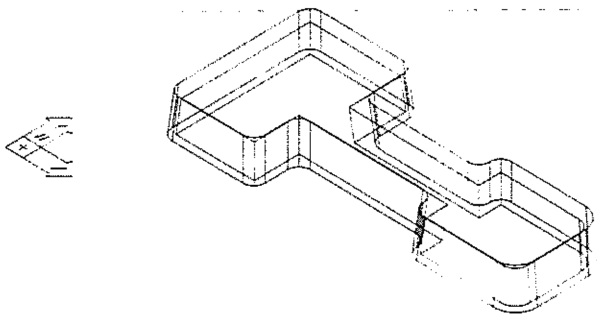


Figure 3.146 Features reordered

Now the Shell operation is placed before the Fillet operation. Because of the change of sequence of the operations, the Fillet operation fillets only the outer edges of the solid, and the internal edges of the shell are not filleted. Compare Figure 3.146 with Figure 3.144 to see the difference. Because we would like to have filleted edges inside the shell as well, select the Shell1 item of the browser and drag it back below the Fillet1 item. (See Figure 3.145.)

To continue, use the AMHOLE command to make a hole in the feature. (See Figure 3.147, the Hole Feature dialog box.) Select Drilled under Operation and Through under Termination. Placement is 2 Edges. Under Drill Size, diameter is 8. Select the [Apply<] button. After that, select two edges and a hole location. On returning to the Hole Feature dialog box, select the [Exit] button. (See Figure 3.148.)

<Part> **<Placed Features>** **<Hole...>**

Command: **AMHOLE**

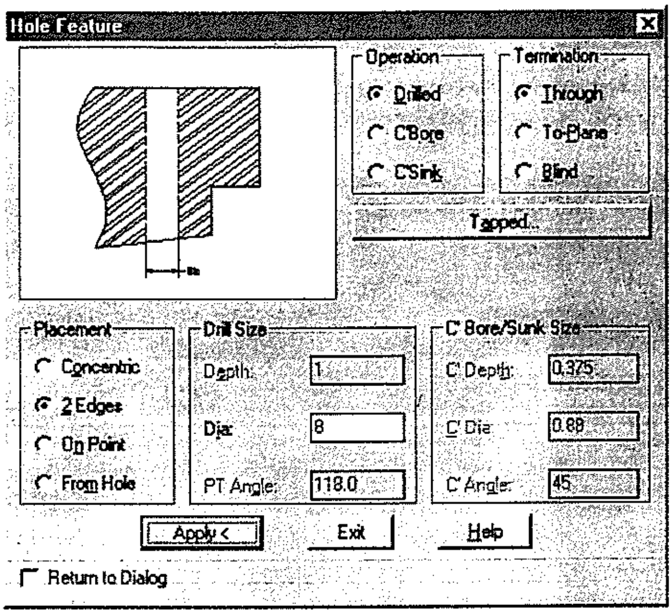


Figure 3.147 Hole Feature dialog box

Select first edge: **[Select A (Figure 3.144).]**
Select second edge: **[Select B (Figure 3.144).]**
Select hole location: **[Select C (Figure 3.144).]**
Distance from first edge: **20**
Distance from second edge: **12**

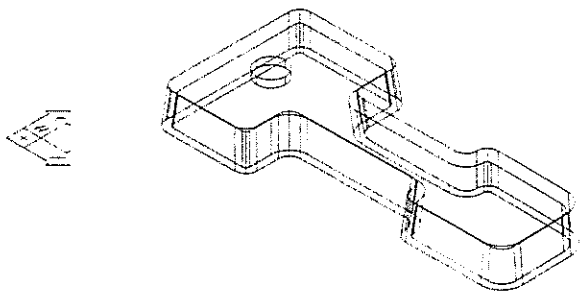


Figure 3.148 Hole placed in the solid

Compare Figure 3.148 with the rendered images in Figure 3.136. You will find that the hole should have been constructed before making the shell. To reorder the operations, you can select the Reorder Feature item of the Part pull-down menu to use the AMREORDFEAT command or use the shortcut in the browser.

<Part> **<Reorder Feature>**

Now select the Hole1 item in the graphic tree of the browser. (See Figure 3.149.) Then drag it to just above the Shell1 item to reorder the commands. The hole feature is reordered. (See Figure 3.150.)

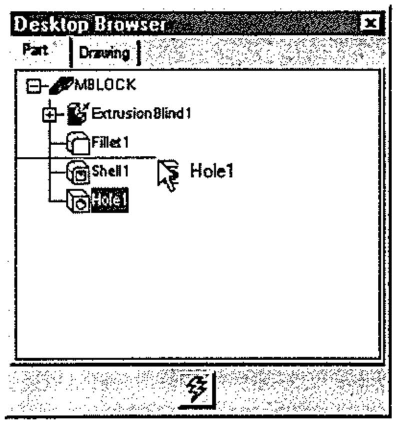


Figure 3.149 Selecting and dragging to reorder feature

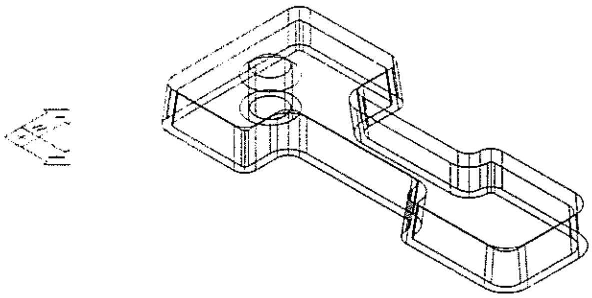


Figure 3.150 Hole feature reordered

The model is complete. Save your drawing.

<File> <Save>

While working on this model, you learned how to edit the parametric dimensions of the base solid feature, update a modified solid, place chamfer features, fillet features, and shell features, delete unwanted features, and reorder features.

Swing Arm Project (Continued)

Figure 3.151 shows the rendered image of the completed swing arm. In Section 3.6, you constructed a sweep solid. Now you will perform the following tasks:

1. Construct a revolve solid and join it to the base solid.
2. Construct a work plane.
3. Copy the sketch of the revolve solid.
4. Revolve the copied sketch to create a revolve solid and join it to the base solid.
5. Place fillet features at the intersections of the sweep solid and the revolve solids.
6. Place hole features concentric to the revolve solids.

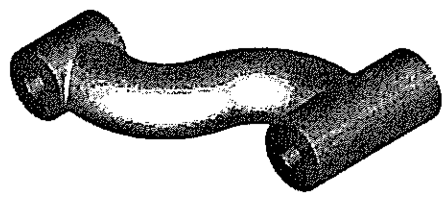


Figure 3.151 Rendered image of the swing arm

Open the drawing that you saved in Section 3.6. Use the AMVISIBLE command to unhide the work plane and set the current layer to Sketch. Then construct a rectangle on the work plane as shown in Figure 3.152. After that, resolve it to a profile.

<File> **<Open...>**

File name: **Arm_u.dwg**

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[Part
Unhide **All**
OK]

[Mechanical Main] **[Layer Control]**

Current Layer: **Sketch**

<Design> **<Rectangle>**

<Part> **<Sketch>** **<Profile>**

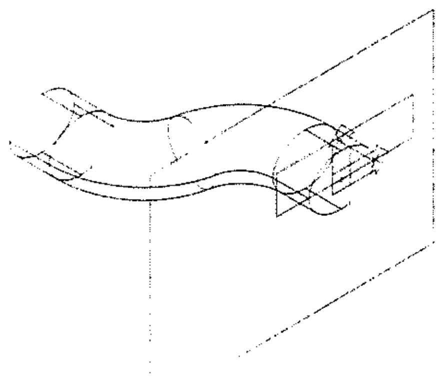


Figure 3.152 Rectangle constructed and resolved

Set the display to the plan view of the current sketch plane by using the shortcut key [9]. Then add parametric dimensions as shown in Figure 3.153.

Command: **9**

<Part> **<Add Dimension>**

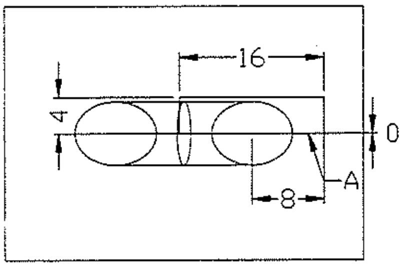


Figure 3.153 Parametric dimensions added

Set the display to an isometric view. Then revolve the profile to create a revolve solid. (See Figure 3.154.)

Command: **8**

<Part> **<Sketch Features...>** **<Revolve...>**

Command: **AMREVOLVE**

[Operation: **Join** Termination: **Full**
OK]

Select revolution axis: **[Select A (Figure 3.153).]**

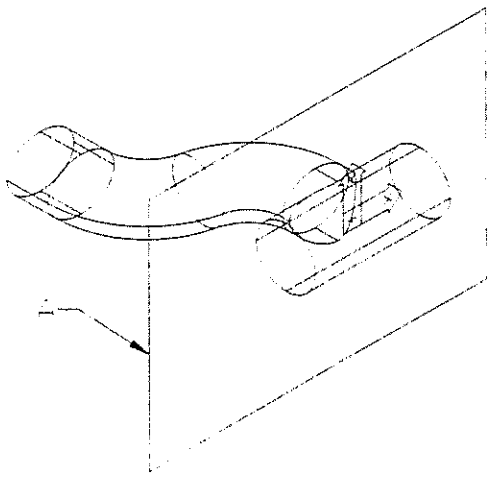


Figure 3.154 Revolve solid joined to sweep solid

Construct a work plane parallel to and offset from the current work plane. (See Figure 3.155.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset**
Offset **28**
OK]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: [**Select A (Figure 3.154).**]
Flip/<Accept>: [**Accept if the work plane is the same as that shown in Figure 3.155.**]
worldX/worldY/worldZ/<Select work axis or straight edge>: **Y**
Rotate/Z-flip/<Accept>: [**Accept if the UCS orientation is the same as that shown in Figure 3.155).**]

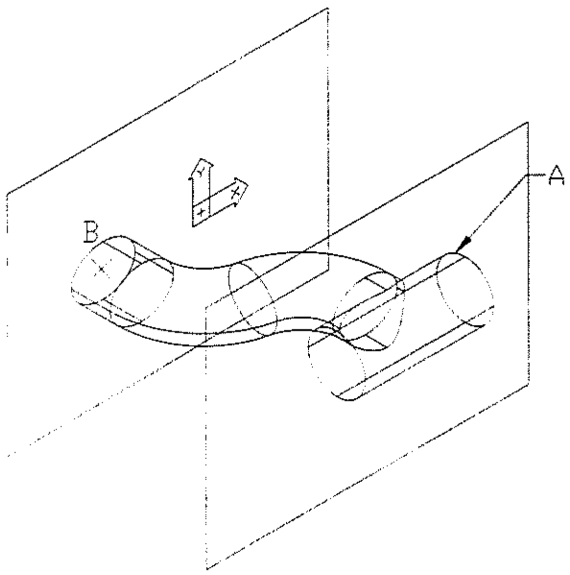


Figure 3.155 Work plane constructed

On the new sketch plane, copy the sketch of the last revolve solid. (See Figure 3.156.)

<Part> **<Sketch>** **<Copy Sketch>**

Command: **AMCOPYSKETCH**
Feature/<Sketch>: **F**
Select feature: [**Select A (Figure 3.155).**]
Sketch center: [**Select B (Figure 3.155).**]
Sketch center: [**Enter**]

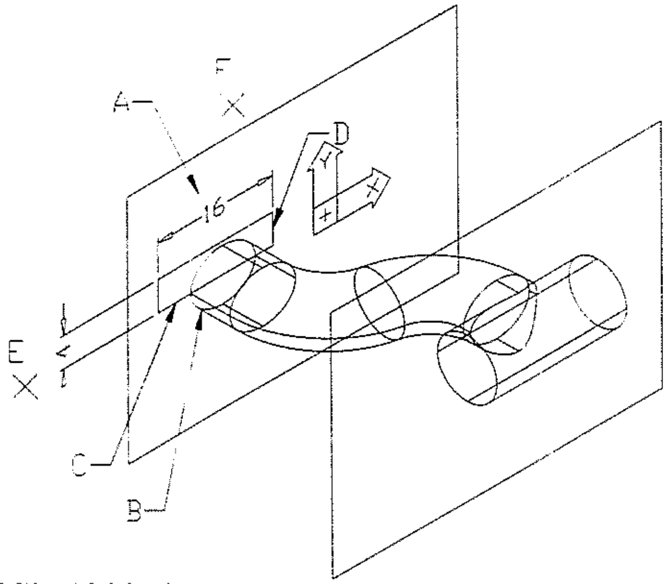


Figure 3.156 Sketch copied

As shown in Figure 3.157, modify and add parametric dimensions to constrain the profile properly.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select D (Figure 3.156).**]
Select second object or place dimension: [**Select B (Figure 3.156).**]
Specify dimension placement: [**Select F (Figure 3.156).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **5**

Select first object: [**Select C (Figure 3.156).**]
Select second object or place dimension: [**Select B (Figure 3.156).**]
Specify dimension placement: [**Select E (Figure 3.156).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 3.156).**]
New value for dimension: **10**
Select dimension to change: [**Enter**]

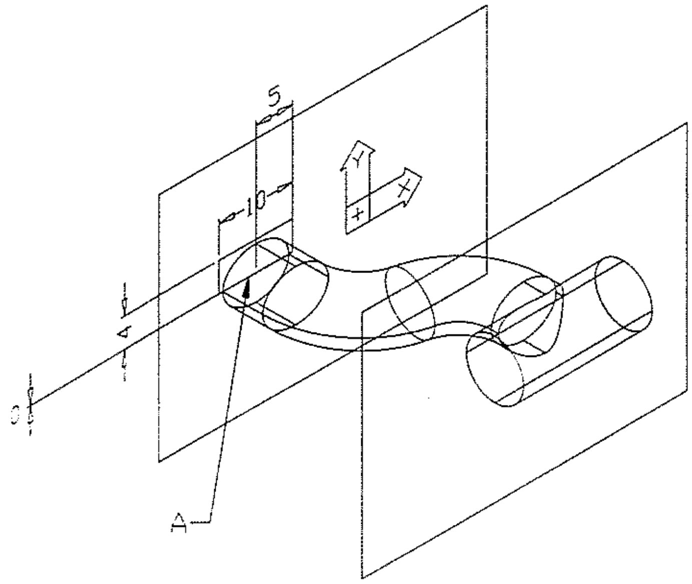


Figure 3.157 Copied sketch constrained

Revolve the profile to create a revolve solid and join it to the sweep solid. (See Figure 3.158.)

<Part> **<Sketch Features...>** **<Revolve...>**

Command: **AMREVOLVE**

[Operation: **Join** Termination: **Full**
OK]

Select revolution axis: [**Select A (Figure 3.157).**]

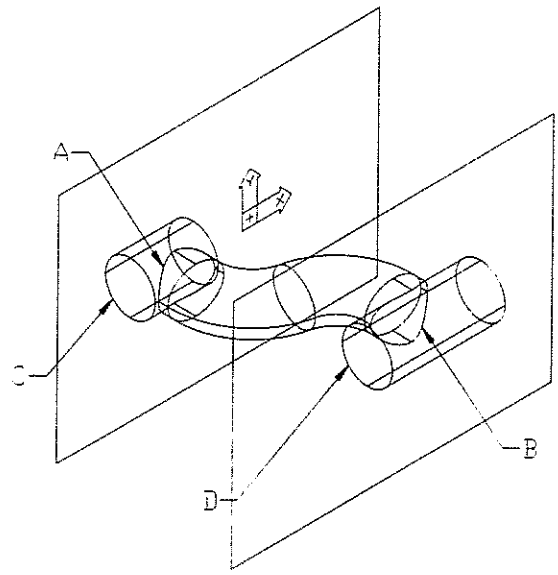


Figure 3.158 Profile revolved and joined

At the intersection of the sweep solid and the revolve solids, add two fixed-width fillet features. (The term *width* refers to the chord width of the fillet.) Then add two concentric hole features. (See Figure 3.159.)

<Part> **<Placed Features>** **<Fillet...>**

Command: **AMFILLET**

[**Fixed Width**
Chord Length: **0.5**
Apply]

Select edge: [**Select A (Figure 3.158).**]

Command: [**Enter**]
AMFILLET

[**Apply**]

Select edge: [**Select B (Figure 3.158).**]

<Part> **<Placed Features>** **<Hole...>**

Command: **AMHOLE**

[Operation **Drilled** Termination **Through**
Placement **Concentric**
Drill Size **3**
Apply<]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: [**Select C (Figure 3.158).**]

Select concentric edge: **[Select C (Figure 3.158).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select D (Figure 3.158).]**
Select concentric edge: **[Select D (Figure 3.158).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Enter]**

[Exit]

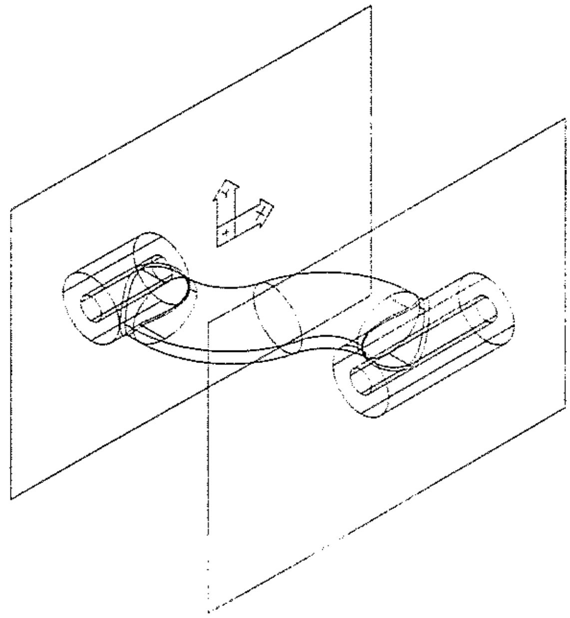


Figure 3.159 Constant fillet and concentric hole features placed

The model is complete. Use the AMVISIBLE command to hide the work planes. Then save your drawing.

<File> <Save>

While working on this project, you learned how to copy a sketch of a feature and how to place hole features and fillet features.

Machine Hand Wheel

Figure 3.160 shows the rendered image of a machine's hand wheel. Take some time to analyze the model to determine what features are needed. There are five solid features: the central boss, hexagonal hole, ribs, rim, and fillets. (See Figure 3.161.)

Obviously, the fillets are placed features and the others are sketched features. The central boss and the hexagonal hole are extrude solids. You will make profiles and extrude them. The rib is a sweep solid feature. You will make a path and a profile and sweep the profile along the path. The rim is a revolve solid feature. You will make a profile and revolve it about an axis. From the four sketched features, you have to choose one to be the base feature. In the process we will describe, the central boss is constructed first, the hole next, then the ribs, and the rim the last.

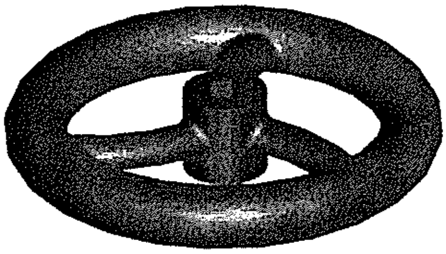


Figure 3.160 Rendered image of the hand wheel

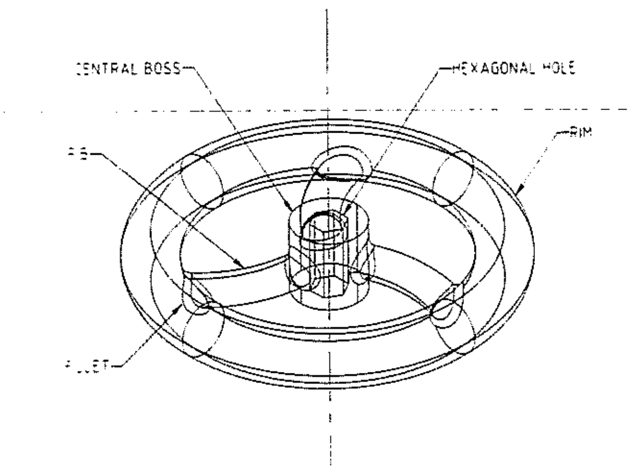


Figure 3.161 Features of the hand wheel

Start a new single-part drawing; use the template Solid.dwt.

<File> **<New Part File>**

Use the AMDIMDSP command to display the parametric dimensions in the form of an equation that consists of a parameter name and an expression.

<Part> **<Sketch>** **<Dimensions As Equations>**

Command: **AMDIMDSP**
Parameters/<Equations>/Numeric: **E**

Construct a circle as shown in Figure 3.162. Then resolve it to a profile.

<Design> **<Circle>** **<Center, Radius>**

<Part> **<Sketch>** **<Profile>**

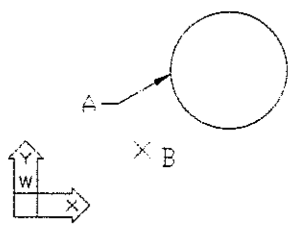


Figure 3.162 Circle constructed and resolved

Use the AMPARDIM command to add a parametric dimension. Then set the display to an isometric view and set the current layer to Solid. After that, use the AMEXTRUDE command to extrude the profile a distance of 40 units. (See Figure 3.163.)

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.162).]**
Select second object or place dimension: **[Select B (Figure 3.162).]**
Undo/Ordinate/Placement point/Enter dimension value: **40**
Select first object: **[Enter]**

Command: **8**

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Base** Termination: **Blind**
Size: Distance: **40** Draft Angle: **0**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

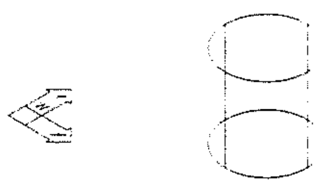


Figure 3.163 Extrude solid constructed

The next feature is a hexagonal hole. Set the display to the plan view by using the shortcut key [9]. Then set the current layer to Sketch. After that, use the POLYGON command to construct a hexagon as shown in Figure 3.164. After that, resolve it to a profile.

Command: **9**

[Mechanical Main] **[Layer Control]**

Current Layer: **Sketch**

<Design> **<Polygon>**

Command: **POLYGON**
Number of sides: **6**
Edge/<Center of polygon>: **[Select a point near the center of the cylinder.]**
Inscribed in circle/Circumscribed about circle (I/C) <I>: **[Enter]**
Radius of circle: **[Select another point within the cylinder.]**

<Part> **<Sketch>** **<Profile>**

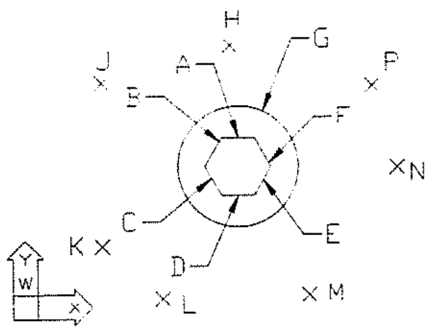


Figure 3.164 Hexagon constructed

In accordance with the following set of requirements, use the AMADDCON command to add appropriate geometric constraints.

Lines A and D (Figure 3.164)	horizontal
Lines B and E (Figure 3.164)	parallel
Lines C and F (Figure 3.164)	parallel

<Part> **<Sketch>** **<Add Constraints>** **<... >**

Now use the AMPARDIM command to set four linear dimensions and two angular dimensions such that the polygon becomes a regular hexagon.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.164).]**
Select second object or place dimension: **[Select H (Figure 3.164).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **10**

Select first object: **[Select B (Figure 3.164).]**
Select second object or place dimension: **[Select J (Figure 3.164).]**

This dimension should align with line B in Figure 3.164. If a vertical dimension is given, change it to an aligned dimension by selecting the A option.

Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **A**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **10**

Select first object: [**Select F (Figure 3.164).**]
Select second object or place dimension: [**Select P (Figure 3.164).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **A**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **10**

Select first object: [**Select C (Figure 3.164).**]
Select second object or place dimension: [**Select K (Figure 3.164).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **A**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **10**

Select first object: [**Select A (Figure 3.164).**]
Select second object or place dimension: [**Select B (Figure 3.164).**]
Specify dimension placement: [**Select M (Figure 3.164).**]

This dimension should be an angular dimension. Depending on where you select the first object, the second object, and the dimension placement location, Mechanical Desktop may give you a linear dimension or an angular dimension. If a linear dimension is given, change it to an angular dimension by selecting the N option.

Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **N**
Undo/Placement point/Enter dimension value: **120**

Select first object: [**Select F (Figure 3.164).**]
Select second object or place dimension: [**Select A (Figure 3.164).**]
Specify dimension placement: [**Select L (Figure 3.164).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **N**
Undo/Placement point/Enter dimension value: **120**

Select first object: [**Select F (Figure 3.164).**]
Select second object or place dimension: [**Select G (Figure 3.164).**]
Specify dimension placement: [**Select N (Figure 3.164).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select B (Figure 3.164).**]
Select second object or place dimension: [**Select G (Figure 3.164).**]
Specify dimension placement: [**Select H (Figure 3.164).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **5**
Select first object: [**Enter**]

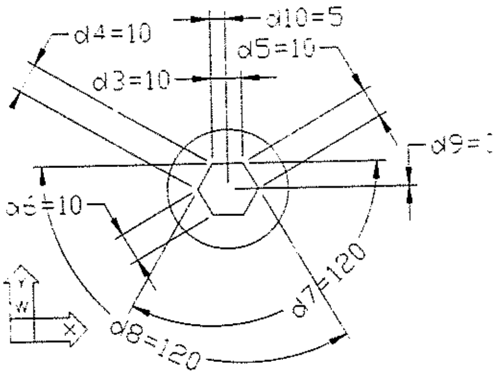


Figure 3.165 Hexagon fully constrained

Now set the display to an isometric view by using the shortcut key [8]. Then use the AMEXTRUDE command to extrude the hexagon to cut through the cylindrical central boss. (See Figure 3.166.)

Command: **8**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Cut** Termination: **Through**
Size: Draft Angle: **0**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

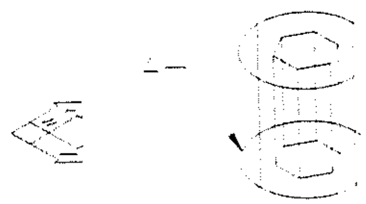


Figure 3.166 Hexagonal hole cut

The next feature is a sweep solid. To build this feature, you need to construct a path and a profile, then sweep the profile along the path. Because the path lies on a plane midway between the top and the bottom faces of the central boss, you need to construct a work plane and then set the sketch plane to this work plane. (See Figure 3.167.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset**

Offset: **20**
Create Sketch Plane **OK**]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.166).]**
Flip/<Accept>: **[Accept if the arrow is pointing upward.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as Figure 3.167.]**

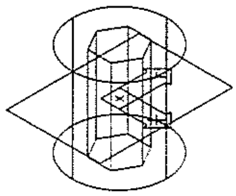


Figure 3.167 New work plane constructed

Use the shortcut key [9] to set the display to the plan view of the new work plane. Then construct an arc as shown in Figure 3.168. This is the path for the sweep solid.

Command: **9**

<Design> **<Arc>** **<3 Points>**

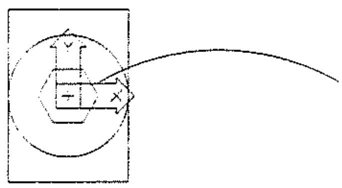


Figure 3.168 Arc constructed

Before resolving this arc to a path, take a moment to think how you can constrain the arc relative to the solid features that you have constructed. You can add some construction lines as shown in Figure 3.169. For Mechanical Desktop to treat these lines as construction lines, you must change their linetype to hidden or any linetype other than continuous.

<Design> **<Line>**

<Edit> **<Properties...>**

Now apply the AMPATH command to the arc and the lines. While resolving, you must include all the related construction lines. Although these lines will not be used in any subsequent operation such as extruding, revolving, or sweeping, they help you to constrain the main sketch. Here, the arc is the path and the lines are construction lines that will be used to constrain the arc.

<Part> <Sketch> <Path>

Command: **AMPATH**
Select objects: **[Select A, B, D, and F (Figure 3.169).]**
Select objects: **[Enter]**
Specify start point of path: **[Select G (Figure 3.169).]**

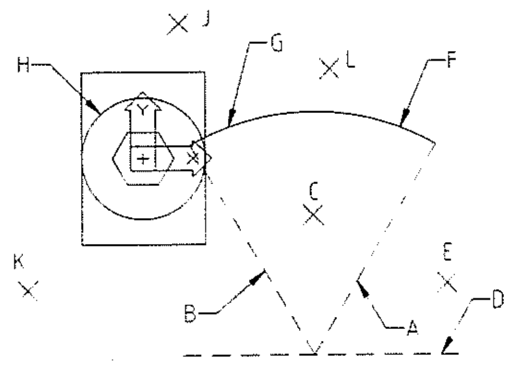


Figure 3.169 Construction lines created

To ensure that the center of the arc coincides with the end points of the two inclined construction lines, use the AMADDCON command to apply the Project constraint in accordance with the following table.

Specify point to project (use a snap mode): **END** of **[Select F (Figure 3.169).]**	Select line, circle, arc, ellipse, or spline: **[Select A (Figure 3.169).]**
Specify point to project (use a snap mode): **END** of **[Select G (Figure 3.169).]**	Select line, circle, arc, ellipse, or spline: **[Select B (Figure 3.169).]**
Specify point to project (use a snap mode): **CEN** of **[Select G (Figure 3.169).]**	Select line, circle, arc, ellipse, or spline: **[Select A (Figure 3.169).]**
Specify point to project (use a snap mode): **END** of **[Select A (Figure 3.169).]**	Select line, circle, arc, ellipse, or spline: **[Select D (Figure 3.169).]**

In addition, use the AMADDCON command to set line D to be horizontal.

Line D (Figure 3.169)	horizontal

Now use the AMPARDIM command to add parametric dimensions to constrain the arc and the lines relative to the central boss. (See Figure 3.170.)

<Part> <Add Dimension>

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.169).]**
Select second object or place dimension: **[Select B (Figure 3.169).]**
Specify dimension placement: **[Select C (Figure 3.169).]**
Undo/Placement point/Enter dimension value: **60**

Select first object: **[Select D (Figure 3.169).]**
Select second object or place dimension: **[Select A (Figure 3.169).]**

Specify dimension placement: **[Select E (Figure 3.169).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **N**
Undo/Placement point/Enter dimension value: **60**

Select first object: **[Select F (Figure 3.169).]**
Select second object or place dimension: **[Select L (Figure 3.169).]**
Undo/Ordinate/Placement point/Enter dimension value: **80**

Select first object: **[Select G (Figure 3.169).]**
Select second object or place dimension: **[Select H (Figure 3.169).]**
Specify dimension placement: **[Select K (Figure 3.169).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **65**

Select first object: **[Select G (Figure 3.169).]**
Select second object or place dimension: **[Select H (Figure 3.169).]**
Specify dimension placement: **[Select J (Figure 3.169).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **50**
Select first object: **[Enter]**

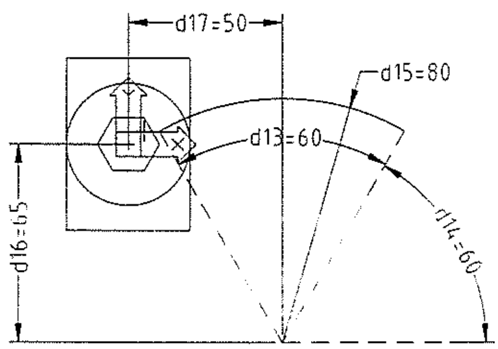


Figure 3.170 Path and its construction lines constrained

The path is now fully constrained. Select the Left Isometric item from the Model Views cascading menu of the Views pull-down menu to set the display to an isometric view seen from the left. Then use the AMWORKPLN to construct a work plane for the sweep profile at the start point of the path. (See Figure 3.171.)

<Views> **<Model Views>** **<Left Isometric>**

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[Work Plane Feature
1st Modifier **Normal to Start box**
Create Sketch Plane **OK**]

Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as Figure 3.171.]**

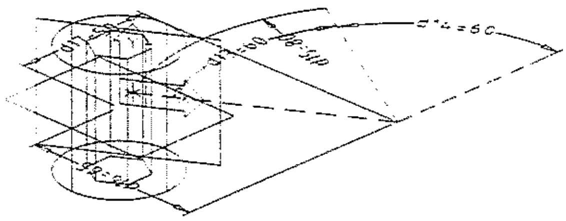


Figure 3.171 Work plane for sweep surface constructed

Use the shortcut key [9] to set the display to the plan view of the new sketch plane. Then construct an ellipse as shown in Figure 3.172. After that, resolve it to a profile.

Command: **9**

<Design> **<Ellipse>** **<Center>**

Command: **ELLIPSE**
Arc/Center/<Axis endpoint 1>: **C**
Center of ellipse: [**Select E (Figure 3.172).**]
Axis endpoint: [**Select A (Figure 3.172).**]
<Other axis distance>/Rotation: [**Select B (Figure 3.172).**]

<Part> **<Sketch>** **<Profile>**

Command: **AMPROFILE**
Select objects for sketch:
Select objects: **LAST**
Select objects: [**Enter**]

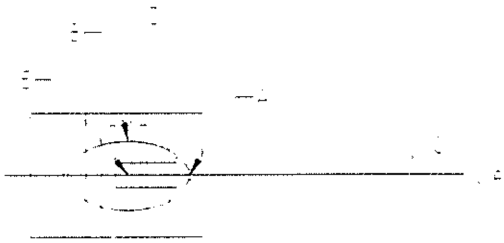


Figure 3.172 Ellipse drawn and resolved

Apply the AMPARDIM command. (See Figure 3.173.) To construct a dimension for the major or minor axis of an ellipse, you need to specify the object snap, QUA. Note that point E (Figure 3.172) is the start point of the path.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **QUA** of [**Select B (Figure 3.172).**]
Select second object or place dimension: **QUA** of [**Select A (Figure 3.172).**]

Specify dimension placement: **[Select C (Figure 3.172).]**
Undo/Ordinate/Placement point/Enter dimension value: **8**

Select first object: **QUA** of **[Select A (Figure 3.172).]**
Select second object or place dimension: **QUA** of **[Select B (Figure 3.172).]**
Specify dimension placement: **[Select D (Figure 3.172).]**
Undo/Ordinate/Placement point/Enter dimension value: **10**

Select first object: **[Select E (Figure 3.172).]**
Select second object or place dimension: **[Select B (Figure 3.172).]**
Specify dimension placement: **[Select D (Figure 3.172).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: **[Select E (Figure 3.172).]**
Select second object or place dimension: **[Select A (Figure 3.172).]**
Specify dimension placement: **[Select F (Figure 3.172).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Select first object: **[Enter]**

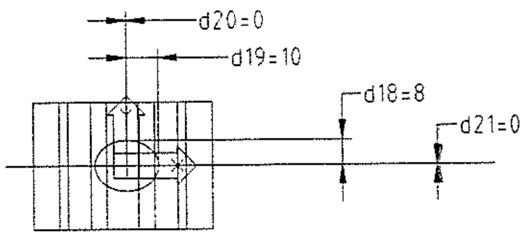


Figure 3.173 Ellipse fully constrained

With a fully constrained path and a fully constrained profile, use the AMSWEEP command to create a sweep solid and join the solid to the central boss. Then change the display to an isometric view. (See Figure 3.174.)

<Part> **<Sketched Features>** **<Sweep...>**

Command: **AMSWEEP**

[Operation	**Join**	Termination	**Path Only**
Body Type	**Normal**	Draft Angle	**0**
OK			]

Command: **8**

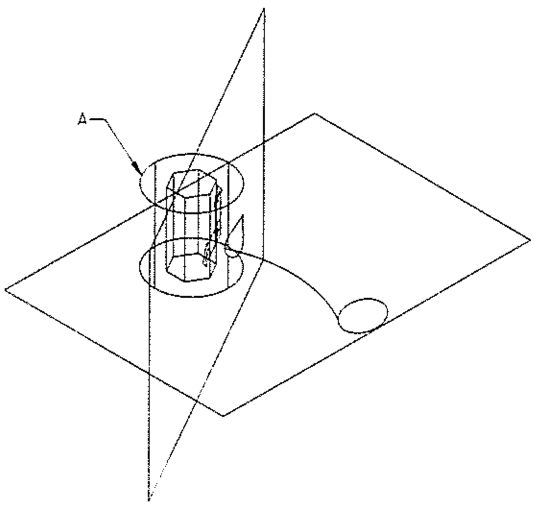


Figure 3.174 Sweep solid joined to the base solid

Note that the silhouette of the rib does not show on your screen because the variable DISPSILH is set to zero (by default). If you want to display the outlines of the curve faces, use the Performance tab of the PREFERENCES command to show silhouettes in wireframes. For the new setting to take effect, apply the REGEN command to regenerate the display.

<Assist> **<Preferences...>**

Command: **PREFERENCES**

[Performance
Show silhouettes in wireframes **OK**]

Command: **REGEN**

Now the two work planes for the sweep solid feature are not needed. Hide them with the AMVISIBLE command.

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[Part
Hide **Work Planes**
OK]

To prepare for a polar array, use the AMWORKAXIS command to create a work axis. (See Figure 3.175.)

<Part> **<Work Features>** **<Work Axis>**

Command: **AMWORKAXIS**
Select cylinder/cone/torus: **[Select A (Figure 3.174).]**

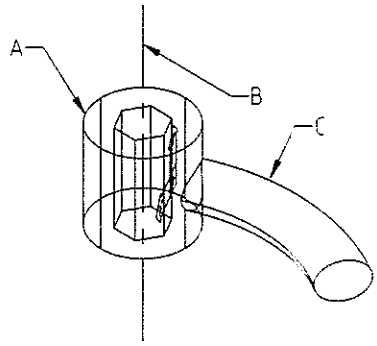


Figure 3.175 Outlines displayed, work planes hidden, and work axis created

There are three ribs in the model. To produce the other two, you will perform a polar array. Before you can array the rib, you need to set the sketch plane parallel to the plane of the array. Use the AMSKETCH command to set the sketch plane to the top face of the central boss. Then use the AMARRAY command to construct the other two ribs. (See Figure 3.176.)

<Part> **<Sketch>** **<Sketch Plane>**

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.175).]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **WORLDX**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as Figure 3.176.]**

<Part> **<Placed Features>** **<Array...>**

Command: **AMARRAY**
Select feature: **[Select C (Figure 3.175).]**

[Polar
Number of Instances **3**
Angle Type **Full Circle**
Rotate as Copied
OK]

Select work point or work axis for center of array: **[Select B (Figure 3.175).]**

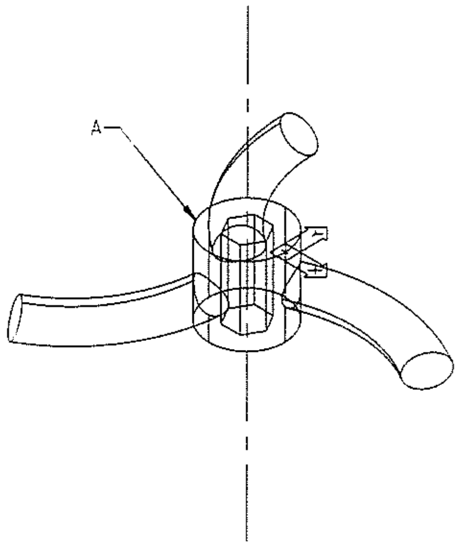


Figure 3.176 Ribs arrayed

The next feature is a solid of revolution. You will construct this feature by revolving a circle about an axis. The circle should reside on a plane that is parallel to the worldxz plane and passes through the work axis.

To construct the required work plane, you will first construct an intermediate work plane as a reference. (See Figure 3.177.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[Work Plane Feature
1st Modifier **World XZ**
OK]

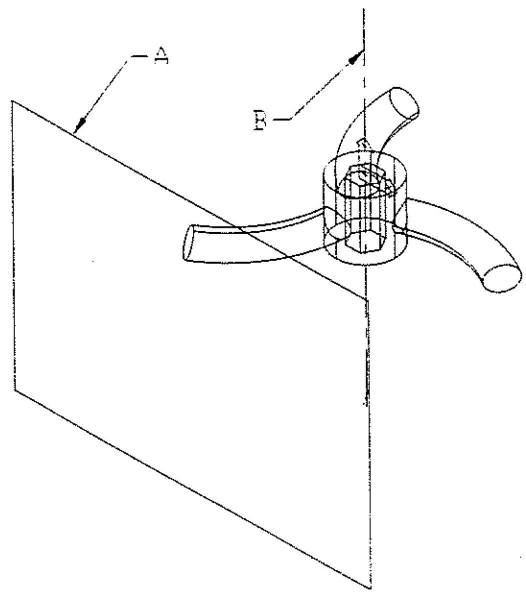


Figure 3.177 Reference work plane constructed parallel to worldxz

As in Figure 3.177, the newly constructed work plane is nonparametric. It has no parametric relationship with the solid model. Its primary function is to act as a reference for making a parametric work plane that is parallel to the worldxz plane and passes through the work axis of the base feature. Repeat the AMWORKPLN command to construct the required work plane for the profile. (See Figure 3.178.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[Work Plane Feature
1st Modifier **On Edge/Axis**
2nd Modifier **Planar Parallel**
Create Sketch Plane **OK**]

worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select B (Figure 3.177).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.177).]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **WORLDX**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as Figure 3.178.]**

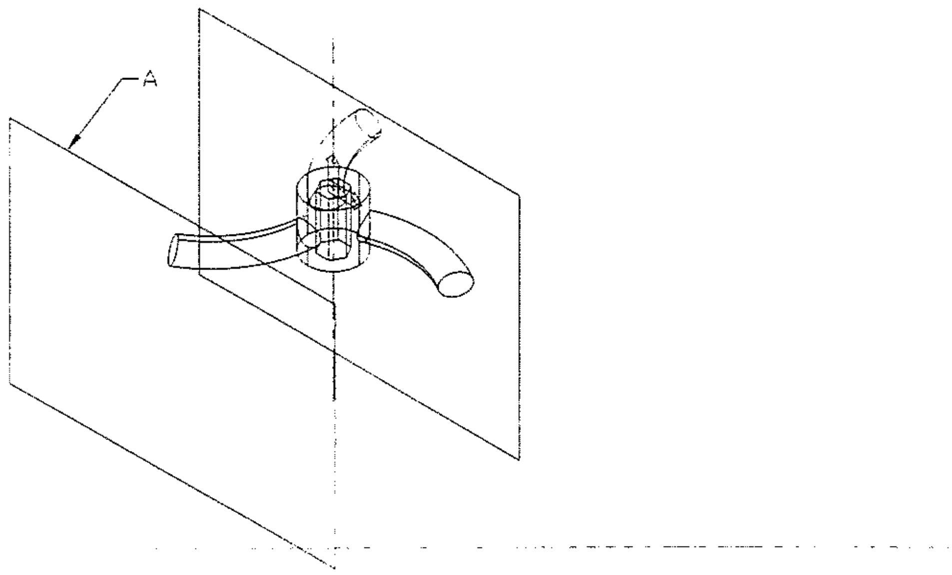


Figure 3.178 Parametric work plane constructed

Figure 3.178 shows two work planes: a nonparametric work plane that is aligned with the worldxz plane and a parametric work plane that is parallel to the worldxz plane and passes through the work axis of the solid model. Because the nonparametric work plane A (Figure 3.178) is not required, use the AMVISIBLE command to hide it. (See Figure 3.179.)

<Part> **<Visibilities...>**

Command: **AMVISIBLE**

[Part
Hide **Select<**
OK]

Select part objects to hide: **[Select A (Figure 3.178).]**
Select part objects to hide: **[Enter]**

[**OK**]

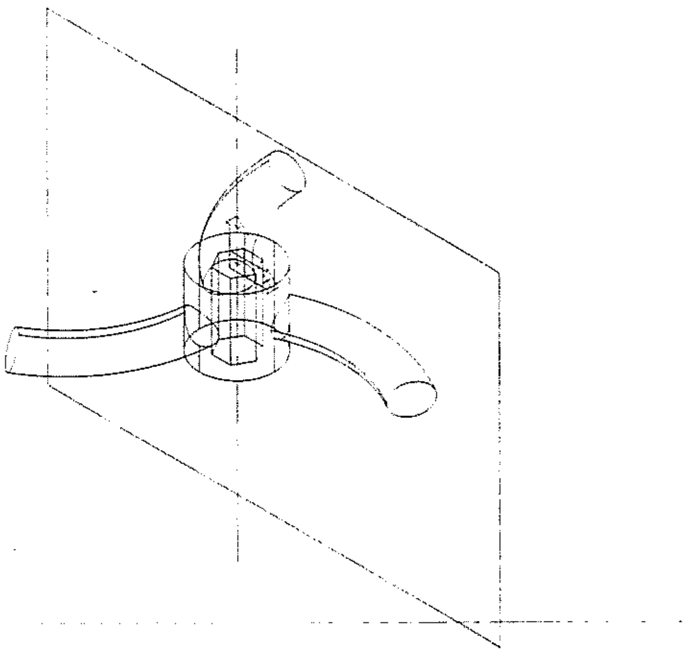


Figure 3.179 Nonparametric work plane hidden

Now set the display to the plan view of the sketch plane by using the shortcut key [9]. Then construct a circle as shown in Figure 3.180.

Command: **9**

<Design> **<Circle>** **<Center, Radius>**

Command: **CIRCLE**

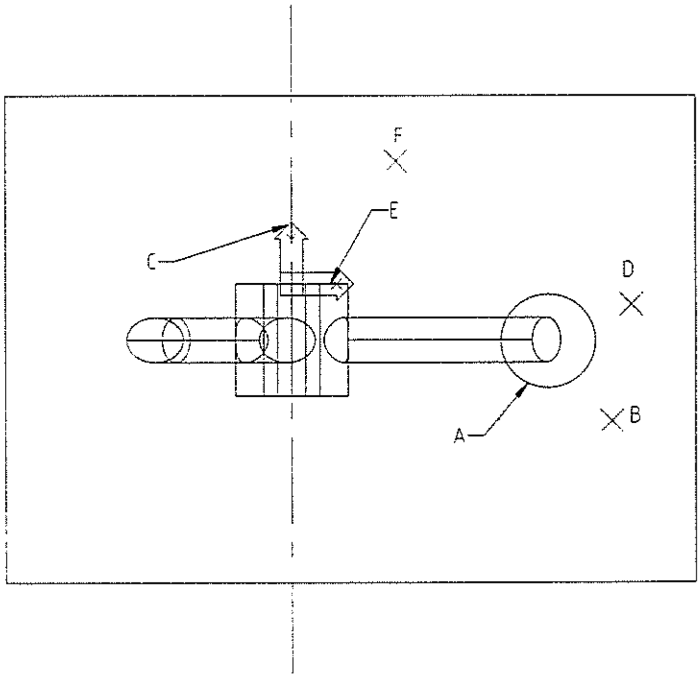


Figure 3.180 Circle drawn on new work plane

Use the AMPROFILE command to resolve the circle to a profile. Then use the AMPARDIM command to fully constrain the circle. (See Figure 3.181.)

<Part> **<Sketch>** **<Profile>**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.180).**]
Select second object or place dimension: [**Select B (Figure 3.180).**]
Undo/Radius/Ordinate/Placement point/Enter dimension value: **30**

Select first object: [**Select A (Figure 3.180).**]
Select second object or place dimension: [**Select C (Figure 3.180).**]
Specify dimension placement: [**Select F (Figure 3.180).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **H**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **90**

Select first object: [**Select A (Figure 3.180).**]
Select second object or place dimension: [**Select E (Figure 3.180).**]
Specify dimension placement: [**Select D (Figure 3.180).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **20**
Select first object: [**Enter**]

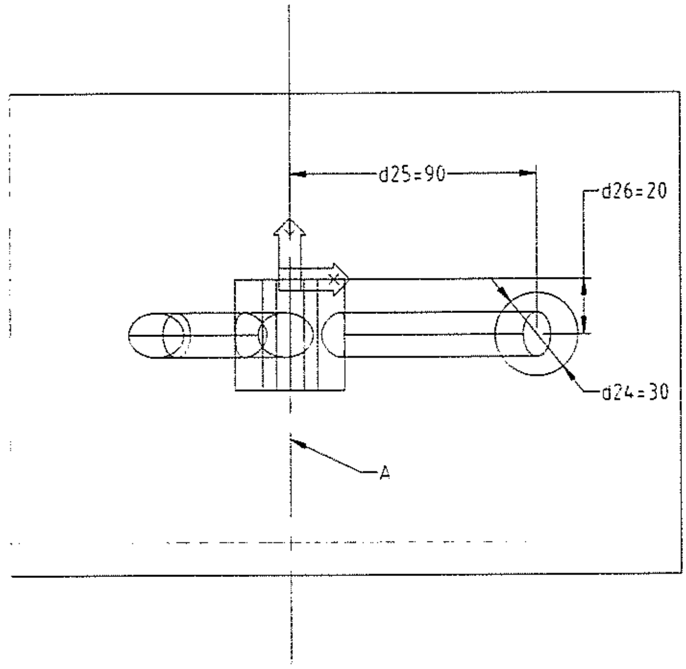


Figure 3.181 Circle fully constrained

Set the display to an isometric view. Then use the AMREVOLVE command to revolve the circle to form a solid of revolution. After that, hide all the work planes. (See Figure 3.182.)

Command: **8**

<Part> **<Sketched Features>** **<Revolve...>**

Command: **AMREVOLVE**

[Operation **Join** Termination **Full**
OK]

Select revolution axis: [**Select A (Figure 3.181).**]

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[Part
Active Part:
Hide **Work Planes**
OK]

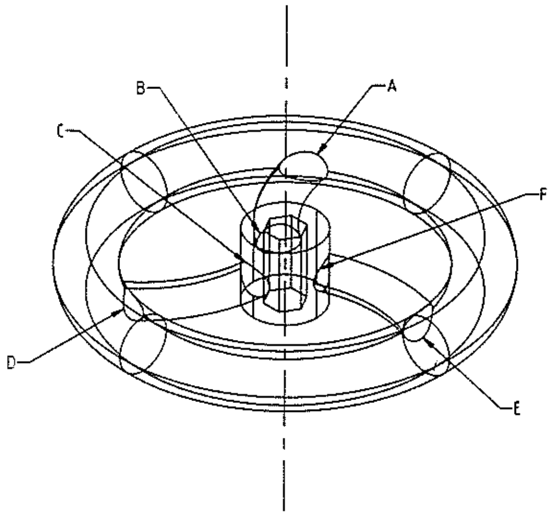


Figure 3.182 Rim constructed

To complete the hand wheel model, use the AMFILLET command to place six fillets of fixed width. (See Figure 3.183.)

<Part> **<Placed Features>** **<Fillet...>**

Command: **AMFILLET**

[**Fixed Width** **5** **Apply**]

Select edge: [**Select A, B, C, D, E, and F (Figure 3.182).**]
Select edge: [**Enter**]

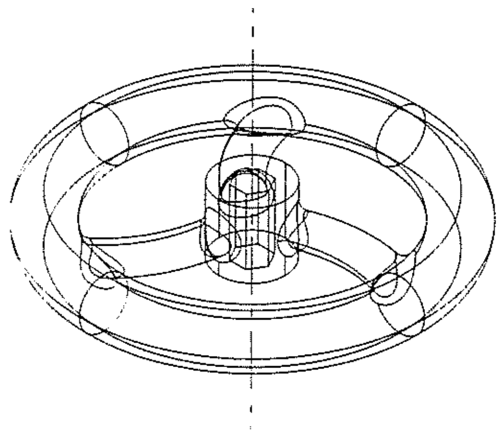


Figure 3.183 Fillets placed

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Hwheel.dwg**

In constructing this model, you practiced using the three kinds of sketched solid features (extrude feature, sweep feature, and revolve feature) in model making, and placing array and fillet features.

Lower Suspension Arm Project

Figure 3.184 shows the rendered image of the lower suspension arm of a scale model car. You will construct an extrude solid from the top view of the solid. Then you will construct an extrude solid from the side view of the model and intersect the second solid feature with the base solid feature. Finally, you will place hole and fillet features to complete the model.

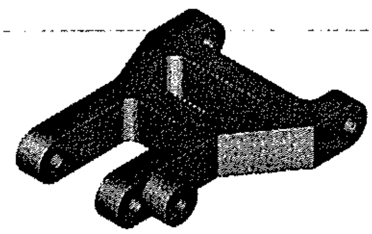


Figure 3.184 Rendered image of the lower suspension arm of a scale model car

Start a new single-part drawing. Use Solid.dwt as the template. Then construct a sketch as shown in Figure 3.185. After that, resolve it to a profile.

<File> **<New Part File...>**

<Design> **<Polyline>**

<Part> **<Sketch>** **<Profile>**

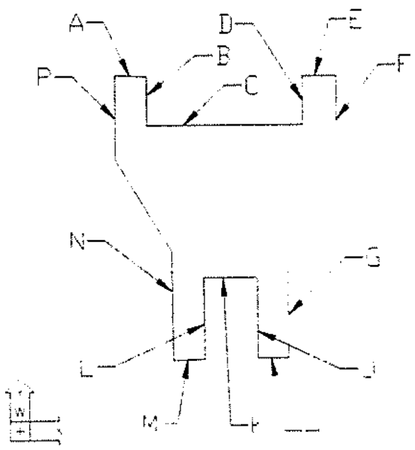


Figure 3.185 Profile constructed and resolved

According to the geometric constraint requirements shown in the following table, delete the inappropriate constraints and add the necessary ones.

Lines A, C, E, H, K, and M (Figure 3.185)	horizontal
Lines B, D, F, G, J, L, N, and P (Figure 3.185)	vertical
Lines A and E (Figure 3.185)	collinear
Lines H and M (Figure 3.185)	collinear
Lines F and P (Figure 3.185)	equal length
Lines G and N (Figure 3.185)	equal length

Use the AMPARDIM command to add parametric dimensions as shown in Figure 3.186.

<Part> **<Add Dimension>**

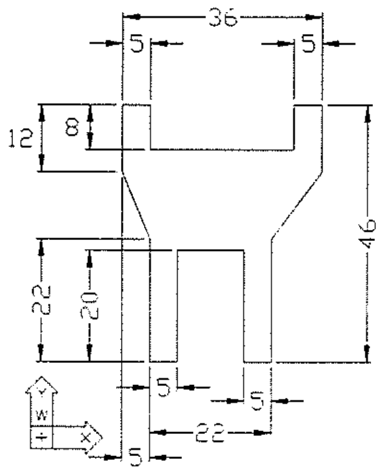


Figure 3.186 Profile fully constrained

Set the display to an isometric view by using the shortcut key [8]. Then set the current layer to Solid. After that, use the AMEXTRUDE command to extrude the profile a distance of 8 units. (See Figure 3.187.)

Command: **8**

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation **Base** Termination **Blind**
Size: Distance: **8** Draft Angle: **0**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

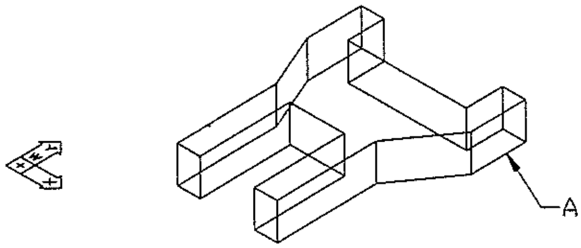


Figure 3.187 Profile extruded

Set the sketch plane to a vertical face of the extrude solid. (See Figure 3.188.)

<Part> <Sketch> <Sketch Plane>

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.187).]**
Next/<Accept>: **[Accept if the vertical face is highlighted.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select A (Figure 3.187).]**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that of the icon shown in Figure 3.188).]**

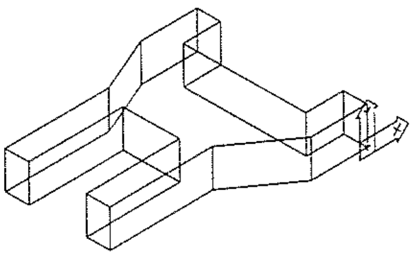


Figure 3.188 Sketch plane set to the vertical face

Set the display to the plan view of the new sketch plane. Then set the current layer to Sketch. After that, construct a sketch as shown in Figure 3.189. Then resolve it to a profile.

Command: **9**

[Mechanical Main] [Layer Control]

Current Layer: **Sketch**

<Design> <Polyline>

<Part> <Sketch> <Profile>

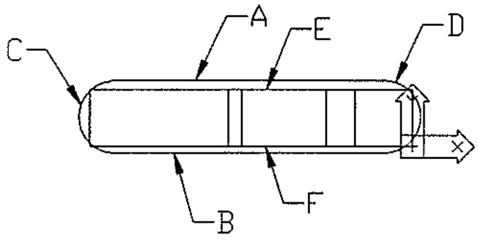


Figure 3.189 Display set, sketch constructed

Set the geometric constraints in accordance with the following requirements:

Line A and arc D (Figure 3.189)	tangent
Arc D and line B (Figure 3.189)	tangent
Line B and arc C (Figure 3.189)	tangent
Arc C and line A (Figure 3.189)	tangent
Lines A and E (Figure 3.189)	collinear
Lines B and F (Figure 3.189)	collinear

Add parametric dimensions to the profile as shown in Figure 3.190.

<Part> **<Add Dimension>**

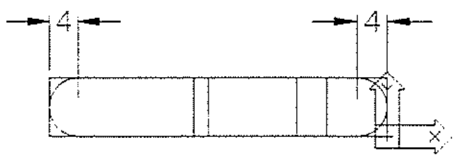


Figure 3.190 Profile fully constrained

Set the display to an isometric view. Then extrude the resolved profile into a solid, and intersect the extrude solid with the base solid. (See Figure 3.191.)

Command: **8**

<Part> **<Sketch Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation **Intersect** Termination **Through**
Size: Draft Angle: **0**
OK]

Direction Flip/<Accept>: **[Accept if the direction of the arrow is pointing toward the base solid.]**

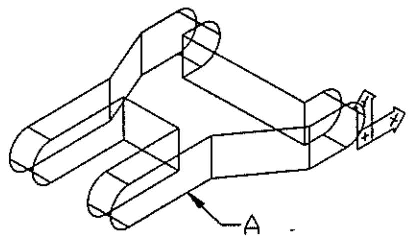


Figure 3.191 Second extrude solid intersected with base solid

Set the sketch plane to a new position. Then construct a circle as shown in Figure 3.192.

<Part> <Sketch> <Sketch Plane>

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.191).]**
Next/<Accept>: **[Accept if the vertical face is highlighted.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select A (Figure 3.191).]**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that of the icon shown in Figure 3.192).]**

<Design> <Circle> <Center, Radius>

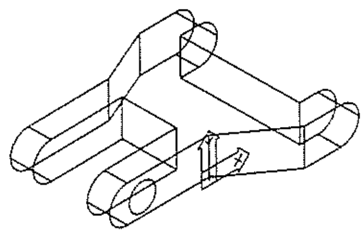


Figure 3.192 Circle constructed on new sketch plane

Resolve the circle to a profile. Then add parametric dimensions as shown in Figure 3.193.

<Part> <Sketch> <Profile>

<Part> <Add Dimension>

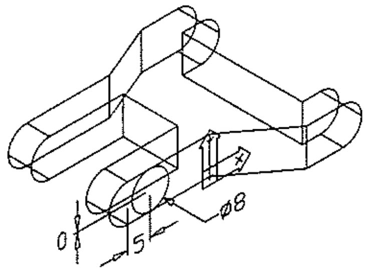


Figure 3.193 Circle resolved and constrained

Now extrude the profile to a solid and join it to the base feature. See Figure 3.194.

<Part> **<Sketch Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation **Join** Termination **Blind**
Size: Distance: **5** Draft Angle: **0**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing away from the solid.]**

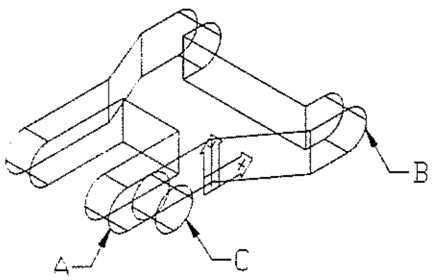


Figure 3.194 Circle extruded and joined

As shown in Figure 3.195, place two concentric through holes of diameter 3 units and a blind tapped hole (M3 size). The drill size and depth of the blind tapped hole are 2.3 and 7 respectively.

<Part> **<Placed Features>** **<Hole...>**

Command: **AMHOLE**

[Operation **Drilled** Termination **Through**
Placement **Concentric**
Drill Size **3**
Apply<]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.194).]**
Select concentric edge: **[Select A (Figure 3.194).]**

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select B (Figure 3.194).]**
Select concentric edge: **[Select B (Figure 3.194).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Enter]**

[Operation **Drilled** Termination **Blind**
Tapped...]

[**Tapped** Thread Options **3** **Full Depth**
OK]

[Placement **Concentric**
Drill Size Depth **7** Dia **2.3** PT Angle **118**
Apply<]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>:>: **[Select C (Figure 3.194).]**
Select concentric edge: **[Select C (Figure 3.194).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Enter]**

[**Exit**]

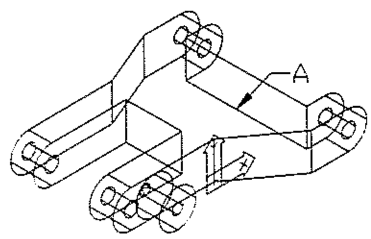


Figure 3.195 Hole features placed

Note in Figure 3.195 that the thread will not show in the 3D model. However, it will show in the associative engineering drawing. Now set the sketch plane to the bottom face of the solid as shown in Figure 3.196.

<Part> <Sketch> <Sketch Plane>

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.195).]**
Next/<Accept>: **[Accept if the horizontal, bottom face is highlighted.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select A (Figure 3.195).]**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that of the icon shown in Figure 3.196).]**

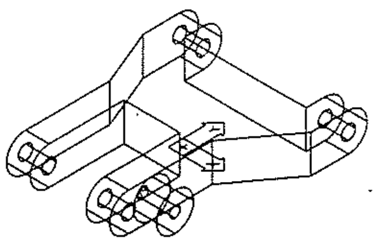


Figure 3.196 Sketch plane set to the bottom face

Set the display to the plan view. Then construct a polyline as shown in Figure 3.197. After that, resolve it to a profile.

Command: **9**

<Design> **<Polyline>**

<Part> **<Sketch>** **<Profile>**

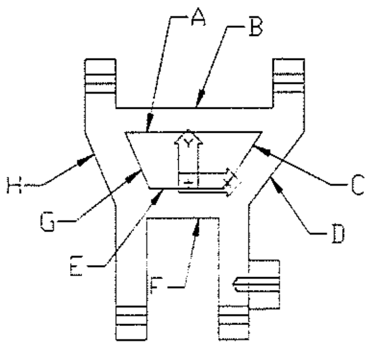


Figure 3.197 Display set to plan view, lines constructed

Add geometric constraints in accordance with the following requirements and add four parallel parametric dimensions to constrain the profile as shown in Figure 3.198.

Lines A and B (Figure 3.197)	parallel
Lines C and D (Figure 3.197)	parallel
Lines E and F (Figure 3.197)	parallel
Lines G and H (Figure 3.197)	parallel

<Part> **<Add Dimension>**

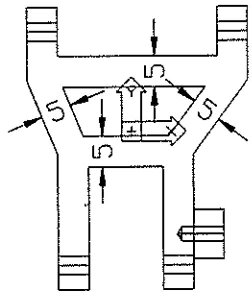


Figure 3.198 Profile constrained

Set the display to an isometric view. Then extrude the profile to cut an opening. (See Figure 3.199.)

Command: 8

<Part> **<Sketch Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation **Cut** Termination **Through**
Size: Draft Angle **0**
OK]

Direction Flip/<Accept>: [**Accept if the arrow is pointing toward the solid.**]

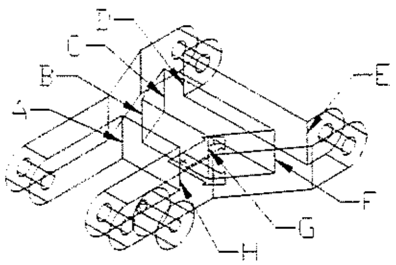


Figure 3.199 Profile extruded and cut

To complete the model, add eight fillets of radius 2 units. (See Figure 3.200.)

<Part> **<Placed Features>** **<Fillet...>**

Command: **AMFILLET**

[**Constant** Radius **2**
Apply]

Select edges: [**Select A, B, C, D, E, F, G, and H (Figure 3.199).**]
Select edges: [**Enter**]

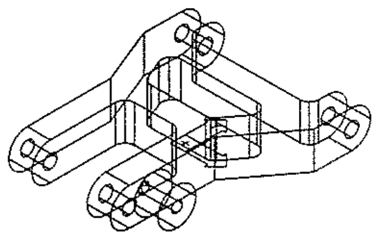


Figure 3.200 Fillet features placed

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Arm_l.dwg**

In making this model, you learned how to combine sketched solid features by intersecting, cutting, and joining. In addition, you made a number of placed features. While constructing this model, you made use of existing faces of the solid as sketch planes and you did not use any work plane.

Remote Control Project

Figure 3.201 shows the rendered image of the exploded assembly of the casing of a remote control unit. This model consists of two solid parts: the upper casing and the lower casing. You will construct the solid models in this chapter and assemble them together in the next chapter.

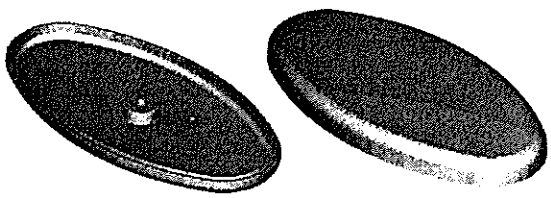


Figure 3.201 Rendered image of the exploded view of the assembly

Upper Casing

Figure 3.202 shows the upper casing of the assembly. Before starting to make this model, you need to analyze the model and think critically about the course of action that you will take. This model has eight features:

Main body	You will construct a sketch and construct an extrude solid.
Free-form surface	You will construct a loft u surface and use the surface to cut the solid.
Fillet edges	You will place a fillet feature.
Shell	You will place a shell feature.
Boss	You will construct a sketch and extrude it to meet the free-form surface.
Threaded hole	You will place a hole feature.

Web	You will construct a sketch and extrude it to meet the top face.
Recess	You will construct a sketch and extrude it to cut the main body.

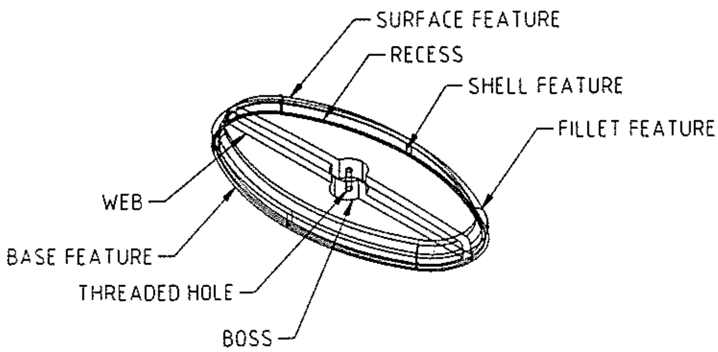


Figure 3.202 Solid features of the upper casing

Start a new single-part drawing. Use the template Solid.dwt. As shown in Figure 3.203, use the ARC command to construct four arcs.

<File> **<New Part File >**

Template File: **Solid.dwt**

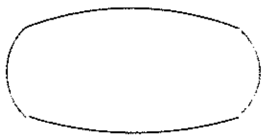


Figure 3.203 Sketch of four arcs

Use the AMPROFILE command to resolve the rough sketch to a profile. (See Figure 3.204.) Your resolved profile may not appear the same as the one in Figure 3.204.

<Part> **<Sketch>** **<Profile>**

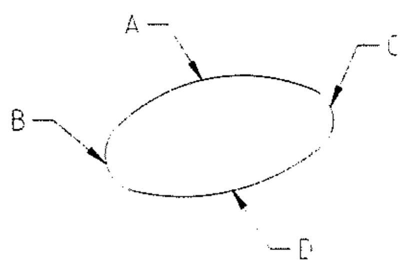


Figure 3.204 Resolved sketch

Use the AMADDCON command to add geometric constraints to the profile in accordance with the following requirements. (See Figure 3.205.)

Arcs A and B (Figure 3.204)	tangent
Arcs A and C (Figure 3.204)	tangent
Arcs D and B (Figure 3.204)	tangent
Arcs D and C (Figure 3.204)	tangent
Arcs A and D (Figure 3.204)	equal radii
Arcs B and C (Figure 3.204)	equal radii
Arcs A and D (Figure 3.204)	centers have same X values

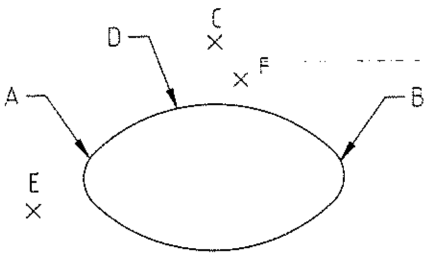


Figure 3.205 Geometric constraints applied

Use the AMDIMDSP command to set the display of dimensions as equations so that the parameter names as well as the values of the dimensions are displayed.

<Part> **<Sketch>** **<Dimensions As Equations>**

Now use the AMPARDIM command to add three parametric dimensions to fully constrain the sketch. The first dimension is the center distance of the two smaller arcs. Accept the default value. The second dimension is the radius of the smaller arc. It is 1/4 the center distance. If the center distance is d0, this dimension should be d0/4. Compare Figure 3.206 with your screen display. If the parameter name of the first dimension in your sketch is not d0, write down its name and replace d0 with this name in the following description. The third dimension is the radius of the larger arc. It is five times that of the smaller arc. In Figure 3.206, the parameter name of the smaller arc is d1. Because the larger arc is five times the length of the smaller arc, its value should be d1*5.

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.205).**]
Select second object or place dimension: [**Select B (Figure 3.205).**]
Specify dimension placement: [**Select C (Figure 3.205).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: [**Enter to accept the default value.**]

Select first object: [**Select A (Figure 3.205).**]
Select second object or place dimension: [**Select E (Figure 3.205).**]
Undo/Ordinate/Placement point/Enter dimension value: **d0/4**

Select first object: **[Select D (Figure 3.205).]**
Select second object or place dimension: **[Select F (Figure 3.205).]**
Undo/Ordinate/Placement point/Enter dimension value: **d1*5**
Solved fully constrained sketch.
Select first object: **[Enter]**

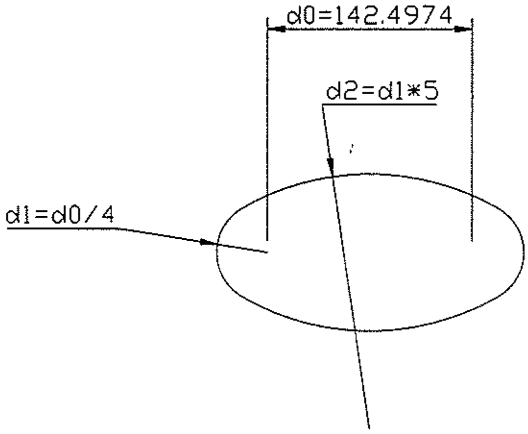


Figure 3.206 Parametric dimension added

Now the resolved sketch is fully constrained. Change the center distance of the two smaller arcs (dimension d0 of Figure 3.206) to 70, using the AMMODDIM command. Set the display to an isometric view by using the shortcut key [8]. Then set the current layer to Solid. After that, use the AMEXTRUDE command to extrude the sketch a distance equal to the radius of the large arc. Set the draft angle to –1°. (See Figure 3.207.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: **[Select dimension d0 of Figure 3.206.]**
New value for dimension: **70**
Select dimension to change: **[Enter]**

Command: **8**

[Mechanical Main] **[Layer Control]**

Current Layer: **Solid**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Base** Termination: **Blind**
Size: Distance: **d2** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

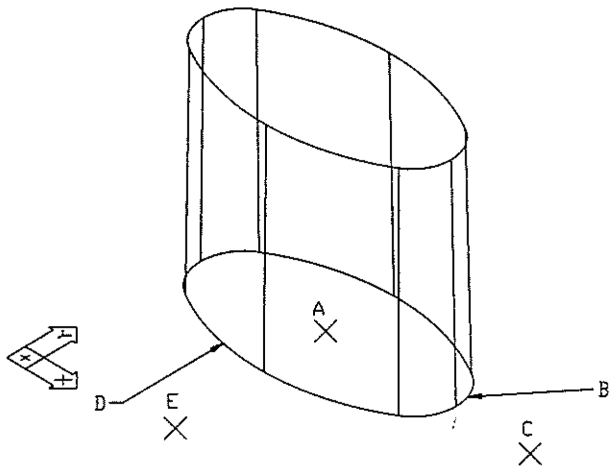


Figure 3.207 Profile extruded

The next feature is a free-form surface. To locate a surface parametrically in relation to the solid, you need a work point as a reference. Use the AMWORKPT command to create a work point. Then use the AMPARDIM command to dimension the work point. (See Figure 3.208.)

<Part> **<Work Features>** **<Work Point>**

Command: **AMWORKPT**
Location on sketch plane: **[Select A (Figure 3.207).]**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.207).]**
Select second object or place dimension: **[Select B (Figure 3.207).]**
Specify dimension placement: **[Select C (Figure 3.207).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Select first object: **[Select A (Figure 3.207).]**
Select second object or place dimension: **[Select D (Figure 3.207).]**
Specify dimension placement: **[Select E (Figure 3.207).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Select first object: **[Enter]**

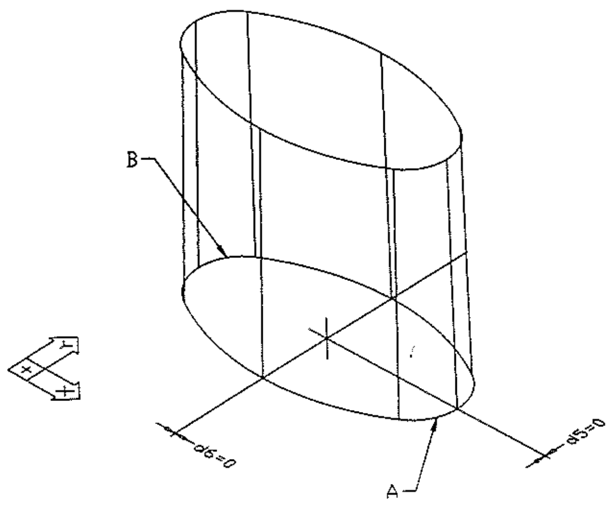


Figure 3.208 Work point constructed and constrained

You will construct a loft u surface from three splines. To get a better understanding of the sizes and shapes of the splines relative to the solid, you will set up a work plane to locate the splines. To make this work plane, you will use the AMWORKAXIS command to construct two work axes.

<Part> <Work Features> <Work Axis>

Command: **AMWORKAXIS**
Select cylinder/cone/torus: **[Select A (Figure 3.208).]**

<Part> <Work Features> <Work Axis>

Command: **AMWORKAXIS**
Select cylinder/cone/torus: **[Select B (Figure 3.208).]**

Now use the AMWORKPLN command to construct a work plane and set the sketch plane to that work plane. (See Figure 3.209.)

<Part> <Work Features> <Work Plane...>

Command: **AMWORKPLN**

[1st Modifier **On Edge/Axis**
2nd Modifier **On Edge/Axis**
Create Sketch Plane **OK**]

worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select the left work axis.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select the right work axis.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **WORLDX**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that in Figure 3.209.]**

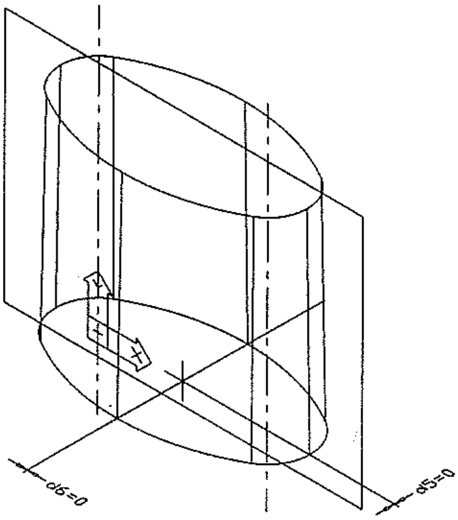


Figure 3.209 Work axes and work plane constructed

Use the LAYER command to construct a new layer, Surface, and set the current layer to Surface. Then construct three splines for making a loft u surface. (See Figure 3.210.)

<Assist> **<Format>** **<Layer...>**

New layer: **Surface**
Current layer: **Surface**

<Design> <Spline>

Command: **SPLINE**
Object/<Enter first point>: **-40,7**
Enter point: **35,10**
Close/Fit Tolerance/<Enter point>: **110,7**
Close/Fit Tolerance/<Enter point>: **[Enter]**
Enter start tangent: **[Enter]**
Enter end tangent: **[Enter]**

<Construct> <Copy>

Command: **COPY**
Select objects: **LAST**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,-2,40**
Second point of displacement: **[Enter]**

<Construct> <Copy>

Command: **COPY**
Select objects: **LAST**
Select objects: **[Enter]**
<Base point or displacement>/Multiple: **0,0,-80**
Second point of displacement: **[Enter]**

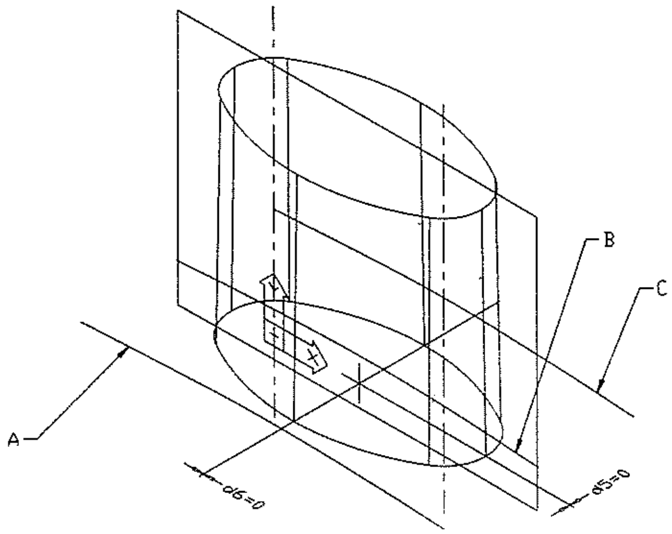


Figure 3.210 Three splines constructed

Using the three splines as U lines, use the AMLOFTU command to construct a loft u surface. Align the curve direction, smoothen the curves, and respace the curve ends. (See Figure 3.211.)

<Surface> **<Create Surface>** **<LoftU...>**

Command: **AMLOFTU**

Select U wires: [**Select A (Figure 3.210).**]
Select U wires: [**Select B (Figure 3.210).**]
Select U wires: [**Select C (Figure 3.210).**]
Select U wires: [**Enter**]

[Input Wires Curve Direction **Align** Curve Fit **Smooth**
Curve Ends Fit **Respace**
OK]

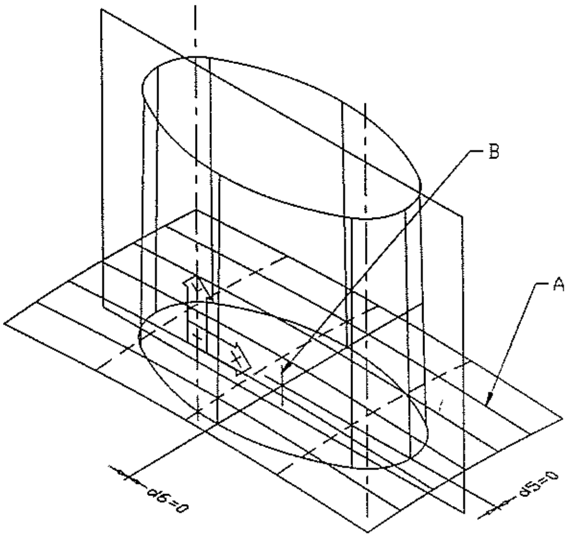


Figure 3.211 Loft u surface constructed

Now use the AMSURFCUT command to cut the solid with the surface. Then set the current layer to Sketch and turn off layer Surface. After that, hide the work plane. (See Figure 3.212.)

[Mechanical Main] **[Layer Control]**

Off layer: **Surface**
Current Layer: **Sketch**

<Part> **<Placed Features>** **<Surface Cut>**

Command: **AMSURFCUT**
Type/<Select surface>: **[Select A (Figure 3.211).]**
Select work point: **[Select B (Figure 3.211).]**
Portion to remove: Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

[Mechanical Main] **[Layer Control]**

Off layer: **Surface**

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[Active part
Hide **Work Planes**
OK **]**

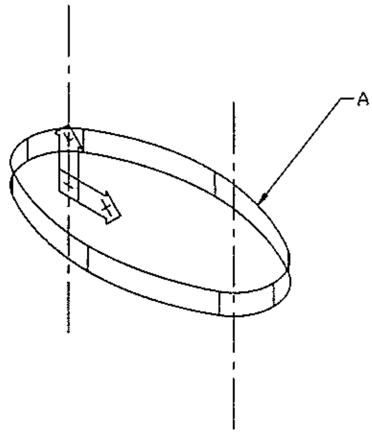


Figure 3.212 Surface feature cut on the solid

Note that Mechanical Desktop has two commands to cut a solid with a surface, AMSOLCUT and AMSURFCUT. The AMSOLCUT command cuts a native solid; the AMSURFCUT command cuts a parametric solid.

Now use the AMFILLET command to place a constant-radius fillet at the upper edges. (See Figure 3.213.)

<Part> **<Placed Features>** **<Fillet...>**

Command: **AMFILLET**

[**Constant** Radius: **4**
Apply]

Select edges: [**Select A (Figure 3.212).**]
Select edges: [**Enter**]

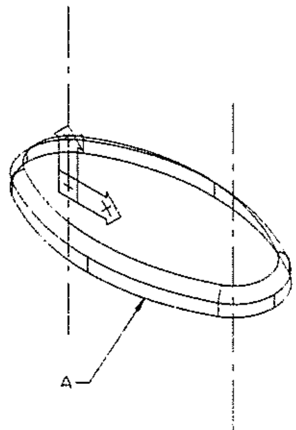


Figure 3.213 Fillet feature placed

The next feature is a thin shell. Use the AMSHELL command to carve out the core. Because the lower face is to be opened, exclude it during the shelling process. (See Figure 3.214.)

<Part> **<Placed Features>** **<Shell...>**

Command: **AMSHELL**

[Default Thickness **Inside** **2**
Excluded Faces **Add<**]

Select faces to exclude: [**Select A (Figure 3.213).**]
Next/<Accept>: [**Accept if the bottom face is highlighted.**]
Select faces to exclude: [**Enter**]

[**OK**]

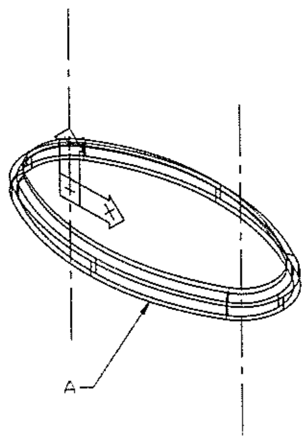


Figure 3.214 Shell feature placed

The two work axes are no longer needed. Hide them with the AMVISIBLE command.

<Part> **<Visibility...>**

Command: **AMVISIBLE**

[**Active part**
Hide **Work Axes**
OK]

As shown in Figure 3.215, use the AMSKPLN command to set the sketch plane to the bottom face A (Figure 3.214) of the solid. Then construct a circle.

<Part> **<Sketch>** **<Sketch Plane>**

<Design> **<Circle>** **<Center, Radius>**

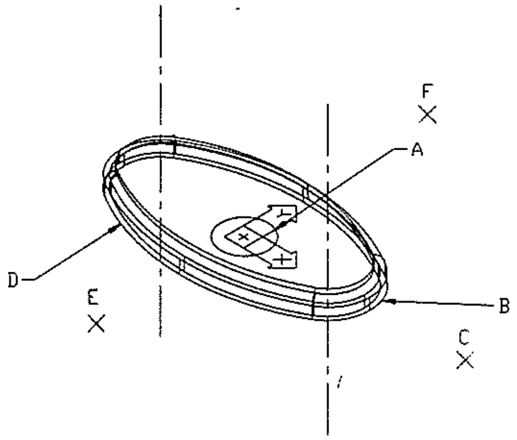


Figure 3.215 Sketch plane set, circle constructed

Resolve and constrain the circle. (See Figure 3.216.)

<Part> **<Sketch>** **<Profile>**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.215).**]
Select second object or place dimension: [**Select B (Figure 3.215).**]
Specify dimension placement: [**Select C (Figure 3.215).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.215).**]
Select second object or place dimension: [**Select D (Figure 3.215).**]
Specify dimension placement: [**Select E (Figure 3.215).**]
Undo/Ordinate/Placement point/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.215).**]
Select second object or place dimension: [**Select F (Figure 3.215).**]
Undo/Ordinate/Placement point/Enter dimension value: **12**
Select first object: [**Enter**]

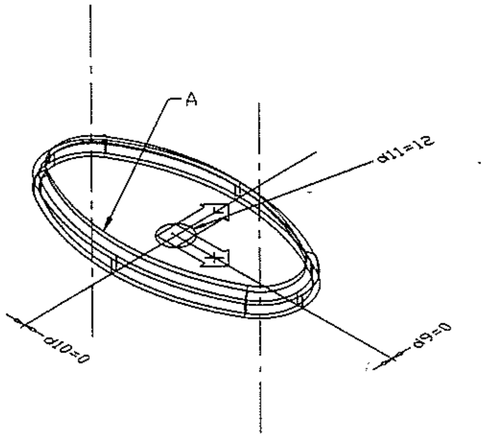


Figure 3.216 Profile fully constrained

Use the AMEXTRUDE command to extrude the circle to meet the free-form face and join the base solid feature. (See Figure 3.217.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Join** Termination: **To Face**
Size: Draft Angle: **1**
OK]

Select termination Face: [**Select A (Figure 3.216).**]
Next/<Accept>: [**Accept if the upper face is highlighted.**]

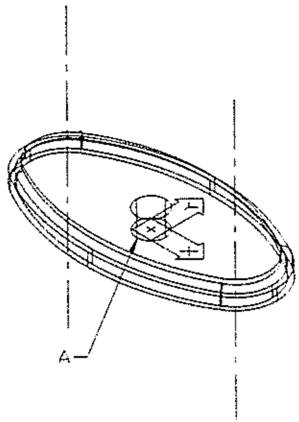


Figure 3.217 Profile extruded and joined

To construct a threaded hole concentric to the boss, use the AMHOLE command. The thread size is M3 (metric thread with diameter 3 units). The drill size is 2.3 units. The depth of the hole is 8 units. The depth of thread is 6 units. (See Figure 3.218.)

<Part> **<Placed Features>** **<Hole...>**

Command: **AMHOLE**

[Operation: **Drilled** Termination: **Blind**
Tapped...]

[Tapped Thread Options **3** **Depth: 6**
OK]

[Placement **Concentric**
Drill Size Depth: **8** Dia: **2.3** PT Angle: **118**
Apply<]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.217).]**
Select concentric edge: **[Select A (Figure 3.217).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Enter]**

[Exit]

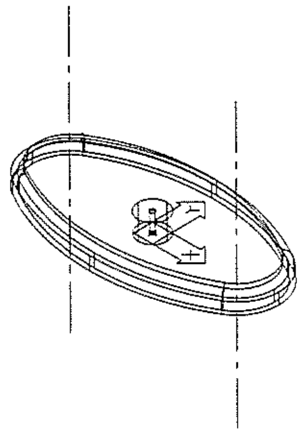


Figure 3.218 Threaded hole feature placed

The seventh feature is a web. Set the display to a plan view with the shortcut key [9]. Then construct a rectangle as shown in Figure 3.219.

Command: **9**

<Design> **<Rectangle>**

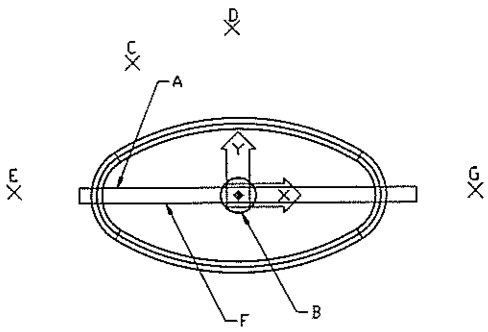


Figure 3.219 Rectangle drawn

Resolve the rectangle to a profile, using the AMPROFILE command. Then add four parametric dimensions to fully constrain it. After that, set the display to an isometric view with the shortcut key [8]. (See Figure 3.220.)

<Part> <Sketch> <Profile>

<Part> <Add Dimension>

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.219).**]
Select second object or place dimension: [**Select B (Figure 3.219).**]
Specify dimension placement: [**Select C (Figure 3.219).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **51**

Select first object: [**Select A (Figure 3.219).**]
Select second object or place dimension: [**Select D (Figure 3.219).**]
Undo/Ordinate/Placement point/Enter dimension value: **102**

Select first object: [**Select A (Figure 3.219).**]
Select second object or place dimension: [**Select B (Figure 3.219).**]
Specify dimension placement: [**Select E (Figure 3.219).**]
Undo/Ordinate/Placement point/Enter dimension value: **1.5**

Select first object: [**Select A (Figure 3.219).**]
Select second object or place dimension: [**Select F (Figure 3.219).**]
Specify dimension placement: [**Select G (Figure 3.219).**]
Undo/Ordinate/Placement point/Enter dimension value: **3**
Select first object: [**Enter**]

Command: **8**

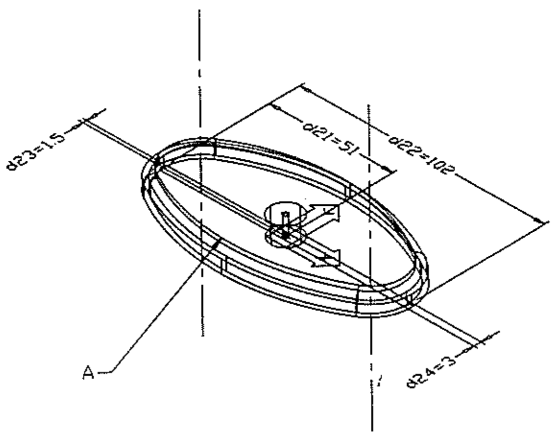


Figure 3.220 Rectangle fully constrained

Extrude the rectangle to meet the top face and join with the base solid. (See Figure 3.221.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Join** Termination: **To Face**
Size: Draft Angle: **1**
OK]

Select termination Face: [**Select A (Figure 3.220).**]
Next/<Accept>: [**Accept if the upper face is highlighted.**]

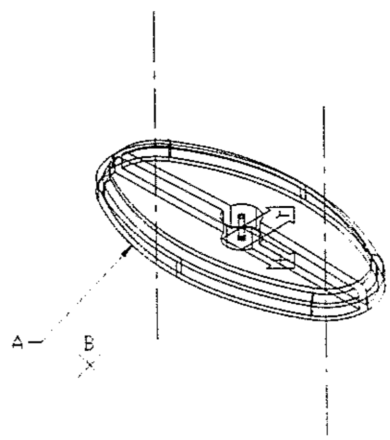


Figure 3.221 Web constructed

The last feature is a recess. To speed up model production, you will copy the sketch from the base feature. (See Figure 3.222.)

<Part> **<Sketch>** **<Copy Sketch>**

Command: **AMCOPYSKETCH**
Feature/<Sketch>: **F**
Select feature: [**Select A (Figure 3.221).**]
Next/<Accept>: [**Accept if the base feature is highlighted.**]
Sketch center: [**Select B (Figure 3.221).**]
Sketch center: [**Enter**]

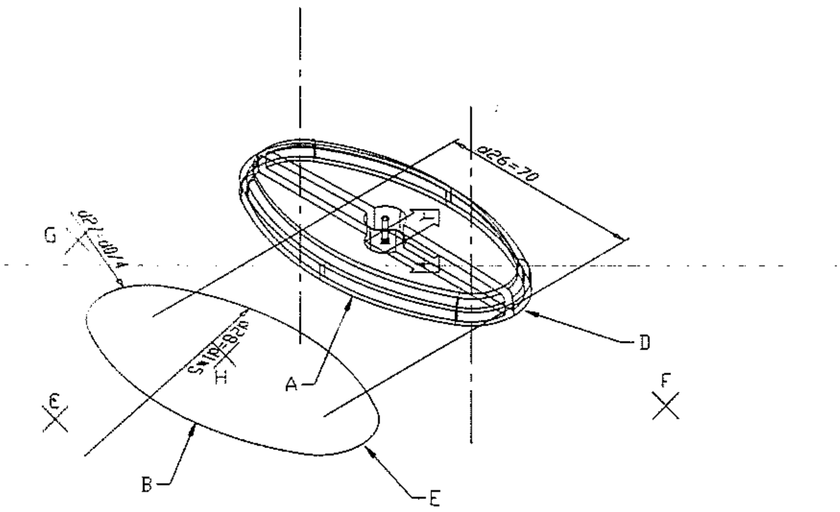


Figure 3.222 Sketch of base feature copied

Modify two dimensions and add two parametric dimensions. (See Figure 3.223.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select H (Figure 3.222).**]
New value for dimension <d1*5>: **d1*5-1**
Select dimension to change: [**Select G (Figure 3.222).**]
New value for dimension <d0/4>: **d0/4-1**
Select dimension to change: [**Enter**]

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.222).**]
Select second object or place dimension: [**Select B (Figure 3.222).**]
Specify dimension placement: [**Select C (Figure 3.222).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Select first object: [**Select D (Figure 3.222).**]
Select second object or place dimension: [**Select E (Figure 3.222).**]
Specify dimension placement: [**Select F (Figure 3.222).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Solved fully constrained sketch.
Select first object: [**Enter**]

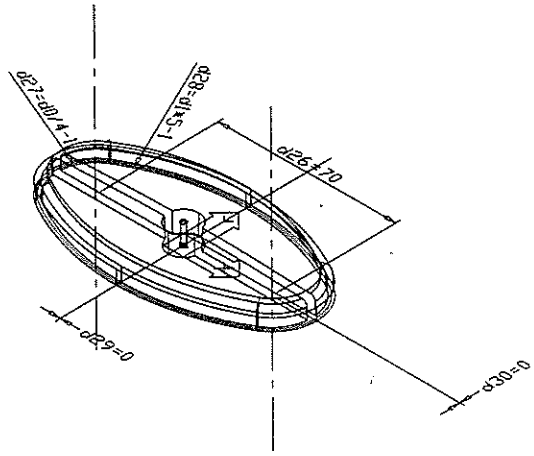


Figure 3.223 Copied sketch constrained and modified

Use the AMEXTRUDE command to extrude the last sketch and cut the base solid. (See Figure 3.224.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Cut** Termination: **Blind**
Size: Distance: **1** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: [**Accept if the arrow is pointing upward.**]

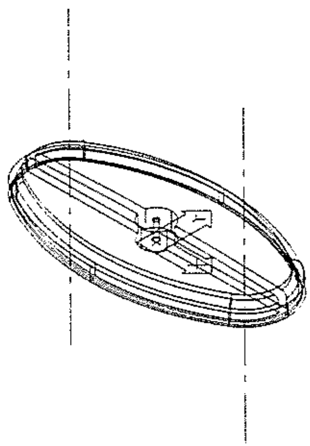


Figure 3.224 Recess cut

The upper casing is complete. Save your drawing.

<File> **<Save>**

File Name: **Ucase.dwg**

In making this model, you learned how to locate a free-form surface by using a work point, use a free-form surface to cut a solid, place a shell feature, extrude a profile to meet the free-form surface after shelling, copy a sketch from a sketched feature, and place fillet and hole features.

Lower Casing

To continue, you will modify the current drawing, Lcase.dwg, to make it the lower casing of the remote control. Save the drawing with another file name. Now you have two identical files, Ucase.dwg and Lcase.dwg.

<File> **<Save As...>**

File Name: **Lcase.dwg**

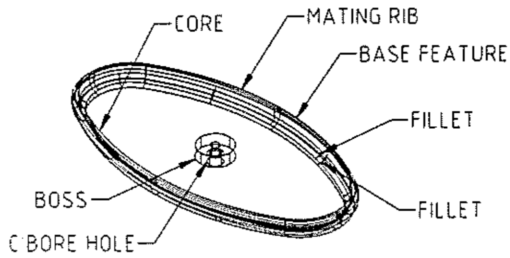


Figure 3.225 Features of the lower casing

Figure 3.225 shows the lower casing of the remote control assembly. It has seven features: four extrude solids, a hole, and two fillets. To learn how to replay and truncate, you will construct this model by modifying the upper casing. If you use the AMREPLAY command, the process of making the solid model can be replayed in a step-by-step fashion. While replaying, you can remove the remaining operations by truncating. (See Figure 3.226.) After truncation, all subsequent operations will be deleted from memory. Here you get the initial sketch.

<Part> **<Part>** **<Replay>**

Command: **AMREPLAY**
Display/Size/Truncate/eXit/<Next>: **T**

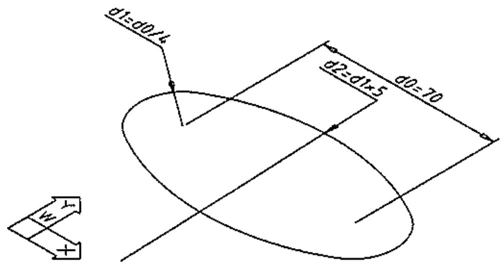


Figure 3.226 Replay truncated

Another way to remove features from a solid model is to use the Desktop Browser. To see how it can be done, undo the last AMREPLAY command to restore the original model. (See Figure 3.227.)

<Edit> **<Undo>**

Command: **U**

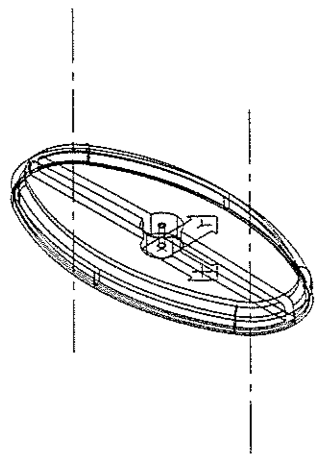


Figure 3.227 Replay undone

From the Desktop Browser, select the Part tab and then the Shell1 item (Figure 3.228). Then right-click. A pop-up menu is displayed. After that, select the Delete item to delete the highlighted object.

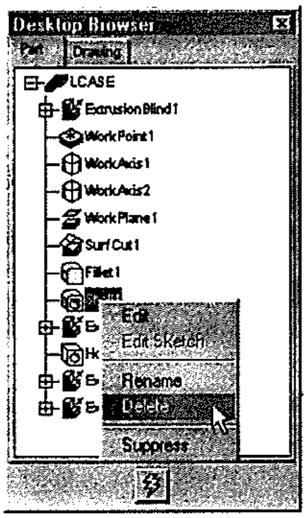


Figure 3.228 Delete features using the Desktop Browser

Command: **AMBROWSERACTION**
Highlighted features will be deleted. Continue? No/<Yes>: **[Enter]**

This way, the highlighted features are deleted from the graphic tree. Now continue to delete the other features until you have only the base feature. After that, use the AMREPLAY command to truncate the model to a sketch. (See Figure 3.226 again.)

<Part> **<Part>** **<Replay>**

Command: **AMREPLAY**
Display/Size/Truncate/eXit/<Next>: **T**

Now use the AMEXTRUDE command to construct an extrude solid feature. The height of extrusion is 6 units. The draft angle is $-1°$. The extrusion is toward the negative Z direction. (See Figure 3.229.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Base** Termination: **Blind**
Size Distance: **6** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing downward.]**

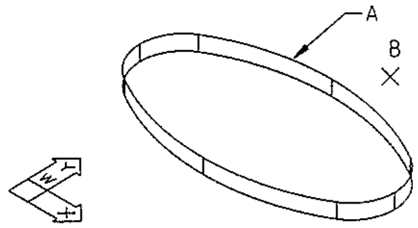


Figure 3.229 Extrude solid constructed

The second feature is an extrude feature. It needs a sketch similar to that for the base feature. Use the AMCOPYSKETCH command to copy a sketch from the sketch of the base feature. (See Figure 3.230.)

<Part> **<Sketch>** **<Copy Sketch>**

Command: **AMCOPYSKETCH**
Feature/<Sketch>: **F**
Select feature: [**Select A (Figure 3.229).**]
Sketch center: [**Select B (Figure 3.229).**]
Sketch center: [**Enter**]

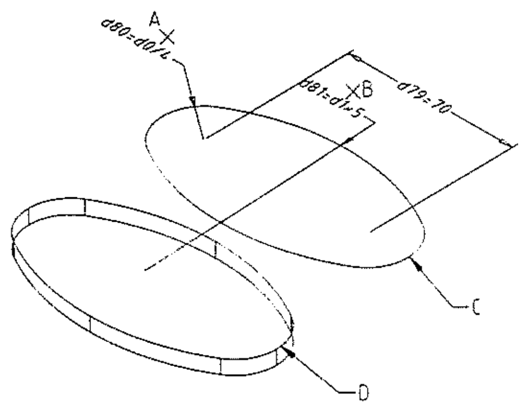


Figure 3.230 Sketch copied

Before extruding the copied sketch, use the AMADDCON command to add a concentric constraint to arcs C and D (Figure 3.230), and use the AMMODDIM command to modify two dimensions. Note that the new dimensions equal the old dimensions minus 1. (See Figure 3.231.)

<Part> **<Sketch>** **<Add Constraints>** <... >

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 3.230).**]
New value for dimension <d0/4>: **d0/4-1**
Select dimension to change: [**Select B (Figure 3.230).**]

New value for dimension <d1*5>: **d1*5-1**
Select dimension to change: **[Enter]**

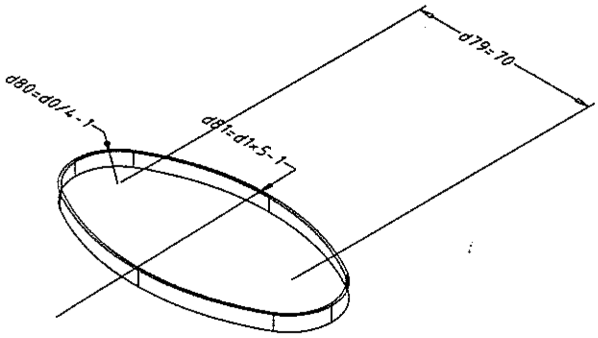


Figure 3.231 Copied sketch fully constrained

Now use the AMEXTRUDE command to extrude the profile and join it to the base solid. (See Figure 3.232.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Join** Termination: **Blind**
Size: Distance: **1** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

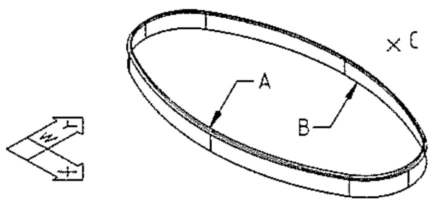


Figure 3.232 Second sketch extruded and joined

Use the AMSKPLN command to set the sketch plane to the top face of the solid. Then use the AMCOPYSKETCH command to copy the sketch from that of the base feature. (See Figure 3.233.)

<Part> **<Sketch>** **<Sketch Plane>**

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.232).]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **WORLDX**
Rotate/Z-flip/<Accept>: **[Enter]**

<Part> **<Sketch>** **<Copy Sketch>**

Command: **AMCOPYSKETCH**
Feature/<Sketch>: **F**
Select feature: [**Select B (Figure 3.232).**]
Sketch center: [**Select C (Figure 3.232).**]
Sketch center: [**Enter**]

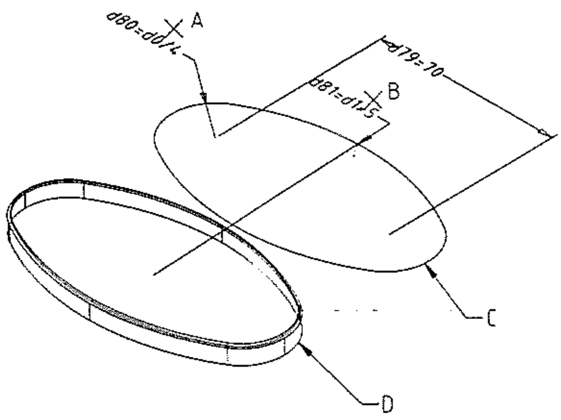


Figure 3.233 Sketch plane set to top face, sketch copied

Use the AMMODDIM command to change two dimensions. The new dimensions equal the original dimensions minus 2. Use the AMADDCON command to add a concentric constraint to arcs C and D (Figure 3.233). (See Figure 3.234.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 3.233).**]
New value for dimension <d0/4>: **d0/4-2**
Select dimension to change: [**Select B (Figure 3.233).**]
New value for dimension <d1*5>: **d1*5-2**
Select dimension to change: [**Enter**]

<Part> **<Sketch>** **<Add Constraints>** **<... >**

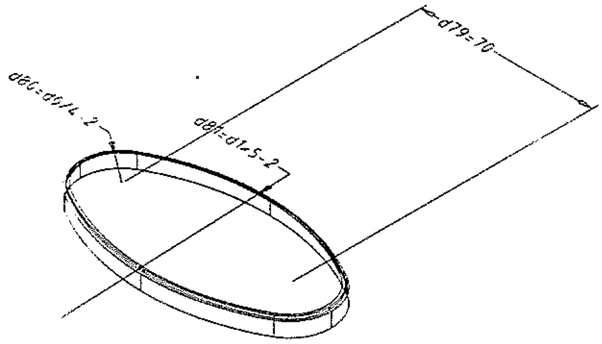


Figure 3.234 Copied sketch fully constrained

Now use the AMEXTRUDE command. The height of extrusion is 5 units. The draft angle is –1°. (See Figure 3.235.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Cut** Termination: **Blind**
Size: Distance: **5** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing downward.]**

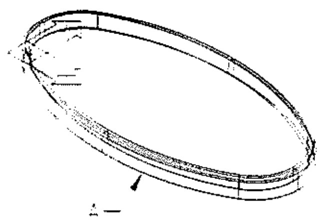


Figure 3.235 Core cut

Use the AMSKPLN command to set the sketch plane to the bottom face of the casing. Then construct a circle as shown in Figure 3.236.

<Part> **<Sketch>** **<Sketch Plane>**

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.235).]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **WORLDX**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that shown in Figure 3.236).]**

<Design> **<Circle>** **<Center, Radius>**

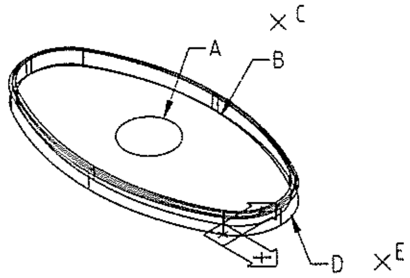


Figure 3.236 Sketch plane set, circle constructed

Use the AMPROFILE command to resolve the circle to a profile. Then use the AMPARDIM command to add three parametric dimensions. (See Figure 3.237.)

<Part> **<Sketch>** **<Profile>**

Command: **AMPROFILE**
Select objects for sketch:
Select objects: **[Select A (Figure 3.236).]**
Select objects: **[Enter]**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **[Select A (Figure 3.236).]**
Select second object or place dimension: **[Select B (Figure 3.236).]**
Specify dimension placement: **[Select C (Figure 3.236).]**
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: **[Select A (Figure 3.236).]**
Select second object or place dimension: **[Select D (Figure 3.236).]**
Specify dimension placement: **[Select E (Figure 3.236).]**
Undo/Ordinate/Placement point/Enter dimension value: **0**

Select first object: **[Select A (Figure 3.236).]**
Select second object or place dimension: **[Select C (Figure 3.236).]**
Undo/Ordinate/Placement point/Enter dimension value: **12**
Select first object: **[Enter]**

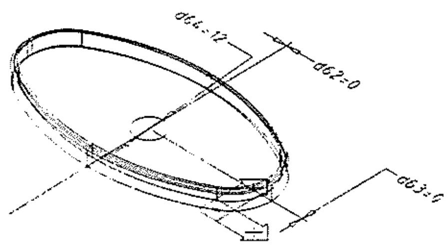


Figure 3.237 Circle resolved and fully constrained

Extrude the circle and join it to the base solid. (See Figure 3.238.)

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Join** Termination: **Blind**
Size: Distance: **6** Draft Angle: **-1**
OK]

Direction Flip/<Accept>: **[Accept if the arrow is pointing upward.]**

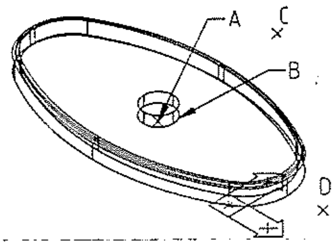


Figure 3.238 Boss constructed

The next feature is a counterbore hole. To place this hole, you need to use the AMWORKPT command to construct a work point. After that, add two dimensions. (See Figure 3.239.)

<Part> **<Work Features>** **<Work Point>**

Command: **AMWORKPT**
Location on sketch plane: [**Select A (Figure 3.238).**]

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: [**Select A (Figure 3.238).**]
Select second object or place dimension: [**Select B (Figure 3.238).**]
Specify dimension placement: [**Select C (Figure 3.238).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**

Select first object: [**Select A (Figure 3.238).**]
Select second object or place dimension: [**Select B (Figure 3.238).**]
Specify dimension placement: [**Select D (Figure 3.238).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0**
Select first object: [**Enter**]

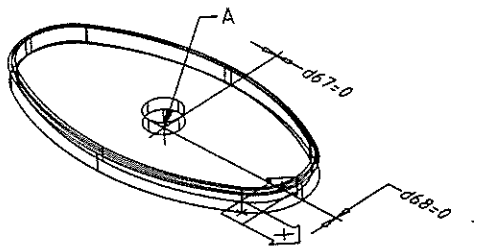


Figure 3.239 Work point constructed and constrained

Use the AMHOLE command to place a counterbore hole at the work point. The drill size is 3 units. The counterbore size is 5 units. The depth of counterbore is 3 units. (See Figure 3.240.)

<Part> <Placed Features> <Hole...>

Command: **AMHOLE**

[Operation: **C'Bore** Termination: **Through**
Placement **On Point**
Drill Size Dia: **3**
C'Bore/Sunk Size C'Depth: **3** C'Dia: **5**]
Apply<

Select work point: **[Select A (Figure 3.239).]**
Select work point: **[Enter]**

[**Exit**]

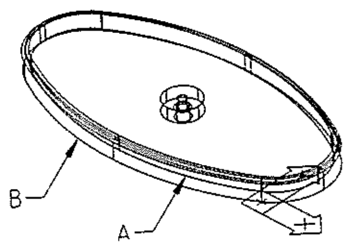


Figure 3.240 Counterbore hole placed

Use the AMFILLET command to place fillet features inside and outside the lower edges. The inside A fillet radius (Figure 3.240) is 2 units and the outside B fillet radius (Figure 3.240) is 4 units. (See Figure 3.241.)

<Part> <Placed Features> <Fillet...>

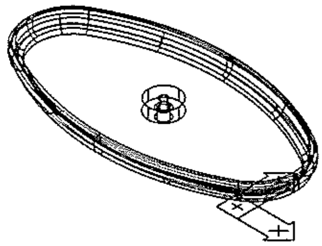


Figure 3.241 Fillet edges placed

The lower casing is complete. Save your drawing.

<File> <Save>

In making this model, you constructed a number of sketches on sketch planes selected from existing faces of the solid. In addition, you learned how to replay and truncate the modeling process. Now you have two part drawings for this project. You will assemble them together in the next chapter.

Shock Absorber Project (Continued)

Figure 3.242 shows the rendered image of the cylinder of a shock absorber. In Section 3.5, you constructed a revolve solid. Now you will cut a helical screw thread. There are two ways to construct this feature: use a sweep surface to cut a solid (see Chapter 2) and sweep a profile along a 3D helical path. Here you will sweep a profile along a specified 3D helical path to cut the solid.

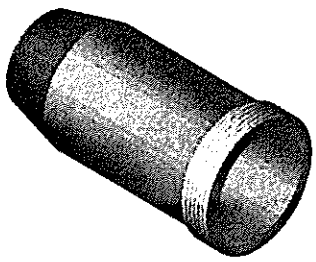


Figure 3.242 Rendered image of the cylinder of the shock absorber

Open the drawing that you saved in Section 3.5. Use the AMWORKPLN command to construct a work plane that is offset from the end face of the solid. (See Figure 3.243.)

<File> <Open...>

File name: **Shock.dwg**

<Part> <Work Features> <Work Plane...>

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset** Offset **4**
Create Sketch Plane
OK]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 3.243).]**
Flip/<Accept>: **[Accept if the arrow is pointing toward B (Figure 3.243).]**
Z-flip/Rotate/<Select edge to align X axis>: **[Enter if the UCS orientation is the same as that shown in Figure 3.243.]**

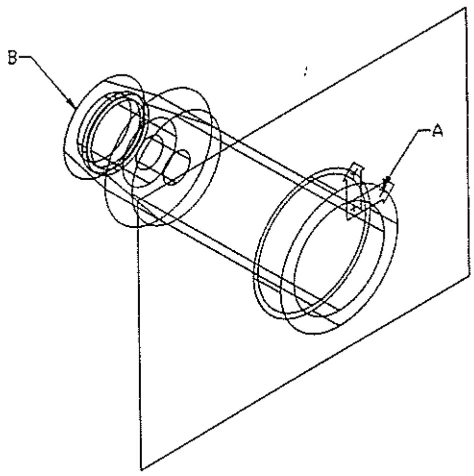


Figure 3.243 Work plane constructed

Set the current layer to Sketch. Then use the AM3DPATH command to specify a 3D helical path. (See Figure 3.244.)

[Mechanical Main] **[Layer Control]**

Current layer: **Sketch**

<Part> **<Sketch>** **<3D Path (Helix)...>**

Command: **AM3DPATH**

Select work axis, circular edge, or circular face for helical center: **[Select A (Figure 3.243).]**

[Type **Pitch and Revolution**
Revolution: **10** Pitch: **0.5**
Shape **Circle**
Diameter: **13** Taper Angle: **0**
OK]

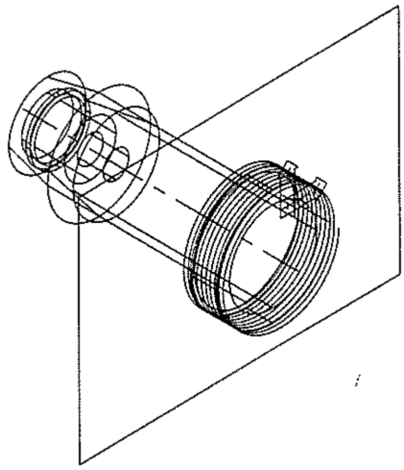


Figure 3.244 Helical path constructed

Use the AMWORKPLN command to construct a work plane at the start point of the 3D helical path. (See Figure 3.245.)

<Part> <Work Features> <Work Plane...>

Command: **AMWORKPLN**

[1st Modifier **Normal to Start**
Create Sketch Plane
OK]

Z-flip/Rotate/<Select edge to align X axis>: **[Enter to accept if the UCS orientation is the same as that shown in Figure 3.245.]**

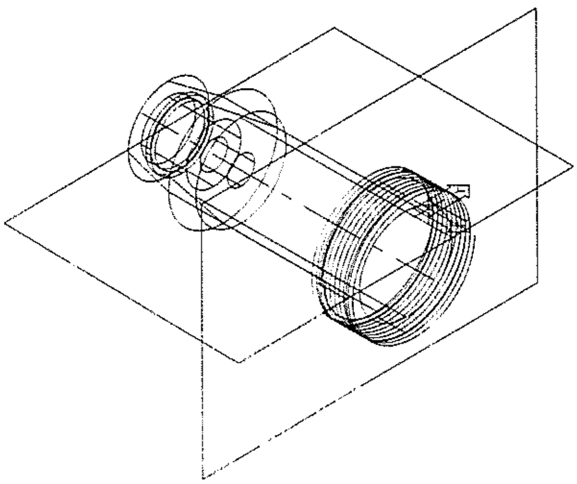


Figure 3.245 Work plane constructed normal to the start point of the helical path

Now use the shortcut key [9] to set the display to the plan view. Then use the AMVISIBLE command to hide the work planes. After that, use the ZOOM command to zoom the display to show the start point of the helical path as shown in Figure 3.246.

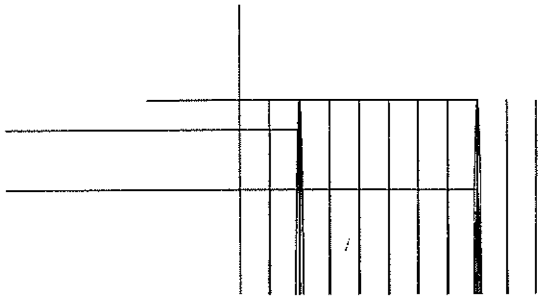


Figure 3.246 Display set to plan view and zoomed to the start point of the helical path

Construct a sketch as shown in Figure 3.247. Then resolve it to a profile. In Figure 3.247, A is the start point of the 3D helical path.

<Design> **<Polyline>**

<Part> **<Sketch>** **<Single Profile>**

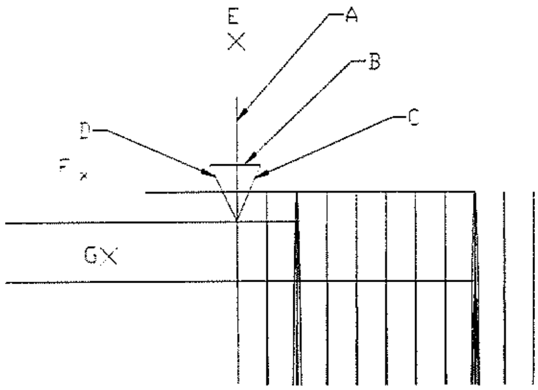


Figure 3.247 Polyline constructed and resolved

Add a horizontal geometric constraint to line B (Figure 3.247) and add the following parametric dimensions. (See Figure 3.248.)

<Part> **<Sketch>** **<Add Constraints>** **<Horizontal>**

<Part> **<Add Dimension>**

Command: **AMPARDIM**
Select first object: **[Select C (Figure 3.247).]**
Select second object or place dimension: **[Select B (Figure 3.247).]**
Specify dimension placement: **[Select G (Figure 3.247).]**
Undo/Placement point/Enter dimension value: **N**

Undo/Placement point/Enter dimension value: **60**

Select first object: [**Select C (Figure 3.247).**]
Select second object or place dimension: [**Select D (Figure 3.247).**]
Specify dimension placement: [**Select E (Figure 3.247).**]
Undo/Placement point/Enter dimension value: **N**
Undo/Placement point/Enter dimension value: **60**

Select first object: [**Select C (Figure 3.247).**]
Select second object or place dimension: [**Select A (Figure 3.247).**]
Specify dimension placement: [**Select E (Figure 3.247).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0.24**

Select first object: [**Select B (Figure 3.247).**]
Select second object or place dimension: [**Select E (Figure 3.247).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0.48**

Select first object: [**Select A (Figure 3.247).**]
Select second object or place dimension: [**Select B (Figure 3.247).**]
Specify dimension placement: [**Select F (Figure 3.247).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/Diameter/pLace/Enter dimension value: **0.05**
Select first object: [**Enter**]

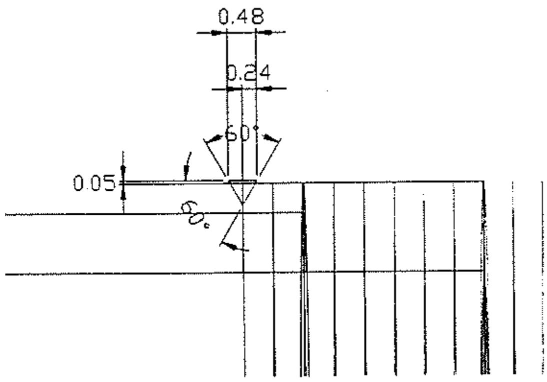


Figure 3.248 Profile fully constrained

Now use the AMSWEEP command to sweep the profile along the helical path to form a sweep solid and cut the sweep solid from the base feature. (See Figure 3.249.)

<Part> **<Sketched Features>** **<Sweep...>**

Command: **AMSWEEP**

[Operation **Cut** Termination **Path Only**
OK]

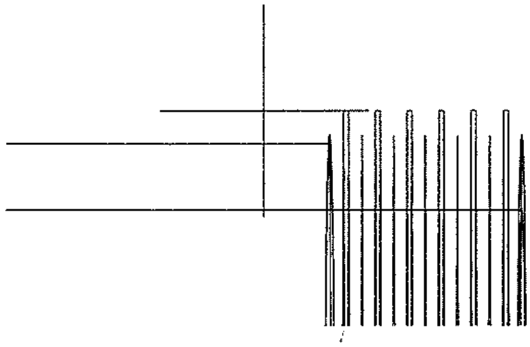


Figure 3.249 Helical thread cut on the base solid

The model is complete. Save your drawing.

<File> **<Save As...>**

File name: **Shockt.dwg**

In making this model, you constructed a 3D helical path and a profile, then swept the profile along the path to form a helical sweep solid to cut a screw thread. There are two ways to make a helical thread or groove. You can construct a helical sweep surface and use the surface to cut the solid, or construct a 3D sweep solid feature.

Helical Spring Project (Continued)

Figure 3.250 shows the rendered image of the completed helical spring. In Section 3.6, you constructed the base solid feature of a helical spring. Now you will cut away the unwanted part by constructing a cylindrical solid to intersect with the base solid feature.

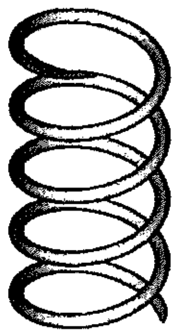


Figure 3.250 Rendered image of the helical spring

Open the drawing that you saved in Section 3.6.

<File> **<Open...>**

File name: **Spring.dwg**

Before intersecting, edit the diameter of the circle that was swept along the 3D helical path. As shown in Figure 3.251, select the sweep item in the browser and press the right mouse button to bring up the pop-up menu. Then select the Edit Sketch item from the menu. The original sketch for the solid appears. (See Figure 3.252.)

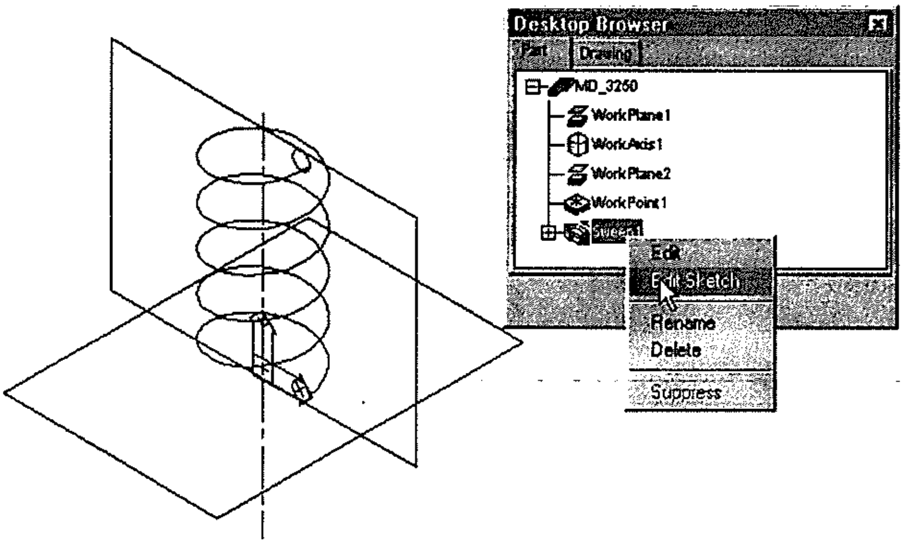


Figure 3.251 Edit Sketch in the browser

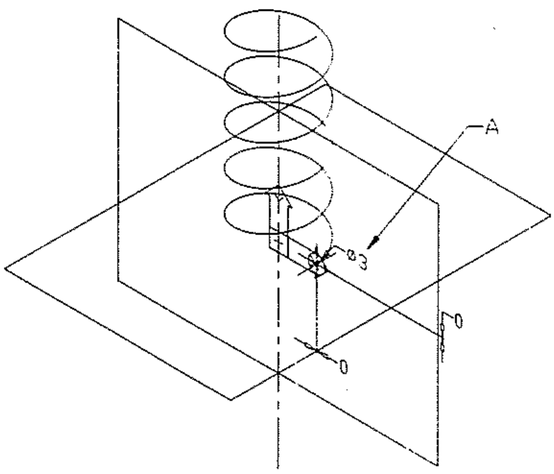


Figure 3.252 Sketch of the profile

Use the AMMODDIM command to change the dimension of the circle. Then update the solid. (See Figure 3.253.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 3.252).**]

New value for dimension: **1.5**
Select dimension to change: **[Enter]**

<Part> **<Update>**

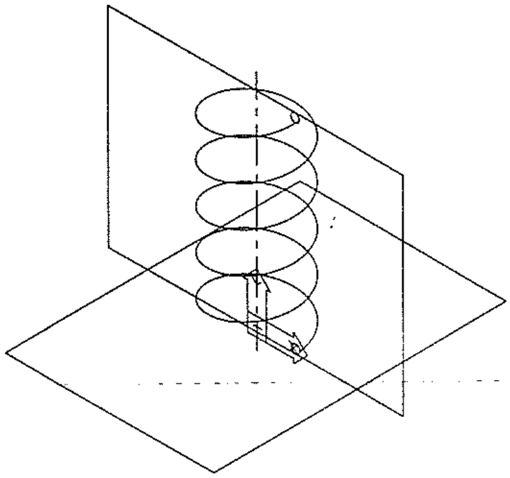


Figure 3.253 Dimension changed, solid updated

Set the display to the plan view of the current sketch plane. Then set the current layer to Sketch. After that, construct a rectangle as shown in Figure 3.254.

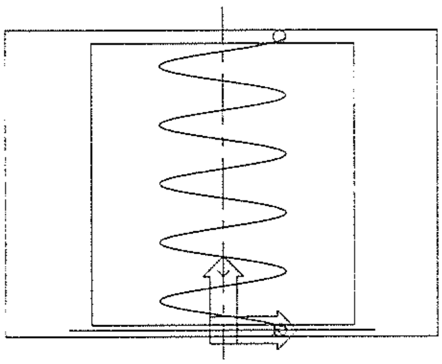


Figure 3.254 Display set, rectangle constructed

Resolve the rectangle into a profile. Then add parametric dimensions as shown in Figure 3.255.

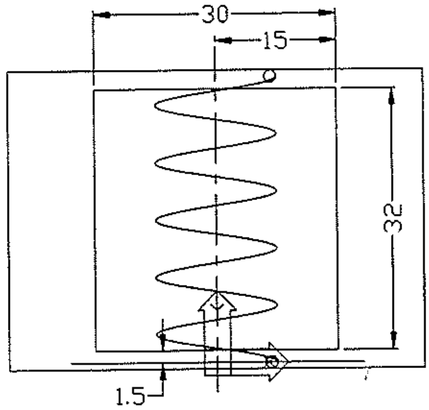


Figure 3.255 Sketch resolved and constrained

Set the display to an isometric view. Then use the AMEXTRUDE command to extrude the profile and intersect it with the base solid feature. (See Figure 3.256.)

Command: **8**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation: **Intersect** Termination: **Mid Plane**
Size: Distance: **30** Draft Angle: **0**
OK]

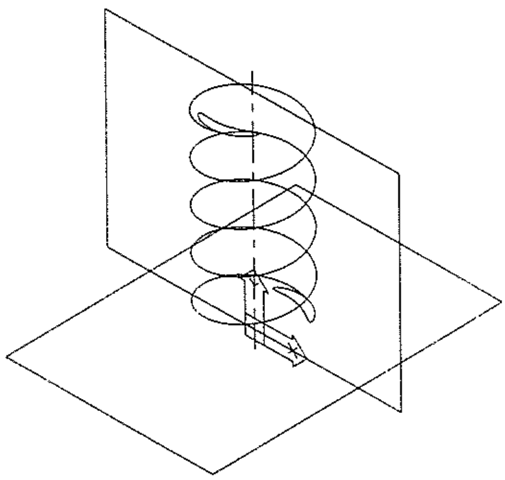


Figure 3.256 Rectangle extruded and intersected

The model is complete. Save your drawing.

<File> **<Save>**

In making this model, you modified the wire diameter and constructed a cylinder to intersect with the base solid. This way, the spring wire is modified and the end faces are cut away.

Forged Link Project (Continued)

Figure 3.257 shows the rendered image of the completed forged link. In Section 3.8, you constructed the base solid feature and split the vertical faces of the extrude solid. The purpose of splitting a face is to enable you to place face drafts in two directions. Now you will modify the base solid and apply face drafts along the split line. A sketched solid feature is derived from a sketch. After making a sketched feature, you can modify it not only by changing the geometric constraints and parametric dimensions; you can also append additional objects to the sketch.

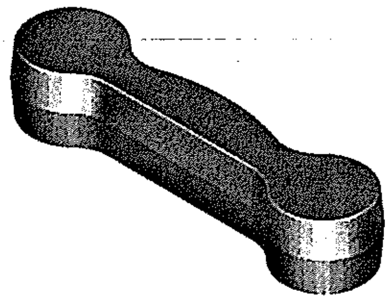


Figure 3.257 Rendered image of the forged link

Open the drawing that you saved in Section 3.8.

<File> **<Open...>**

File name: **Forge.dwg**

Use the AMSKPLN command to set the sketch plane.

<Part> **<Sketch>** **<New Sketch Plane>**

Command: **AMSKPLN**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **X**
Z-flip/Rotate/<Select edge to align X axis>: **[Enter to accept if the UCS orientation is the same as that shown in Figure 3.258).]**

From the graphic tree in the browser, select the ExtrusionMidplane1 item. (See Figure 3.258.) Press the right mouse button to bring up the pop-up menu. After that, select the Edit Sketch item to activate the AMEDITFEAT command. (See Figure 3.259.)

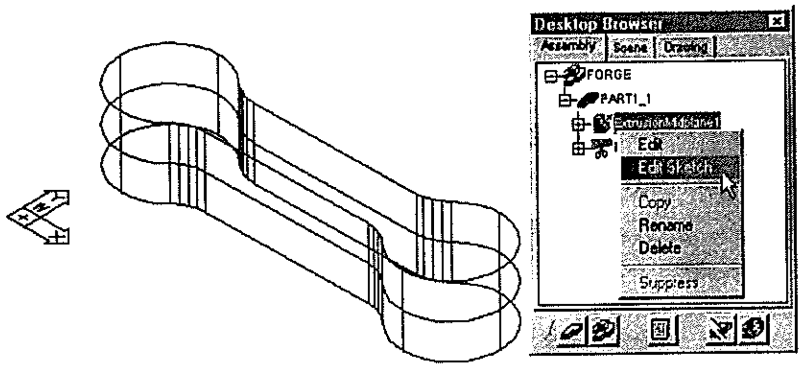


Figure 3.258 Edit Sketch using the Desktop Browser

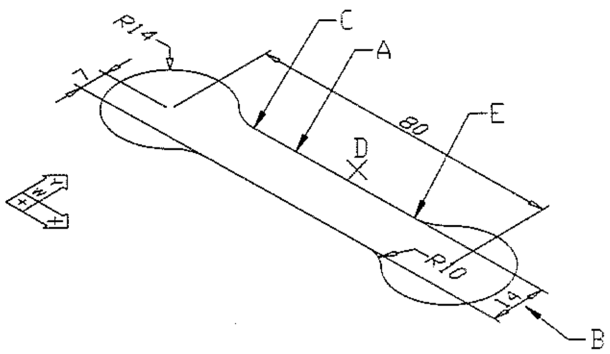


Figure 3.259 Sketch of the profile

Use the ERASE command to erase line A (Figure 3.259) and use the ARC command to construct an arc.

<Modify> **<Erase>**

Command: **ERASE**
Select objects: [**Select A and B (Figure 3.259).**]
Select objects: [**Enter**]

<Design> **<Arc>** **<3 Points>**

Command: **ARC**
Center/<Start point>: **END** of [**Select C (Figure 2.259).**]
Center/End/<Second point>: [**Select D (Figure 2.259).**]
End point: **END** of [**Select E (Figure 2.259).**]

Now select Profile1 in the browser and press the right mouse button to bring up the pop-up menu. (See Figure 3.260.)

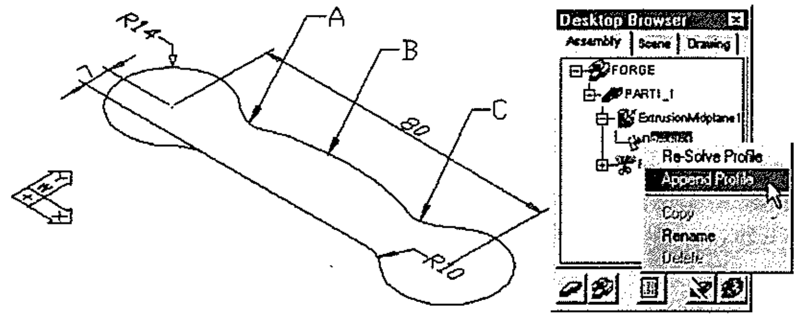


Figure 3.260 Line erased, arc constructed

Select Append Profile (Figure 3.260) to use the AMRSOLVESK command to append the arc to the sketch and resolve the sketch again.

Command: **AMRSOLVESK**
Select geometry to append to sketch: **[Select B (Figure 3.260).]**
Select geometry to append to sketch: **[Enter]**
Redefining existing sketch.

Add geometric constraints in accordance with the following table. Then add parametric dimensions as shown in Figure 3.261.

Arcs A and B (Figure 3.260)	tangent
Arcs B and C (Figure 3.260)	tangent
Arcs A and C (Figure 3.260)	centers have same Y values

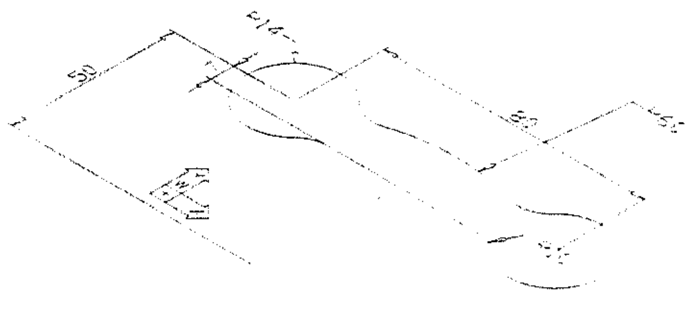


Figure 3.261 Resolved sketch properly constrained

Use the AMUPDATE command to update the changes. Then use the AMWORKPLN command to construct a work plane. (See Figure 3.262.)

<Part> <Update>

<Part> <Work Features> <Work Plane...>

Command: AMWORKPLN

[1st Modifier **World XY**
Create Sketch Plane

OK]

Z-flip/Rotate/<Select edge to align X axis>: **[Enter]**

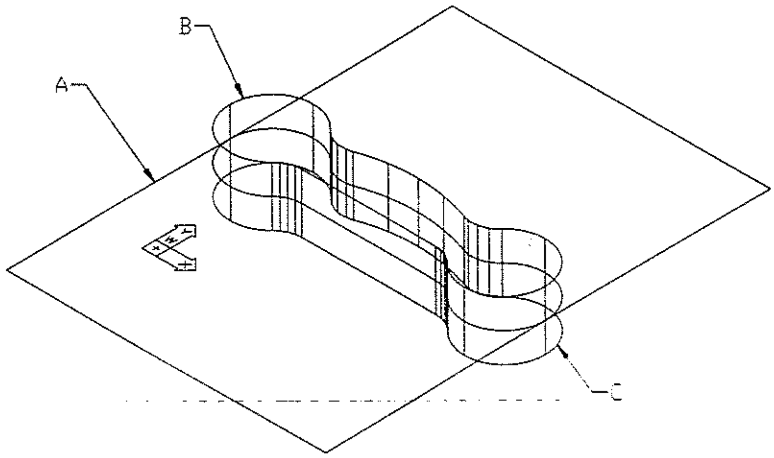


Figure 3.262 Solid updated, work plane constructed

Now use the AMFACEDRAFT command to place face drafts on the solid. (See Figure 3.263, the Face Draft dialog box.) Set Type to From Plane and Angle to 5°. Then select the [Draft Plane] button. After that, select the draft plane A (Figure 3.262). On returning to the Face Draft dialog box, select the [Add] button. Then select the faces of draft B (Figure 3.262). When the Face Draft dialog box reappears, select the [OK] button. A face draft is placed.

<Part> **<Placed Features>** **<Face Draft...>**

Command: **AMFACEDRAFT**

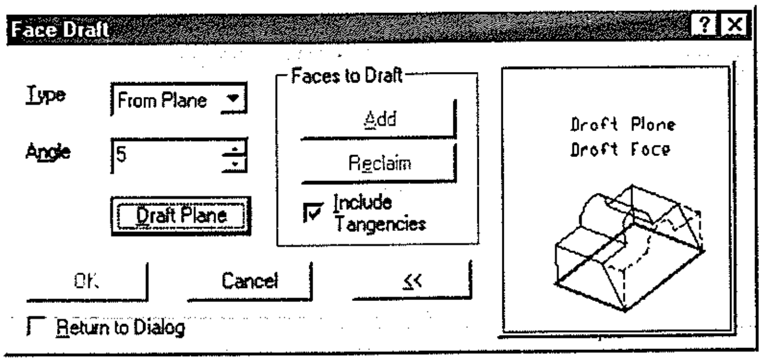


Figure 3.263 Face Draft dialog box

Select draft plane (planar face or work plane): **[Select A (Figure 3.262).]**
Flip/<Accept>: **[Accept if the arrowhead is pointing upward.]**
Select faces to draft (ruled faces only): **[Select B (Figure 3.262).]**
Select faces to draft (ruled faces only): **[Enter]**

Now repeat the AMFACEDRAFT command to place a face draft feature on face C (Figure 3.262). This time, the arrow should be pointing downward. (See Figure 3.264.)

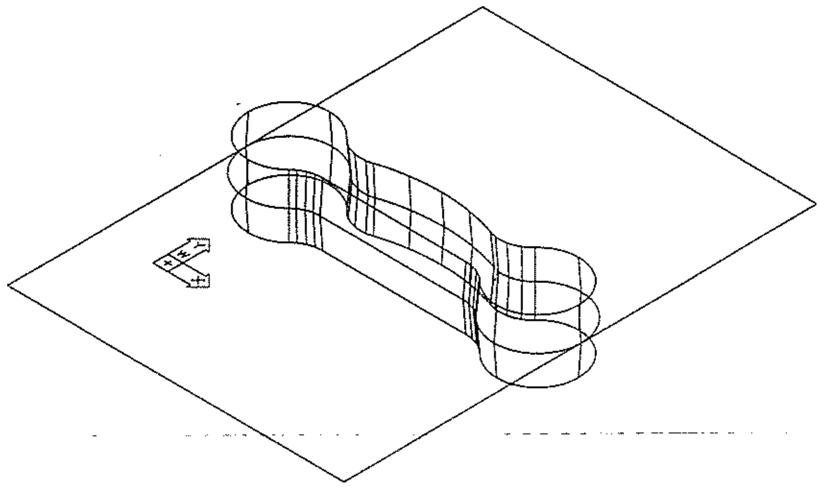


Figure 3.264 Face draft features placed

Features can be suppressed. This way, you can appreciate how the solid model will appear if you remove a particular feature without actually deleting the feature from the solid. In the Desktop Browser, select the face draft features one by one. Then press the right mouse button and select the Suppress item. (See Figure 3.265.)

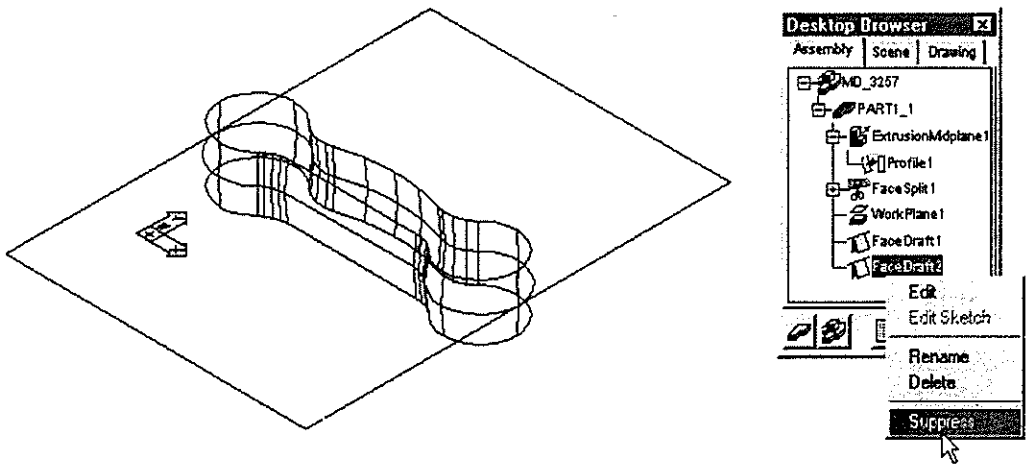


Figure 3.265 Suppressing a feature by using the Desktop Browser

The face draft is suppressed; your display should resemble Figure 3.262. Now unsuppress the suppressed features. The face draft appears again. (See Figure 3.264.) The model is complete. Save your drawing.

<File> **<Save As...>**

File name: **Forge1.dwg**

In making this model, you learned how to modify the model by deleting part of a sketch and appending the sketch, and applied face drafts to faces in two directions. You also observed the effect of suppressing a feature of a solid.

It must be emphasized that splitting is not a prerequisite for making face drafts. You can place face drafts on faces that are not split. However, splitting a face enables you to place face drafts on the face in two directions and at two different draft angles.

Mold Base Project

Figure 3.266 shows a rendered image of a mold base. You will construct an extrude solid and place hole features and array features on it. While making this model, you will copy an instance and copy a solid definition. On completion, you will discover the differences between copying an instance and copying a definition.

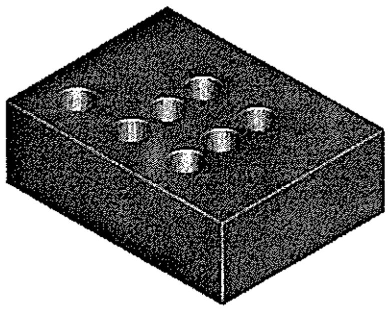


Figure 3.266 Rendered image of a mold base

Start a new multi-part drawing. Use Solid.dwt as the template. As shown in Figure 3.267, construct a rectangle, resolve it to a profile, and add parametric dimensions.

<File> **<New...>**

<Design> **<Rectangle>**

<Part> **<Sketch>** **<Profile>**

<Part> **<Add Dimension>**

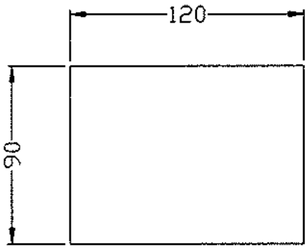


Figure 3.267 Rectangle constructed and resolved, parametric dimensions added

Set the display to an isometric view and set thé current layer to Solid. Then use the AMEXRUDE command to extrude the profile. The height of extrusion is 40 and the draft angle is 0°.

Command: 8

[Mechanical Main] **[Layer Control]**

Current layer: **Solid**

<Part> **<Sketched Features>** **<Extrude...>**

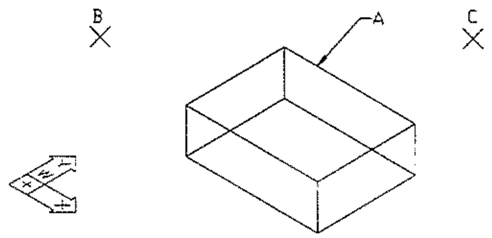


Figure 3.268 Display set, profile extruded

Make two copies of this solid part: Copy the solid as a new definition and as a new instance. (See Figure 3.269.)

<Part> **<Part>** **<Instance>**

Select object to Instance (or ?) <PART1>: **[Select A (Figure 3.268).]**
Select insertion point: **[Select B (Figure 3.268).]**
Select insertion point: **[Enter]**

<Part> **<Part>** **<Copy Definition>**

Cataloged part or subassembly to copy (? for list): **[Select A (Figure 3.268).]**
New part or subassembly name : **NEWPART**
Select insertion point: **[Select C (Figure 3.268).]**
Select insertion point: **[Enter]**

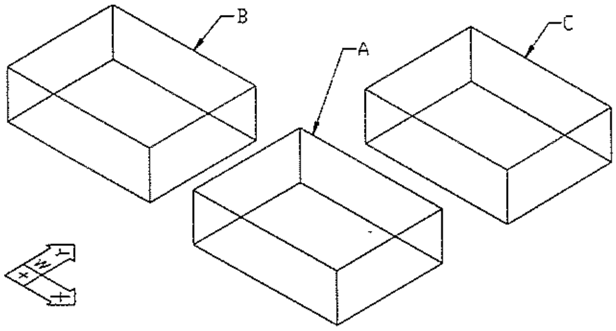


Figure 3.269 Solid copied as an instance and as a new definition

Now you have three solids. Solid B (Figure 3.269) is an instance of solid A (Figure 3.269), and solid C (Figure 3.269) is a new definition.

To see the difference between an instance and a new definition, you will add a feature to solid A (Figure 3.269).

Use the AMACTIVATE command to set solid A (Figure 3.269) as the active solid part. Then set the sketch plane to the top face as shown in Figure 3.270.

<Part> **<Part>** **<Make Active>**

Command: **AMACTIVATE**
Select part to activate (or ?): **[Select A (Figure 3.269).]**

<Part> **<Sketch>** **<Sketch Plane>**

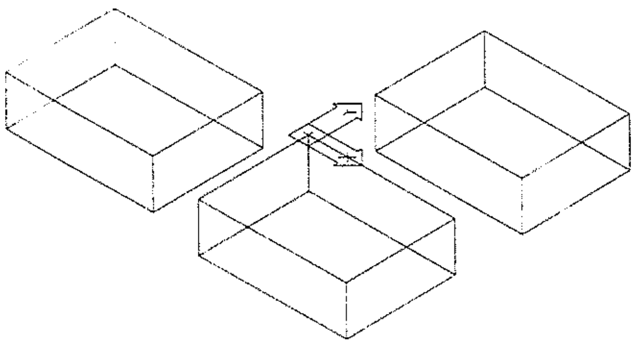


Figure 3.270 Sketch plane set

Set the current layer to Sketch. Then construct a circle and resolve it to a profile. After that, add parametric dimensions as shown in Figure 3.271.

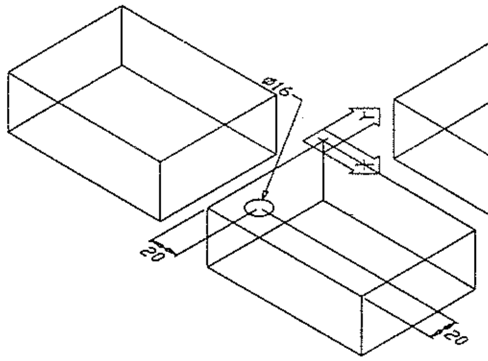


Figure 3.271 Circle constructed and resolved, parametric dimensions added

Use the AMEXTRUDE command to extrude the profile a distance of 20 units to cut the base solid. (See Figure 3.272.)

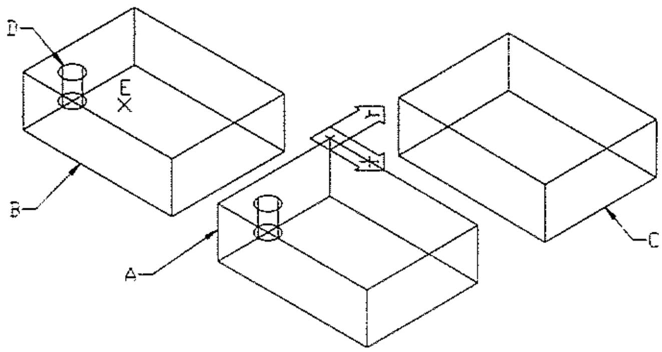


Figure 3.272 Profile extruded and cut

Because solid B is an instance of solid A, operations applied to A will also be executed on B (and vice versa). However, solid C is a new definition; it remains unchanged.

Now set solid B (Figure 3.272) as the active solid. Then use the AMCOPYFEAT command to make a copy of the last solid feature. (See Figure 3.273.)

<Part> **<Part>** **<Make Active>**

Command: **AMACTIVATE**
Select part to activate (or ?): **[Select B (Figure 3.272).]**

<Part> **<Placed Features>** **<Copy>**

Command: **AMCOPYFEAT**
Select feature: **[Select D (Figure 2.272).]**
Next/<Accept>: **[Accept if the cylindrical extrude feature is highlighted.]**
Parameters/<Select location>: **[Select E (Figure 2.272).]**
Parameters/Rotate/Flip/<Select location>: **[Enter]**

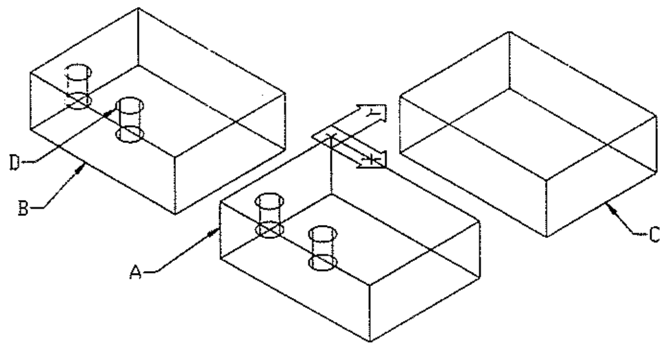


Figure 3.273 Solid feature copied

Once again, solid C (Figure 3.273) remains unchanged; solid A and solid B change concurrently.

After copying, the copied extrude feature needs two parametric dimensions to properly locate its position. To add the dimensions, use the AMEDITFEAT command to edit the sketch. (See Figure 3.274.)

<Part> **<Edit Feature>**

Command: **AMEDITFEAT**
Independent array instance/Sketch/surfCut/Toolbody/<select Feature>: **S**
Select sketched feature: [**Select D (Figure 3.273).**]
Next/<Accept>: [**Accept if the extrude solid feature is highlighted.**]

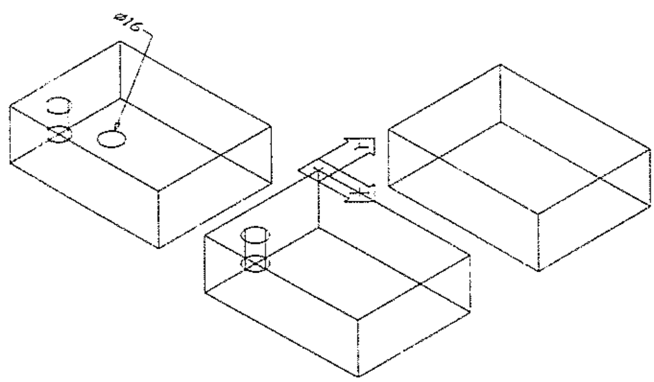


Figure 3.274 Sketch of the copied feature

Add two parametric dimensions as shown in Figure 3.275. Then use the AMUPDATE command to update the change. (See Figure 3.276.)

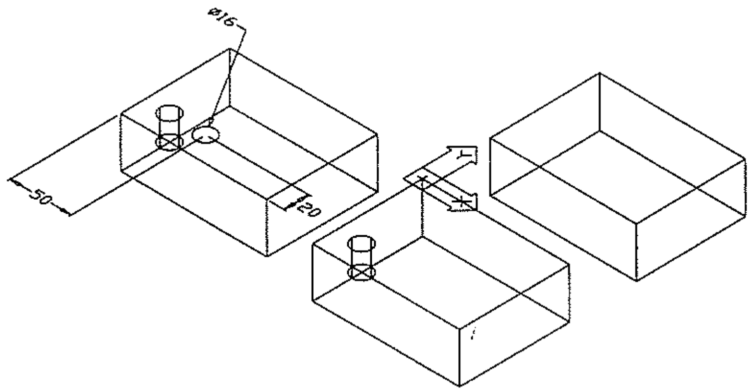


Figure 3.275 Parametric dimensions added

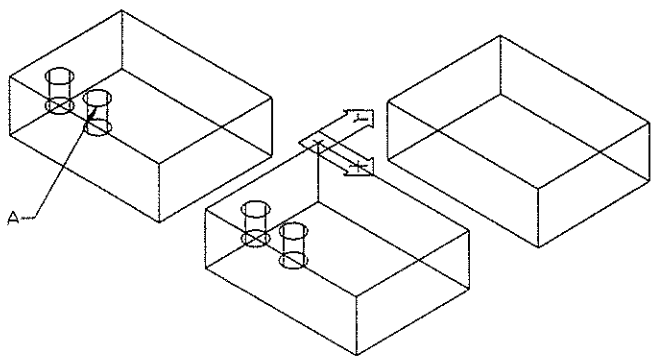


Figure 3.276 Solids updated

To further examine the relationship between the two instances, use the AMARRAY command to make a rectangular array of the solid feature A (Figure 3.276). (See Figure 3.277, the Array dialog box.) You can construct two kinds of arrays: Rectangular and Polar. Select Rectangular with two columns (spacing 30 units) and three rows (spacing 20 units). (See Figure 3.278.)

<Part> **<Placed Features>** **<Array...>**

Command: **AMARRAY**
Select feature: **[Select A (Figure 3.276).]**
Next/<Accept>: **[Accept if the copied solid feature is highlighted.]**

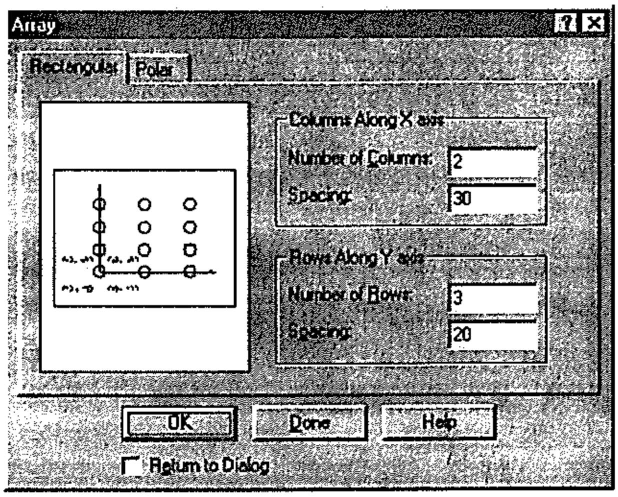


Figure 3.277 Array dialog box

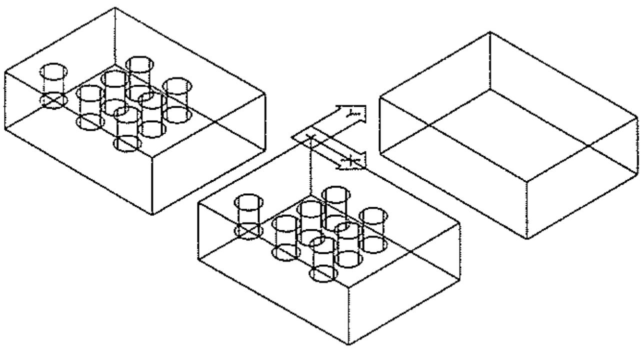


Figure 3.278 Solid feature arrayed

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Mold.dwg**

In making these solids, you copied and instanced a solid model and arrayed and copied features on a solid model. By now, you should be able to differentiate among the three kinds of copying: copying features of a solid model, copying the instance of a solid model, and copying the definition of a solid model. Superficially, a copied instance and a copied definition appear alike. However, they are not the same.

Summary

Before constructing a complex solid, you should analyze the model to decompose it into a number of simple solid features. Then determine how these features are constructed, either by sketching or by placing. Sketched solid features are the extrude solid, the revolve solid, the sweep solid, the 3D helical sweep solid, and the loft solid. Placed solid features are the hole, the fillet, the chamfer, the face draft, the shell, the surface cut, the array, and the copy. While thinking about the solid features, you should consider the work features needed to maintain a proper parametric relationship among the solid features (sketched and placed). Work features are the work plane, the work point, and the work axis.

3.11 Design Variables and Table-Driven Parts

To better control the parametric dimensions of the solids in a single file and across a set of files, you can set up a design variable file or construct an Excel spreadsheet to maintain a set of design variables. By using an Excel spreadsheet, you can construct table-driven parts. The following projects illustrate the use of design variables in solid modeling.

Four-Bar Linkage Project

The four-bar linkage has four bars of different lengths and four identical pivot pins. Here you will construct four linkages and a pivot pin. (See Figure 3.279.) In the next chapter, you will copy three more instances of the pivot pin and then put the parts together to form an assembly.

In this chapter, you will construct a set of design variables and put them in a parameter file, use the design variables to construct a solid part, copy three more definitions from the solid, and change the design variable name.

On completion, you will have four linkage bars, each with a different length that is governed by a parameter file, together with a pivot pin.

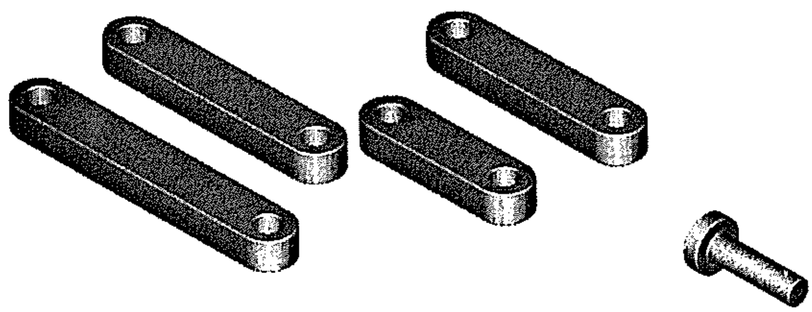


Figure 3.279 Rendered image of the components of a four-bar linkage

Start a new multi-part drawing. Use Solid.dwt as the template.

<File> <New...>

As shown in Figure 3.280, construct a polyline, resolve it, and apply the following geometric constraints.

Lines A and C (Figure 3.280)	horizontal
Line A and arc B (Figure 3.280)	tangent
Arc B and line C (Figure 3.280)	tangent
Line C and arc D (Figure 3.280)	tangent
Arc D and line A (Figure 3.280)	tangent

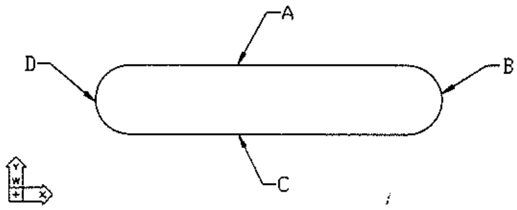


Figure 3.280 Polyline resolved, geometric constraints applied

Use the AMVARS command to build a set of design variables. (See Figure 3.281.) There are two tabs: Active Part and Global. The Active Part tab concerns the variables of the active solid part, and the Global tab concerns the variables for a set of solid parts in the drawing file.

Select the Global tab. Then select the [New...] button. The New Part Variable dialog box appears. (See Figure 3.282.) Add a new variable called Length_A with an equation of 80 and the comment Linkage bar A. Then select the [OK] button.

After that, add six more variables in accordance with the following table.

Name	Equation	Comment
Length_B	100	Linkage bar B
Length_C	50	Linkage bar C
Length_D	70	Linkage bar D
Rad	10	Corner radius
Hole	10	Diameter of the hole
Thk	12	Thickness of the linkage bar

After setting the seven design variables (Figure 3.283), select the [OK] button to exit.

<Part> **<Design Variables...>**

Command: **AMVARS**

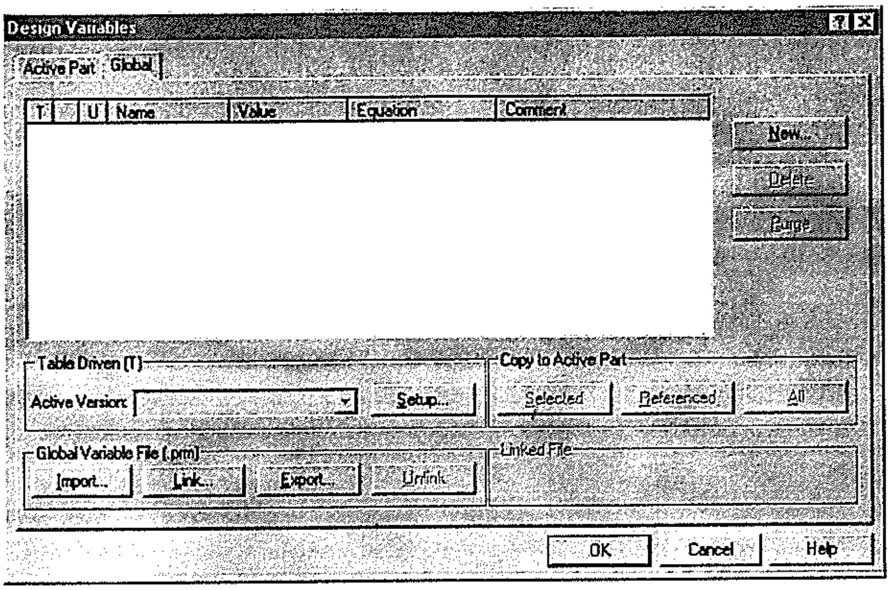


Figure 3.281 Design Variables dialog box

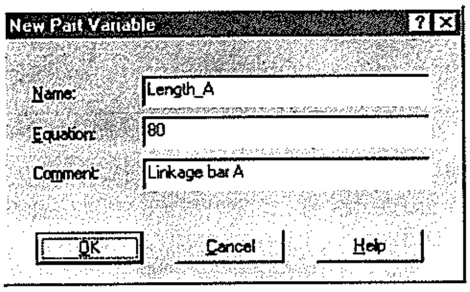


Figure 3.282 Adding a new variable

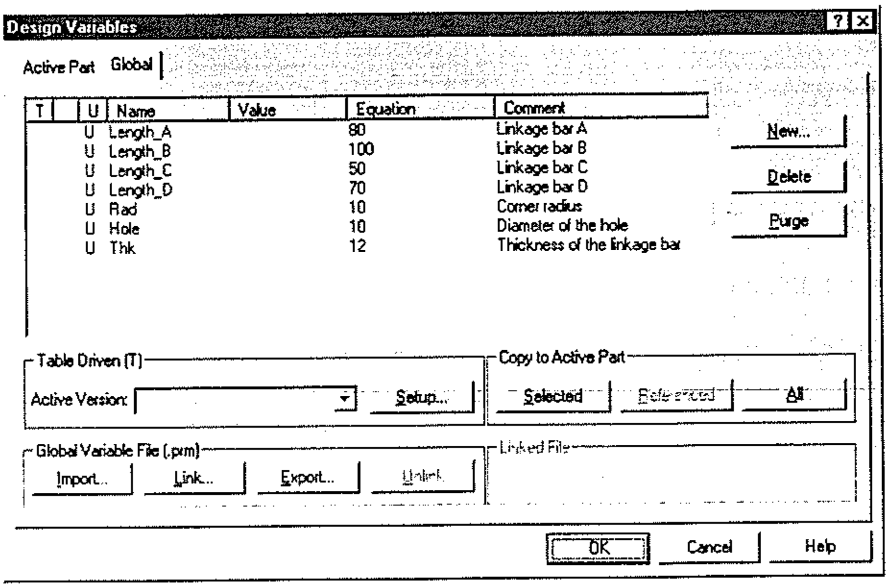


Figure 3.283 Design variables set

Now set the dimension display to equation. Then add parametric dimensions as shown in Figure 3.284.

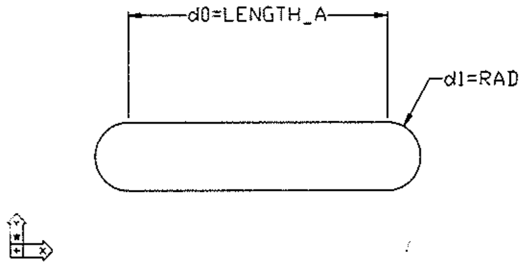


Figure 3.284 Dimension display set to equation, parametric dimensions added

Set the display to an isometric view. Then set the current layer to Solid. After that, use the AMEXTRUDE command to extrude the profile a distance of Thk. Thk is a design variable. (See Figure 3.285.)

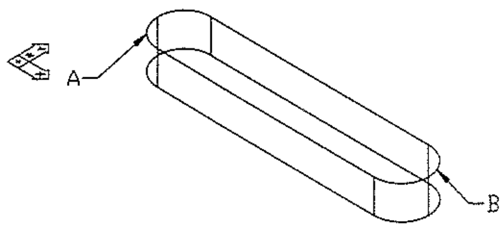


Figure 3.285 Profile extruded to distance Thk

Use the AMHOLE command to place two through holes concentric to A and B (Figure 3.285). The diameter of the hole is Hole. (See Figure 3.286.)

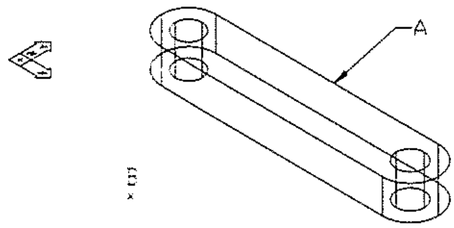


Figure 3.286 Concentric through holes of diameter Hole placed

Now linkage bar A is complete. To see how global design variables can be used on a set of solids in a drawing file, you will copy the definition of linkage A. (See Figure 3.287.)

<Part> **<Part>** **<Copy Definition>**

Cataloged part or subassembly to copy (? for list): **[Select A (Figure 3.286).]**

New part or subassembly name: **Linkage_B**
Select insertion point: [**Select B (Figure 3.286).**]
Select insertion point: [**Enter**]

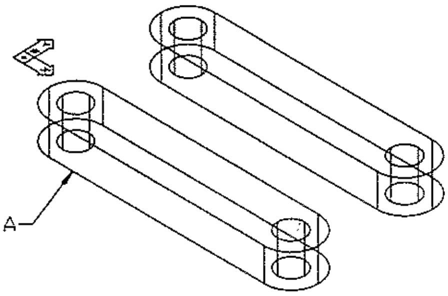


Figure 3.287 Definition copied

Now you have two definitions. Set the new definition as the active solid part. Then use the AMEDITFEAT command to change the dimension A (Figure 3.288) to LENGTH_B. (See Figure 3.288.)

<Part> **<Part>** **<Make Active>**

Command: **AMACTIVATE**
Select part to activate (or ?): [**Select A (Figure 3.287).**]

<Part> **<Edit Feature>**

Command: **AMEDITFEAT**
Independent array instance/Sketch/surfCut/Toolbody/<select Feature>: [**Select A (Figure 3.287).**]
Select object: [**Select A (Figure 3.288).**]
Enter new value for dimension: **LENGTH_B**
Select object: [**Enter**]

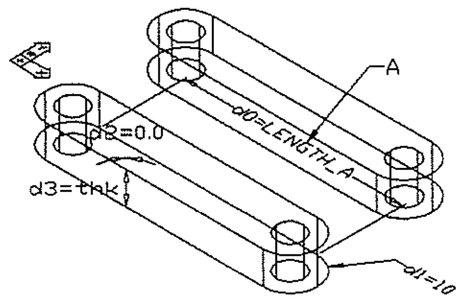


Figure 3.288 New definition made into active solid and edited

Use the AMUPDATE command to update the change. Note that a change in the new definition does not affect the other definition. (See Figure 3.289.)

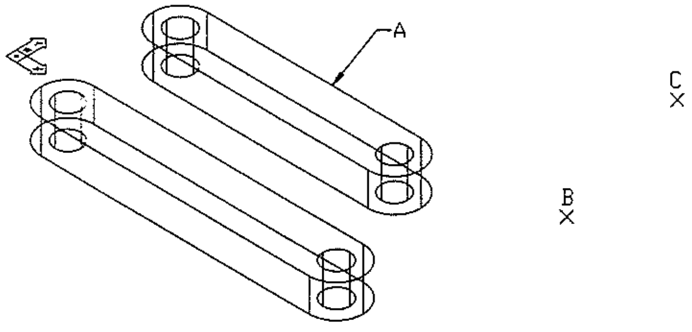


Figure 3.289 Change updated

Now copy the solid definition twice. (See Figure 3.290.)

<Part> **<Part>** **<Copy Definition>**

Cataloged part or subassembly to copy (? for list): **[Select A (Figure 3.289).]**
New part or subassembly name: **Linkage_C**
Select insertion point: **[Select B (Figure 3.289).]**
Select insertion point: **[Enter]**

<Part> **<Part>** **<Copy Definition>**

Cataloged part or subassembly to copy (? for list): **[Select A (Figure 3.289).]**
New part or subassembly name: **Linkage_D**
Select insertion point: **[Select C (Figure 3.289).]**
Select insertion point: **[Enter]**

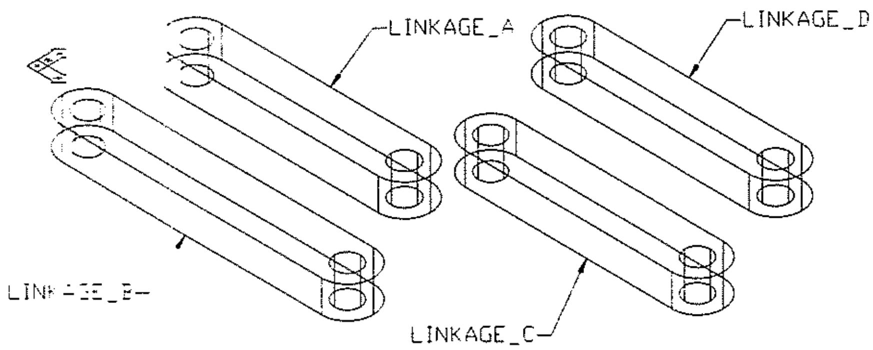


Figure 3.290 Definitions copied

Set Linkage_C (Figure 3.290) as the active solid part. Then change the dimension Length_A to Length_C. After that, set Linkage_D (Figure 3.290) as the active solid part and change the dimension Length_A to Length_D. After changing the dimensions, update the parts. (See Figure 3.291.)

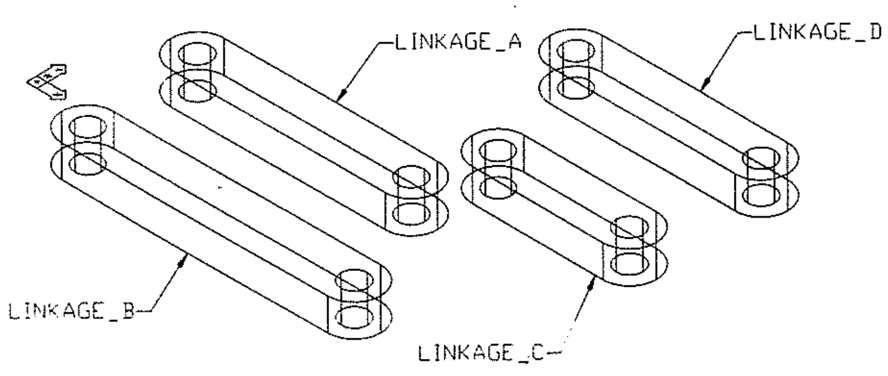


Figure 3.291 Dimensions changed and updated

Now you will construct a pivot pin. Activate a new solid part. Then set the current layer to Sketch. After that, construct a polyline as shown in Figure 3.292.

<Part> **<Part>** **<New Part>**

Select (or): **PIVOT**

[Mechanical Main] **[Layer Control]**

Current layer: **Sketch**

<Design> **<Polyline>**

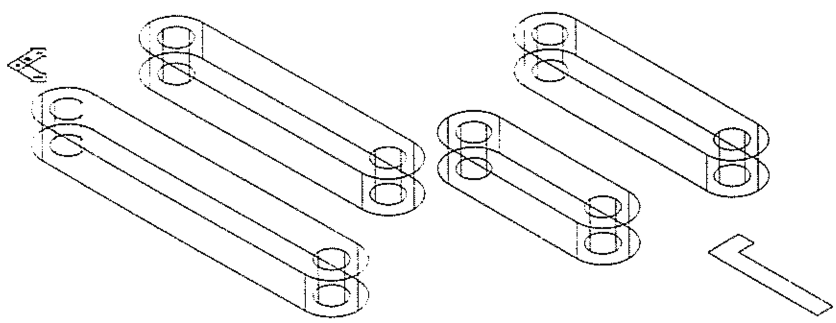


Figure 3.292 New part activated, polyline constructed

Resolve the polyline to a profile. Then add parametric dimensions to the profile as shown in Figure 3.293. When dimensioning, use design variables.

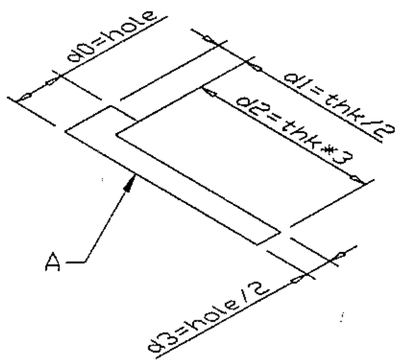


Figure 3.293 Sketch resolved, parametric dimensions added

Set the current layer to Solid. Then use the AMREVOLVE command to revolve the profile about A (Figure 3.293) so that it becomes a revolve solid. After that, set the current layer to Sketch and construct a circle as shown in Figure 3.294.

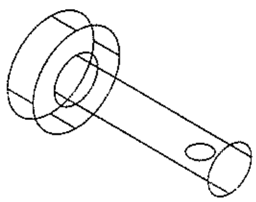


Figure 3.294 Profile revolved, circle constructed

Resolve the circle to a profile. Then add parametric dimensions as shown in Figure 3.295.

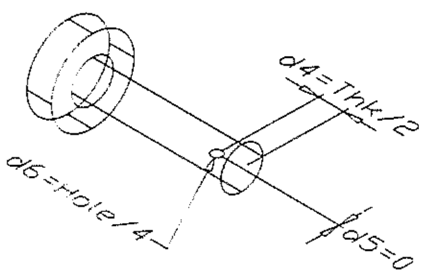


Figure 3.295 Circle resolved, dimensions added

Use the AMEXTRUDE command to extrude the profile from mid-plane to a distance of Hole to cut the base solid feature. (See Figure 3.296.) The model is complete. Save your drawing.

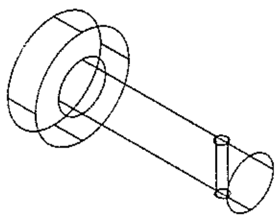


Figure 3.296 Resolved profile extruded

<File> **<Save>**

File name: **4bar.dwg**

In making these models, you learned how to use a set of global design variables in constraining the dimensions of a set of solid parts and you practiced copying solid definitions in a multi-part drawing. Now use the AMVARS command and select the Global tab. (See Figure 3.297.)

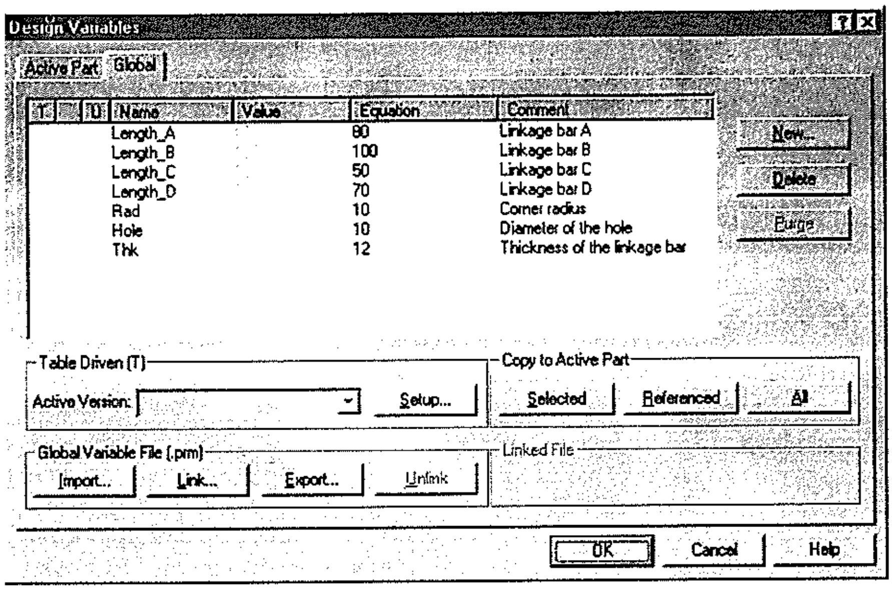


Figure 3.297 Design Variables dialog box

Compare Figure 3.297 with Figure 3.283. Because the variables have been used in the drawing, the letter U (U stands for unused) on each of the entries in Figure 3.283 has disappeared in Figure 3.297. Now you have a set of components for the four-bar linkage. In the next chapter, you will put them together to form an assembly.

For the design variables to be used by other drawing files, you can use the [Export...] button to export the design variables to a file in .prm format, then use the [Import...] button to import the .prm file to another drawing. In addition to importing, you can use the [Link...] button to link to a .prm file. Once the file is linked, any change in it will be reflected in the drawing.

Spur Gear Project

Figure 3.298 shows a rendered image of a spur gear. Before making the model, you will construct an Excel table (spreadsheet). In the columns of the Excel table, you will define a set of design variables. In the rows, you will define a set of versions of design variables. Each design variable will have a different value in each of the versions.

After making the Excel table, you will construct a solid of revolution for the gear blank, cut out an extrude solid for a gear tooth, and array the cut solid to form the gear teeth. When constructing the gear, you will use the design variables defined in the Excel table.

Because there are a number of versions of design variables, you can change the solid to another version by selecting a different version in the Excel table. The solid parts are called table-driven. The term *table-driven* refers to the fact that the value of the design variable is driven by an Excel table (spreadsheet). (You should have Excel properly installed on your computer before you work on this project.)

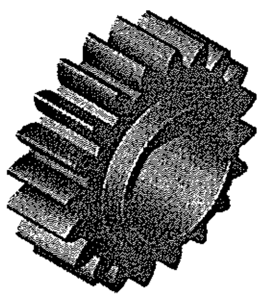


Figure 3.298 Rendered image of a spur gear

Start a new multi-part drawing. Use Solid.dwt as the template. Then construct a sketch and resolve it to a profile as shown in Figure 3.299.

<File> **<New...>**

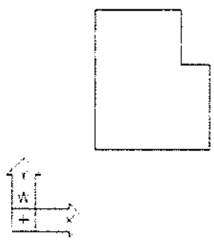


Figure 3.299 Sketch constructed and resolved

Use the AMVARS command to set up an Excel table. (See Figure 3.300, the Design Variables dialog box.) Select the Active Part tab. Then select the [Table Setup...] button. (See Figure 3.301, the Table Setup dialog box.) Here you will decide the start cell of the Excel table to read and the directions of version numbers and variable names. As we have said, you will define the variables in the columns and define the versions in the rows, accept the default, and select the [OK] button. Then select the [Create Table...] button. This will invoke Excel. In the File Name dialog box, specify an Excel file name, Gear.xls. After that, construct an Excel spreadsheet as shown in Figure 3.302.

After filling in the spreadsheet, save and close the file to return to the Design Variables dialog box. Now select the [Update Link] button to link the Excel spreadsheet to the current drawing. Then select the 20teeth09 item in the Active Version pull-down list. (See Figure 3.303.) After that, select the [OK] button.

<Part> **<Design Variables...>**

Command: **AMVARS**

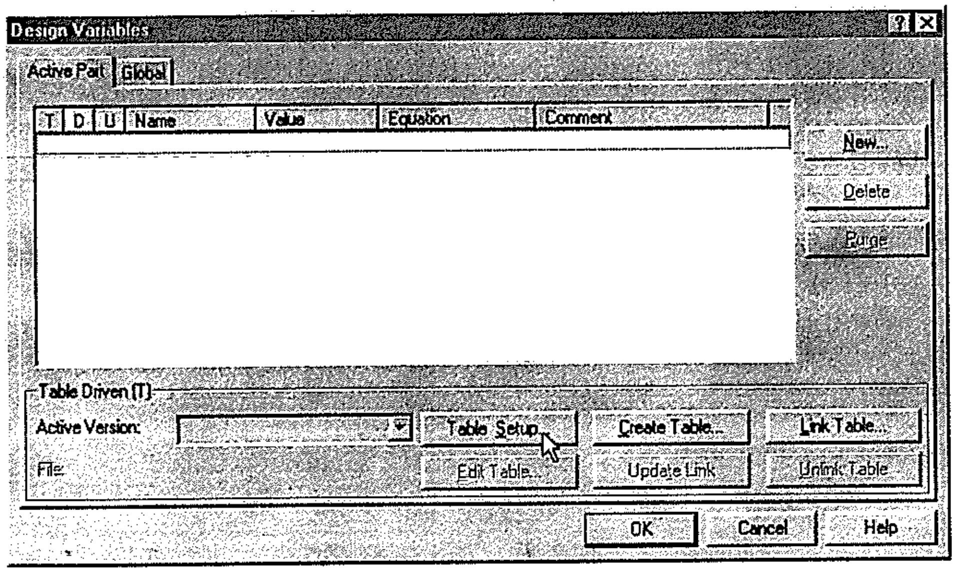


Figure 3.300 Active Part tab of the Design Variables dialog box

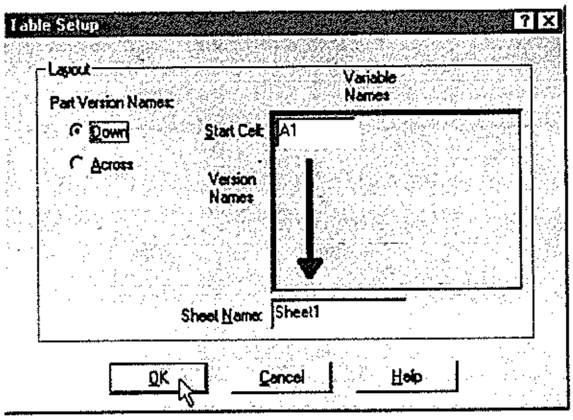


Figure 3.301 Table Setup dialog box

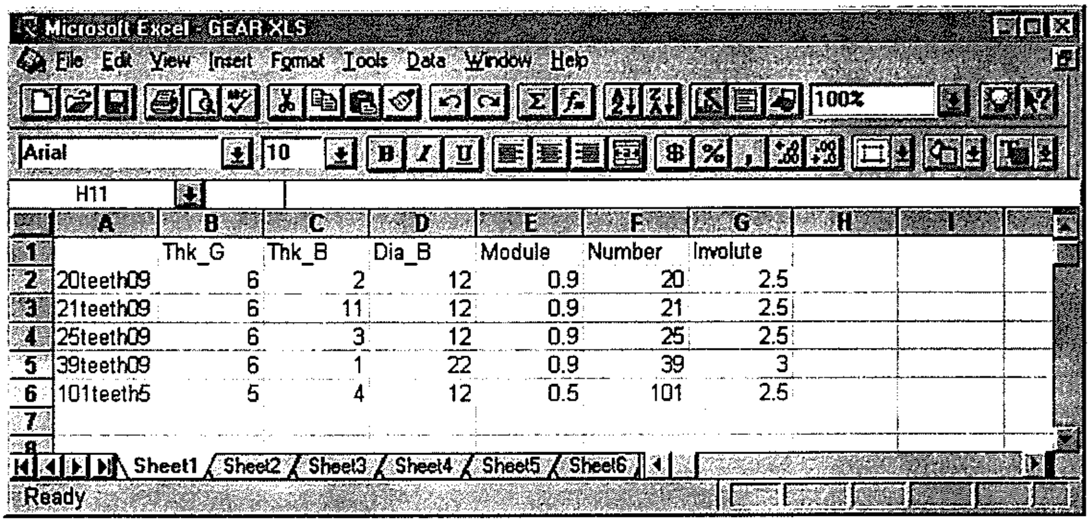


Figure 3.302 Excel spreadsheet

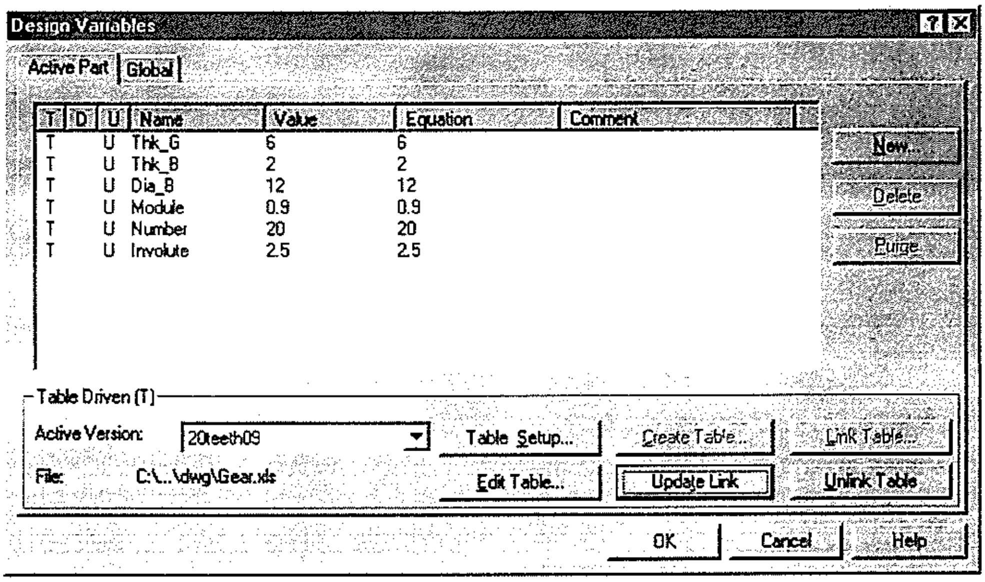


Figure 3.303 Design variables linked to an Excel spreadsheet

Now the design variables in this drawing are linked up with an Excel spreadsheet. Set the parametric dimension display to equation. Then add parametric dimensions to the resolved profile as shown in Figure 3.304.

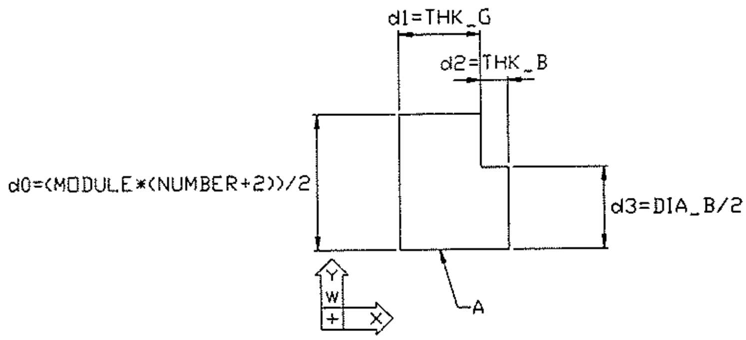


Figure 3.304 Parametric dimensions added

Set the display to an isometric view. Then set the current layer to Solid. After that, use the AMREVOLVE command to revolve the profile to create a revolve solid. The axis of revolution is A (Figure 3.304) and termination is full. (See Figure 3.305.)

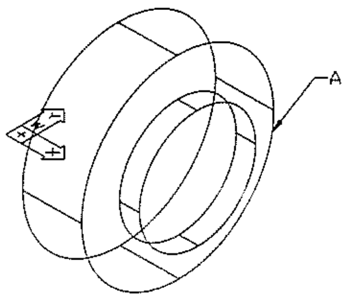


Figure 3.305 Display set, revolve solid constructed

Set the current layer to Sketch, set the sketch plane to face A (Figure 3.305), and set the display to the plan view of the new sketch plane.

Use the AMPREFS command. Uncheck the Assume Rough Sketch box to override the angular tolerancing rule. This way, only the end points of the line and the arc segments are joined while you are resolving.

<Part> **<Preferences...>**

[Yes Apply Constraint Rules
No Assume Rough Sketch
OK]

As shown in Figure 3.306, construct a set of lines and arcs. Note that the main profile consists of six arc segments: AB, BC, CD, DE, EF, and FA (Figure 3.306). Arc FC (Figure 3.306) and lines CG, DG, EG, FG, and HG (Figure 3.306) are construction lines. You should change their linetype to hidden or any linetype other than continuous. After constructing the lines and arcs, and changing the appropriate linetypes, use the AMPROFILE command to resolve them to a profile.

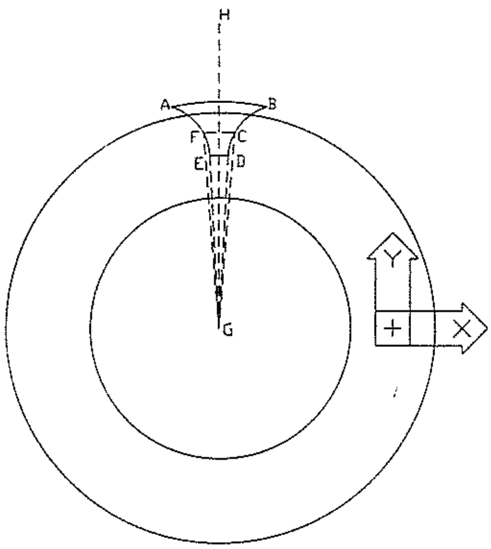


Figure 3.306 Sketch plane set, display set to plan view, and sketch constructed

Add two parametric dimensions (d8 and d9 in Figure 3.307) to locate the end point G (Figure 3.306). Then use the AMSHOWCON command to find out what geometric constraints are applied to your sketch while it is resolved and compare them with the constraint requirements below. After that, use the AMDELCON command to remove unwanted constraints and use the AMADDCON command to complete the constraint requirements.

Line HG (Figure 3.306)	vertical
Arc AB (Figure 3.306) and the outer diameter of the solid	concentric
Arcs AB, FC, and ED (Figure 3.306)	concentric
Arcs BC and CD (Figure 3.306)	tangent
Arc CD and line DG (Figure 3.306)	tangent
Arcs AF and FE (Figure 3.306)	tangent
Arc FE and line EG (Figure 3.306)	tangent
Arcs BC, CD, EF, and FA (Figure 3.306)	equal radii

Now add the parametric dimensions as shown in Figure 3.307.

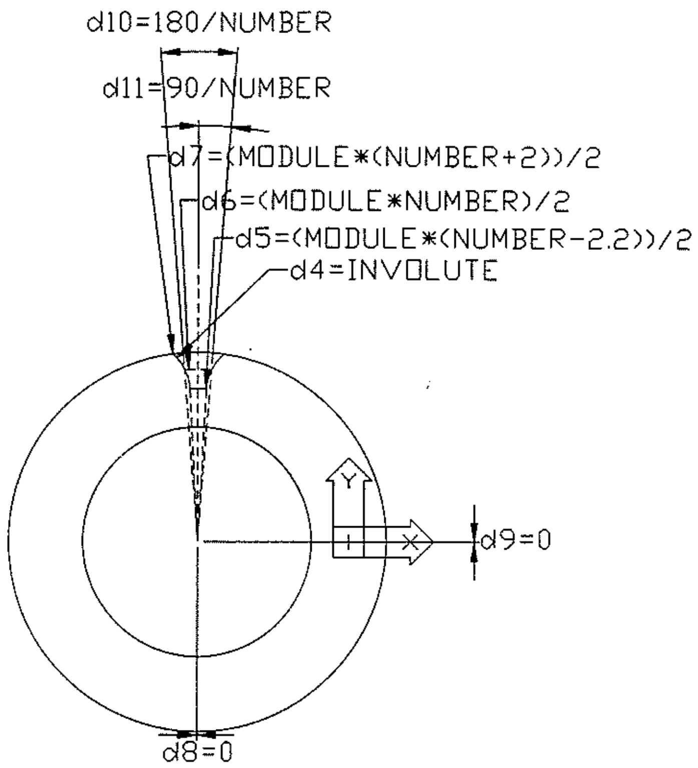


Figure 3.307 Profile constrained

Set the display to an isometric view. Then extrude the profile to create an extrude solid and cut through the base solid. In addition, construct a work axis. (See Figure 3.308.)

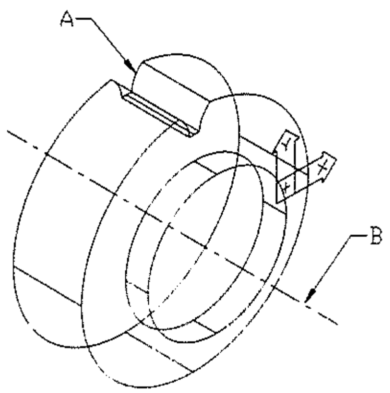


Figure 3.308 Profile extruded and cut, work axis constructed

Now a tooth is cut on the gear blank. To cut the other gear teeth, use the AMARRAY command to make a polar array. (See Figure 3.309.)

<Part> **<Placed Features>** **<Array...>**

Command: **AMARRAY**
Select feature: **[Select A (Figure 3.308).]**
Next/<Accept>: **[Enter]**

[Polar
Number of Instances: **NUMBER** Angle Type **Full Circle**
Rotate as Copied
OK **]**

Select work point or work axis for center of array: **[Select B (Figure 3.308).]**

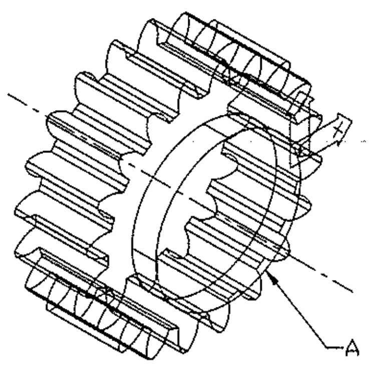


Figure 3.309 Extrude solid arrayed

Use the AMSKPLN command to set the sketch plane to A (Figure 3.309). Then set the display to the plan view. After that, construct a horizontal line and a concentric arc as shown in Figure 3.310.

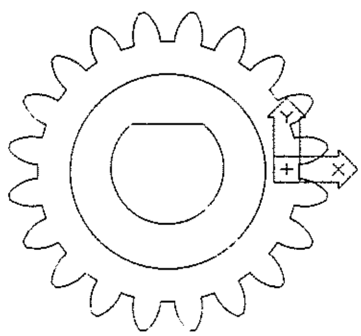


Figure 3.310 Sketch plan set, display set, and sketch constructed

Resolve the sketch to a profile. Then add the following geometric constraints and add parametric dimensions as shown in Figure 3.311.

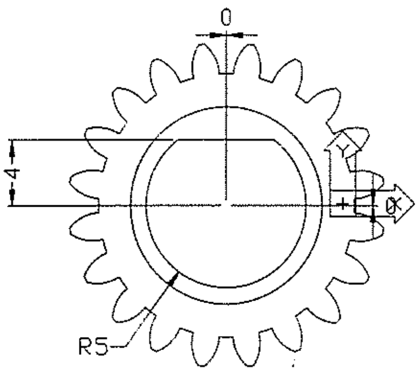


Figure 3.311 Geometric constraints and parametric dimensions added

Set the display to an isometric view. Then extrude the profile to cut through the solid. (See Figure 3.312.)

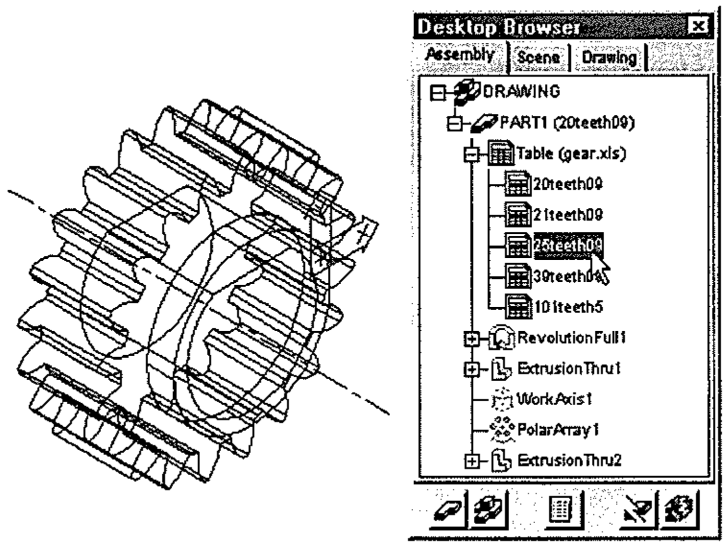


Figure 3.312 Profile extruded to cut through the solid

Now the gear is complete. To see how an Excel spreadsheet can be used to drive a solid, select the 25teeth09 item (Figure 3.312) in the browser and double-click. Note how the solid changes. Try out the other items in the table. After that, change the solid back to the 20teeth09 item.

The model is complete. Save your drawing.

<File> **<Save>**

File name: **Gear20t.dwg**

In making this model, you set up a set of global variables in Excel. In the Excel table, you defined a set of design variables in the columns and defined a number of versions of design variables in the rows. Thus, each design variable has a number of versions. After using the design variables in a solid part, you can select a version in the Excel table to make changes in the solid part. The solid part is called a table-driven solid part.

3.12 Split and Combine

A solid part can be split into two solid parts and two solid parts can be combined to form a single solid part. To combine two solid parts, you set one of them as the active solid and use the other as the tool solid. Then you combine them by joining, subtracting, or intersecting. For a proper parametric relationship between the active solid and the tool solid to be maintained, they have to be assembled together with assembly constraints applied properly. Because you will learn how to assemble solids and how to apply assembly constraints in the next chapter, combination of two solid parts will be dealt with in the next chapter. Here you will work on two projects. In the first, you will split a solid into two. In the second, you will prepare two solids for assembly and combination in the next chapter.

Die Cast Mold Project

Figure 3.313 shows a rendered image of a die cast component and two halves of a die cast mold. Note that the runners and risers for the mold have been omitted for simplicity. To make these parts, you will construct a rectangular extrude solid feature. Then you will construct a second sketch to represent the profile of the die cast component. When extruding the second sketch, you will use the Split option. This way, the first solid feature is cut by the second sketched feature and the cut-out portion becomes a split part. After using the split option in the extrusion process, you will further split the rectangular solid part into two. Thus, you will have three solid parts in the drawing.

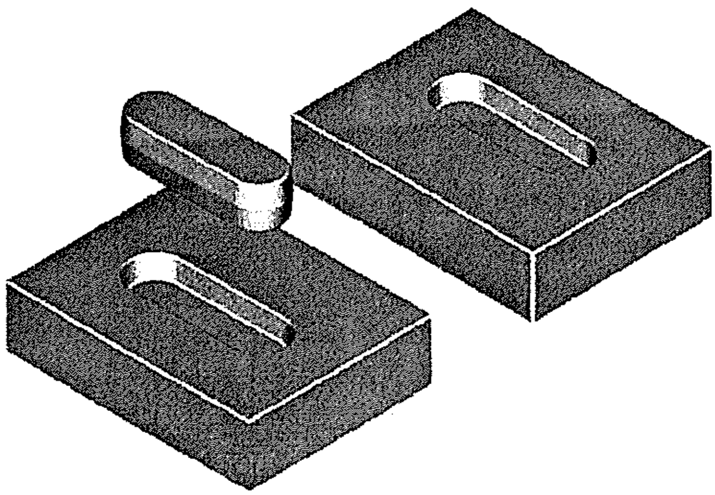


Figure 3.313 Rendered image of the die cast mold

Start a new multi-part drawing. Use Solid.dwt as the template. As shown in Figure 3.314, construct a rectangle, resolve it to a profile, and add appropriate parametric dimensions.

<File> **<New...>**

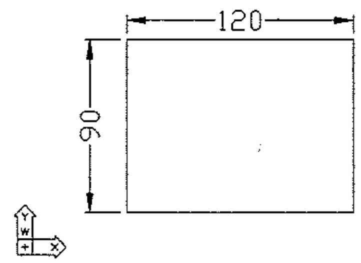


Figure 3.314 Rectangle resolved

Set the display to an isometric view. Then set the current layer to Solid. After that, use the AMEXTRUDE command to extrude the resolved profile a distance of 60 units. (See Figure 3.315.)

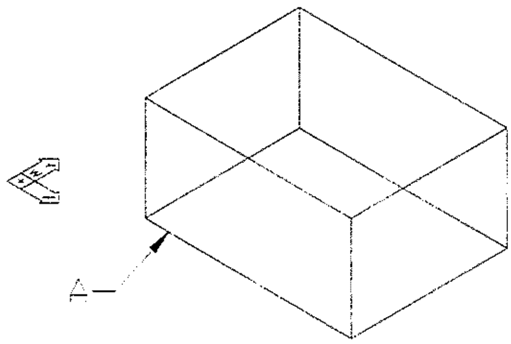


Figure 3.315 Display set to an isometric view, profile extruded

Construct a work plane that is offset 30 units from the bottom face A of the extrude solid (Figure 3.315). Set the sketch plane to the new work plane. (See Figure 3.316.)

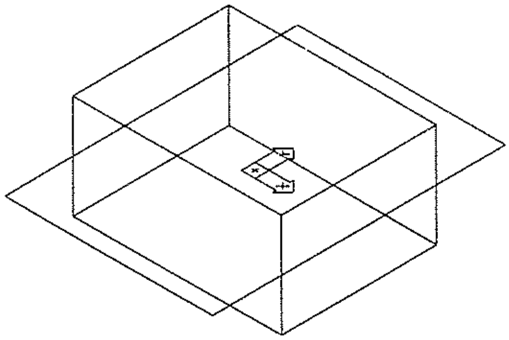


Figure 3.316 Offset work plane constructed

Set the display to the plan view of the new sketch plane and set the current layer to Sketch. Then construct a sketch, resolve it, and constrain it as shown in Figure 3.317.

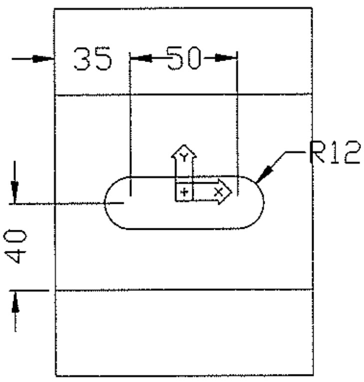


Figure 3.317 Display set, sketch constructed, resolved, and constrained

Now set the display to an isometric view. Then use the AMEXTRUDE command to extrude the profile. Use the Split option. (See Figure 3.318.)

Command: **8**

<Part> **<Sketched Features>** **<Extrude...>**

Command: **AMEXTRUDE**

[Operation **Split** Termination **Mid Plane**
Size Distance: **20** Draft Angle: **-5**
OK]

Enter name of the new part <PART2>: **FORGE**

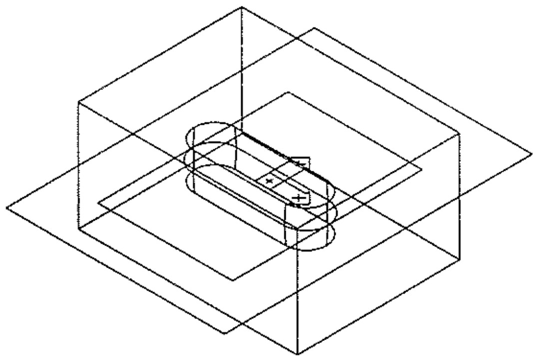


Figure 3.318 Base solid split by the extruded solid

When you use the Split option to combine a sketched solid feature with an existing solid, the original solid is cut by the newly formed solid, and the cut-out portion of the original solid is split to form a new solid part. To see the two individual parts clearly, select them in the browser (Figure 3.319). Then hide and unhide them one by one.

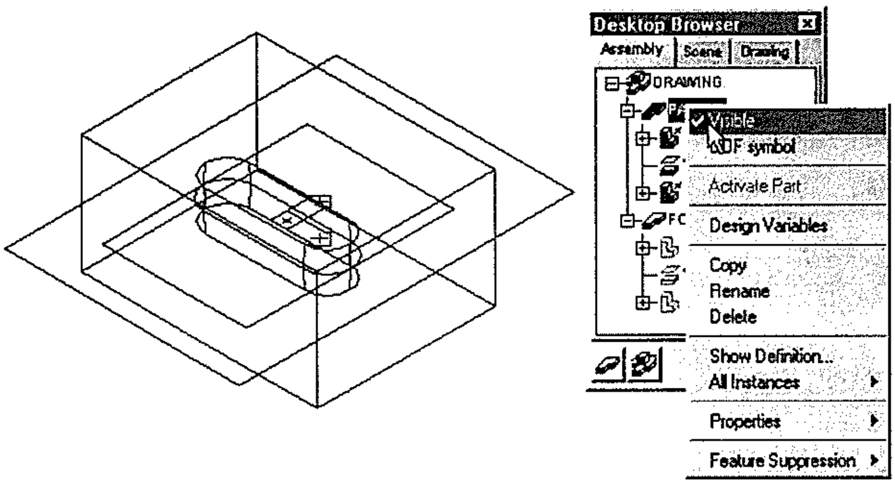


Figure 3.319 Controlling visibility in the browser

Now you have two solids: a rectangular extrude solid (die cast mold base) and an object split from the base solid (die cast component). Hide the split solid part (Forge). (See Figure 3.320.)

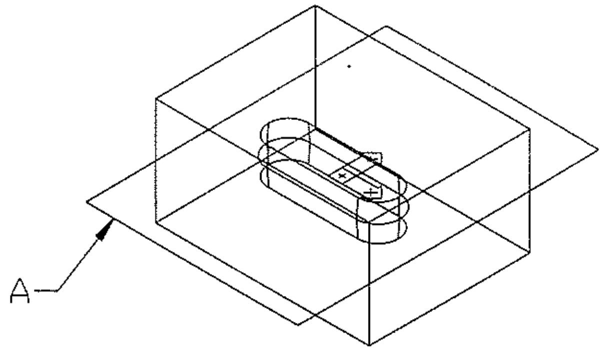


Figure 3.320 Split part hidden

Double-click on the main solid part in the browser to make it the active solid part. Then use the AMPARTSPLIT command to split it into two. When splitting, you have to specify a plane to split the solid. Then you specify which part of the solid is to be treated as a new solid part. After specifying the direction, you need to supply a name for the part. The solid is split. (See Figure 3.321.)

<Part> **<Placed Features>** **<Part Split>**

Command: **AMPARTSPLIT**
Select planar face, work plane, or split line for split: **[Select A (Figure 3.320).]**
Define side for new part: Flip/<Accept>: **[Accept if the arrow is pointing upward.]**
Enter name of the new part: **UPPER**

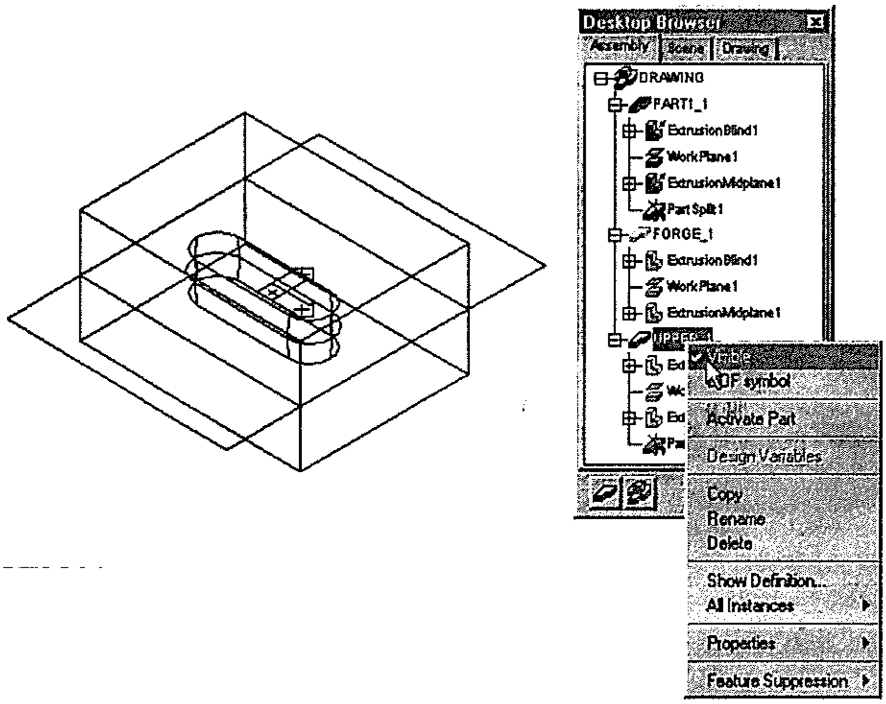


Figure 3.321 Solid part split

To see the splitting effect, select the new solid in the browser and press the right mouse button (Figure 3.321). Then select the Visible item to hide the solid. (See Figure 3.322.)

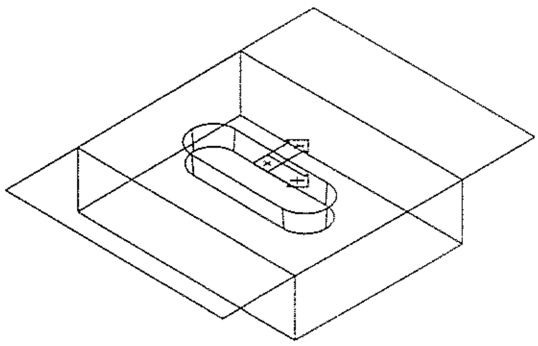


Figure 3.322 Upper solid hidden

The model is complete. Save your drawing.

<File>   <Save>

File name: **Diecast.dwg**

While working on this project, you learned how to use the Split option of making a sketched solid feature to cut a solid and construct a second solid from the cut-out portion. You also learned how to split a solid into two pieces by specifying a split plane.

Die Set for a Gear

Figure 3.323 shows a rendered image of a pair of dies for making a gear. This project is quite similar to the last project in that a set of dies is to be constructed from a mold base that is an extrude solid. Unlike the last project in which you started from making the mold base, you start from the molded part. You will work on this project in two stages. In this chapter, you will open a drawing and construct a second solid part in the drawing. Then in the next chapter, you will assemble and combine them.

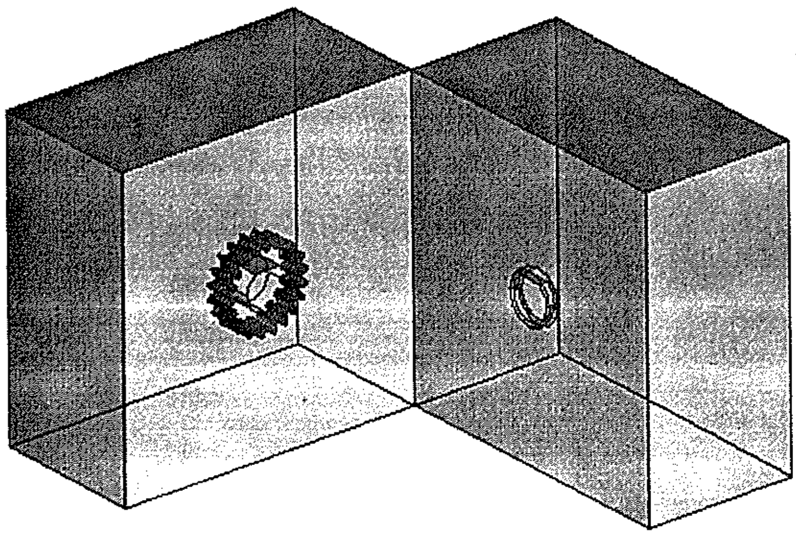


Figure 3.323 Rendered image of a pair of dies

Open the drawing Gear20t.dwg that you saved earlier. Then save it as Dieset.dwg.

<File> **<Open...>**

File name: **Gear20t.dwg**

<File> **<Save As...>**

File name: **Dieset.dwg**

Activate a new solid part. Then construct a rectangle, resolve it to a profile, and add parametric dimensions as shown in Figure 3.324.

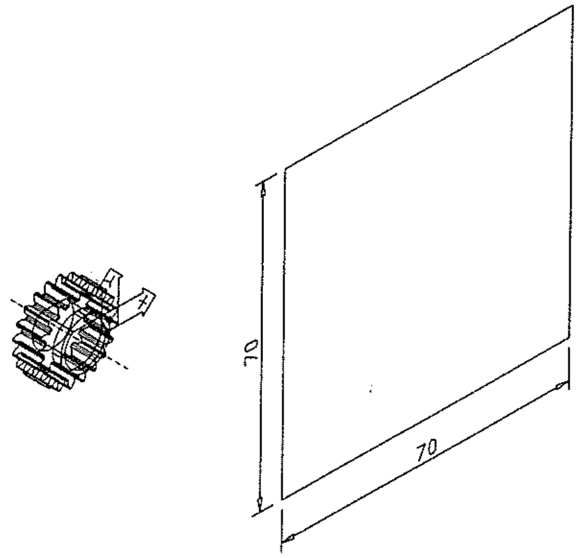


Figure 3.324 Rectangle constructed and resolved, parametric dimensions added

Extrude the profile a distance of 70 units. This is the mold base. (See Figure 3.325.)

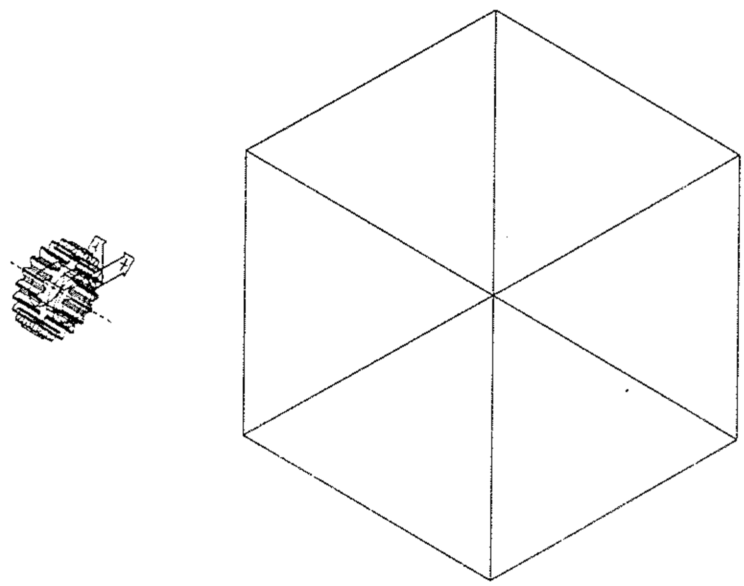


Figure 3.325 Resolved profile extruded

To complete this project, you have to assemble the gear and the mold base together properly, combine them by subtracting the gear from the mold base, and split the mold base into two. Because you need to learn assembly modeling techniques before you can go on, this project will be continued in the next chapter.

3.13 Part Modeling Utilities

Now you should have a thorough understanding of how to use Mechanical Desktop to construct parametric solid models. To conclude this chapter, let us have a tour of the part modeling utilities provided.

Mass Properties

With a solid, you can get seven sets of mass-properties information: general physical properties (mass, volume, and surface area) that are based on the value set in the density input box, moments of inertia, centroids, products of inertia, radii of gyration, principal axes, and principal moments.

Open the drawing file Arm_l.dwg. Then use the AMPARTPROP command. (See Figure 3.326, the Part Mass Properties dialog box.)

<File> **<Open...>**

File name: **Arm_l.dwg**

<Part> **<Part>** **<Mass Properties...>**

Command: **AMPARTPROP**

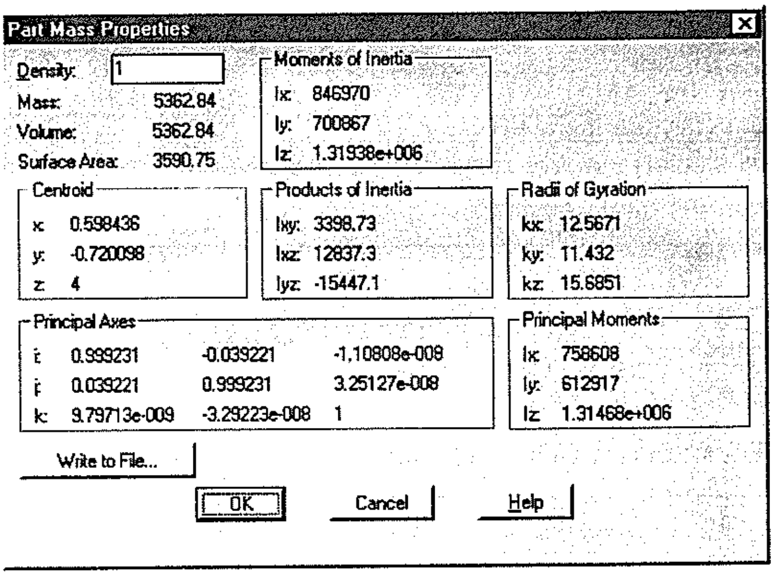


Figure 3.326 Part Mass Properties dialog box

By selecting the [Write to File...] button, you can save the mass properties to a file.

Downgrading to a Static Solid Part

You can modify a parametric solid because it has a parametric history. To transform a solid part into a static solid feature that cannot be modified, you can use the

AMMAKEBASE command. After you use this command, the parametric history is removed, and the solid becomes static and cannot be changed. However, you can add features to it and the added features can be edited.

<Part> <Part> <Make Base>

Command: **AMMAKEBASE**
Drawing view information and ability to edit existing features will be lost.
Create base from highlighted part ? No/<Yes>: **N**

Conversion with AutoCAD Native Solids

Mechanical Desktop is an application that runs on top of AutoCAD. By using AutoCAD, you can construct native solids. To convert a parametric solid to a native solid, you can use the EXPLODE command. Note that the EXPLODE command reduces an object to a simpler form each time you apply this command. If you apply the command once on a parametric solid, the solid is reduced to a native solid. If you further apply the command, the solid will be reduced to faces and lines. On the screen, you can hardly find any differences among a parametric solid, a native solid, and a set of faces and lines. If you are not sure what kind of object is on the screen, use the LIST command on the object.

<Modify> <Explode>

Command: **EXPLODE**

Note that both the AMMAKEBASE command and the EXPLODE command are irreversible once your drawing is saved.

You can also convert a native solid to a parametric solid. The converted solid cannot be edited, because the native solid does not have any parametric history. However, you can add features to it; they will be parametric and you can edit them.

<Part> <Part> <Convert Solid>

Stereolithography (STL) Output

Rapid prototyping (RP) is a technology used to construct physical prototypes from 3D CAD virtual prototypes. Although there are different RP systems employing different techniques, they function similarly. Unlike traditional machining processes that either remove material from a large solid or form objects from molten or semimolten material, RP systems use an additive process.

The starting point for making a rapid prototype is a 3D CAD model. You export the file electronically to the RP systems in Stereolithography Apparatus (STL) format. The STL file describes a set of triangular facets by specifying their coordinates. After exporting, the RP systems slice the electronic 3D CAD model (STL format) into very thin layers. To make the rapid prototypes, the RP systems make these thin layers one by one and compose them together.

Before exporting a solid to a STL file, you have to translate the model in 3D such that the entire model is lying on the positive sides of the X, Y, and Z axes, because many RP systems require positive X, Y, and Z coordinate values.

<Construct> **<Move>**

Command: **MOVE**

Because STL format describes a solid model in facets, curved faces are reduced to triangular facets during outputting to STL. Therefore, you have to set the facet resolution by using the FACETRES variable. The value of this variable ranges from 0.01 to 10.0. A value of 10 gives the highest resolution.

Command: **FACETRES**
New value for FACETRES: **10**

To check the smoothness of the model before sending the STL file to RP machines, you can use the HIDE or SHADE command to display the facets.

Command: **HIDE**

Command: **SHADE**

After translating the model to the positive sides of the X, Y, and Z axes and setting the FACETRES variable, you can output the model. Use the AMSTLOUT command.

Command: **AMSTLOUT**
Select object: [**Select the object.**]
(Angular tolerance = 15, aSpect ratio = 0, sUrface tolerance = 0, Vertex spacing = 0)
Angular tolerance/aSpect ratio/sUrface tolerance/Vertex spacing/<Continue>: [**Enter**]

If you have a native solid cut by a surface and you want to output an STL file, you can use the STLOUT command.

Command: **STLOUT**
Select a single solid for STL output:
Select objects: [**Select a model.**]
Select objects: [**Enter**]
Create a binary STL file ? <Y>: **Y**

Outputting to a Web Page

For a solid part to be viewed on a Web page, you can use the AMVRMLOUT command.

Command: **AMVRMLOUT**
Select objects: [**Select an object.**]

Select objects: **[Enter]**
File name: **[Specify a valid file name.]**

The output file can then be inserted into an HTML Web page file. To view the VRML object, you need plug-in applications.

Import and Export

With a multi-part drawing file, you can import a solid from another drawing file and output a solid to a drawing file. To write a solid part to an external drawing file, use the AMPARTOUT command. (See Figure 3.327.)

<Part> **<Part>** **<Write Out...>**

Command: **AMPARTOUT**

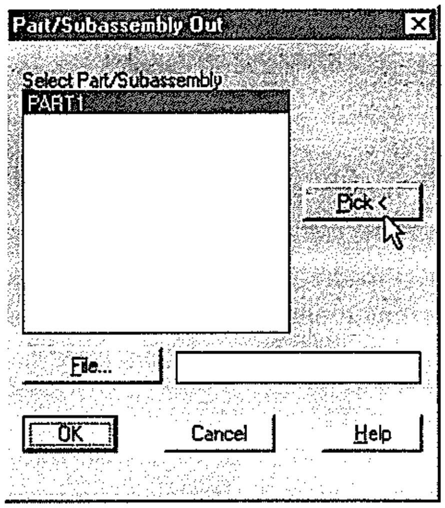


Figure 3.327 Part/Assembly Out dialog box

To copy a solid part into the current drawing, you can use the AMPARTIN command.

<Part> **<Part>** **<Copy In...>**

Command: **AMPARTIN**

3.14 Key Points and Exercises

In this chapter, you learned the key concepts of parametric solid modeling and the techniques of using parametric solid modeling tools to construct 3D parametric solid parts. In addition, you learned ways to use a NURBS surface to cut a parametric solid and saw the use of part modeling utilities.

A parametric solid part is constructed from a sketch that need not be accurate. Lines and arcs can be drawn to their approximate lengths and their end points need not coincide. You should concentrate on the shape, but not the size. When you are satisfied with the form and shape, you then resolve the sketch into a profile, path, cut line, or split line. After that, you constrain the resolved sketch by adjusting the geometric constraints and adding parametric dimensions. With resolved sketches, you can construct extrude solids, revolve solids, 2D sweep solids, 3D sweep solids, and loft solids. These are called sketched solid features because they are derived from sketches. You can also set up a cut line in an offset section and use a split line to split a face into two.

The first solid feature that you construct in a solid part is called the base solid feature. To add features to it, you construct other solids by using a building-block approach (join, cut, intersect, or split). In addition, you can combine two solid parts together and split a solid part into two.

Apart from sketching, you can construct common engineering features by specifying type, location, and parameters. These features are called placed solid features.

To maintain parametric relationships among the features, you use work features (work plane, work point, and work axis).

To sum up, a solid part can have three kinds of features: sketched solid features, placed solid features, and work features.

When constructing a solid part, you can make use of the Desktop Browser. Many commonly used commands are available in the browser as shortcuts. In the next chapter, you will learn assembly modeling and learn how to combine two solid parts into one. Now work on the following exercises to further enhance your knowledge.

Exercise 3.1

Give a brief account of the sketching approach in making a solid model. What are the three kinds of features of a solid? Use simple sketches to illustrate the various kinds of features.

Exercise 3.2

Outline the way NURBS surfaces can be incorporated into a parametric solid model, the way a parametric solid model can be made static, the way a parametric solid model can be changed to a native solid, and the way a native solid can be used as a static solid feature of a parametric solid model.

Exercise 3.3

Figure 3.328 shows a drawing of the pin of a differential casing.

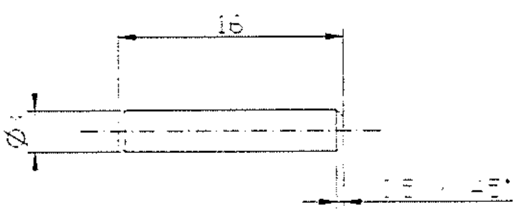


Figure 3.328 Pin of a differential casing

In Figure 3.328, the model consists of a cylindrical base feature and two placed chamfer features. To construct the base feature, you can either extrude or revolve a profile. To extrude, you can construct a circle, resolve it to a profile, add a parametric dimension to set the diameter to 3 units, and extrude the profile a distance of 16 units. To revolve, you can construct a rectangle, resolve it to a profile, add parametric dimensions to set the width to 16 units and the height to 1.5 units, and revolve the profile about the lower edge of the rectangular profile.

After making the base solid, place two chamfer features of 0.5 units from each end. Save your drawing after completion.

<File> **<Save>**

File name: **Difshaft.dwg**

Exercise 3.4

Figure 3.329 shows a rendered image of a plain bearing of a scale model car. There will be bearings of two sizes. You will construct an Excel spreadsheet in association with the solid part.

Figure 3.329 Rendered image of a plain bearing

As indicated in Figure 3.331, construct a rectangle and a line. Then change the linetype of the line to hidden. After that, resolve them to a profile.

As shown in Figure 3.330, use the AMVARS command to set up an Excel spreadsheet to define three design variables: Int_dia, Ext_dia, and Thk. They represent the internal diameter, external diameter, and thickness of the plain bearings. Set Bearing1 as the active version to construct the solid model.

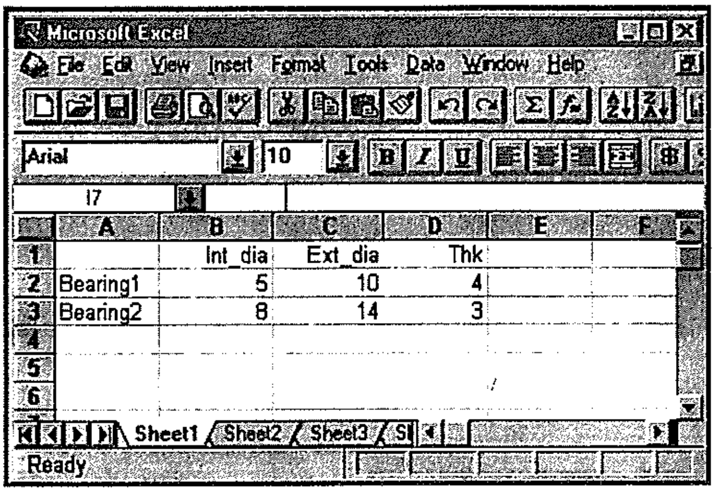


Figure 3.330 Excel spreadsheet

Construct a sketch, resolve it to a profile, and add parametric dimensions as shown in Figure 3.331.

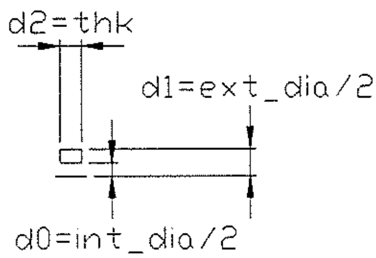


Figure 3.331 Fully constrained profile

Revolve the profile 360° about the construction line. Then save the drawing.

<File> **<Save>**

File name: **Bearing1.dwg**

Now set Bearing2 as the active version. Update the solid. Then save the drawing.

<File> **<Save>**

File name: **Bearing2.dwg**

Exercise 3.5

Figure 3.332 shows a rendered image of a hinge pin of a scale model car. You will construct three pins of different lengths.

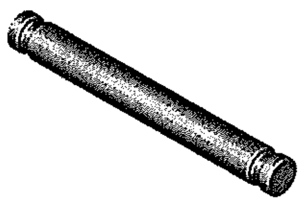


Figure 3.332 Rendered image of the hinge pin of a scale model car

In accordance with the following table, use the AMVARS command to construct a set of global design variables. Then export the set of variables to a global variable file. After that, link to the file.

Name	Equation
ID	2
OD	3
C	1
LG1	22
LG2	29
LG3	44

Construct a sketch, resolve it to a profile, and add parametric dimensions as shown in Figure 3.333.

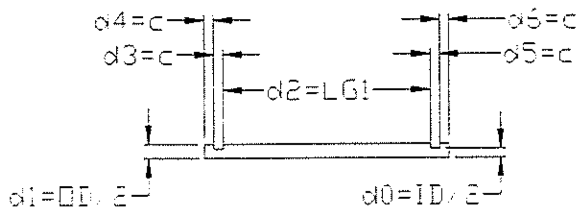


Figure 3.333 Fully constrained sketch

Revolve the profile 360° about the lower edge of the profile to construct a revolve solid. Save your drawing.

<File> **<Save>**

File name: **Hpin1.dwg**

Use the SAVEAS command to save the file to a new file name.

<File> **<Save As...>**

File name: **Hpin2.dwg**

Use the AMEDITFEAT command to change the d2 dimension (Figure 3.333) to LG2. Save the drawing. Then save the drawing to a new file.

<File> **<Save As...>**

File name: **Hpin3.dwg**

Use the AMEDITFEAT command to change the d2 dimension (Figure 3.333) to LG3. Then save the drawing. Now you have three sets of hinge pins.

Exercise 3.6

Figure 3.334 shows a drawing of the upper chassis of a scale model car, a rectangular solid part with four rounded corners and four drill holes. To make this solid part, you construct a rectangle with rounded corners, resolve the rectangle to a profile, constrain the profile, extrude the profile to a solid, and place drilled holes in it. After that, save your drawing.

<File> **<Save>**

File name: **Chassis_u.dwg**

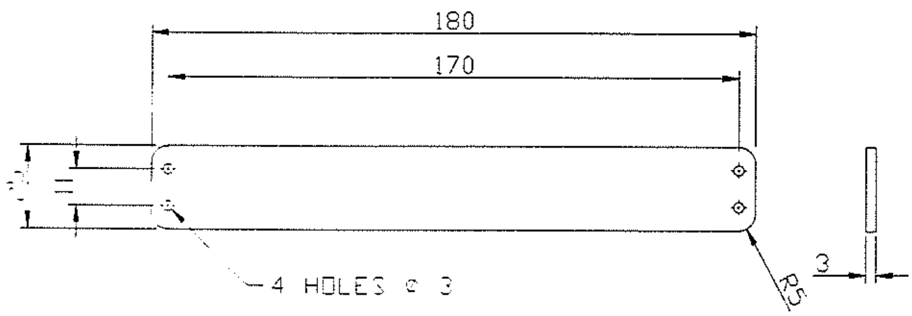


Figure 3.334 Upper chassis of a scale model car

Exercise 3.7

Figure 3.335 shows a drawing of the lower chassis of a scale model car.

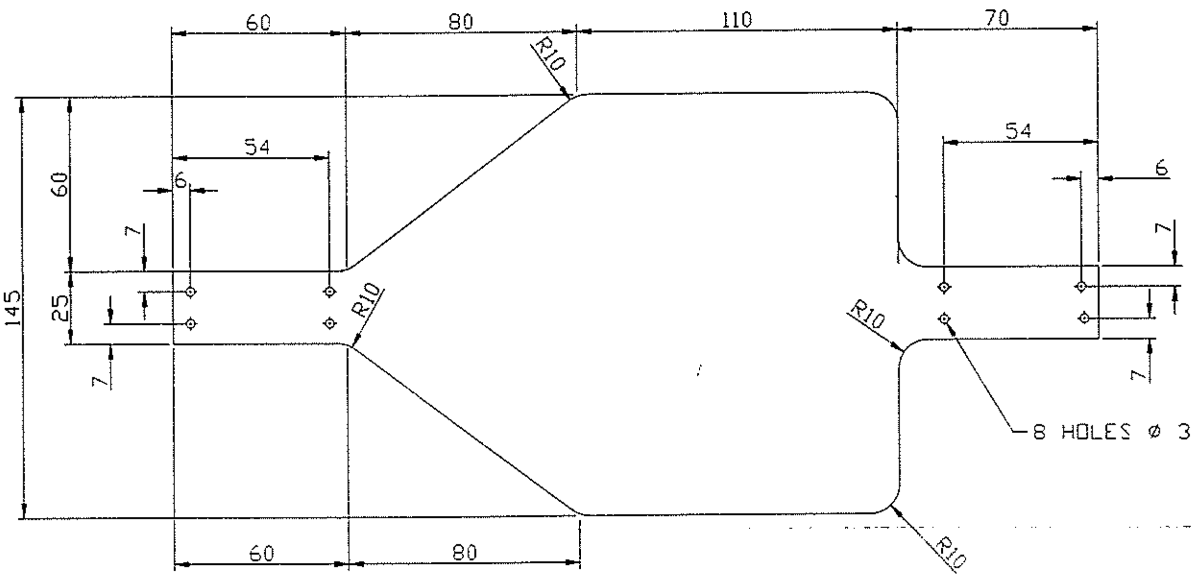


Figure 3.335 Lower chassis of a scale model car

Construct a profile. Then extrude it a distance of 3 units. After that, place eight through holes of diameter 3 units. Upon completion, save your drawing.

<File> <Save>

File name: **Chassis_l.dwg**

Exercise 3.8

Figure 3.336 shows the left front mounting of a scale model car. This solid part is an extrude solid with a number of drill holes, counterbore (C'bore) holes, countersink (C'sink) holes, and an oval-shaped opening. The C'sink holes are used for attaching the part to other parts; the through holes are used for placing the hinge pins of the suspension arms; the C'bore holes are used for setting the drive shafts; and the oval opening is used for putting in the electric motor.

According to functional requirements, the centers of the three C'bore holes need to be lying on a straight line and they should maintain a parametric relationship with the through holes. Therefore, you will construct a series of circular profiles, extrude the profiles to cut through the base solid, and use the extrude features as reference data to control the parametric dimensions of the C'bore holes, through holes, and oval-shaped opening.

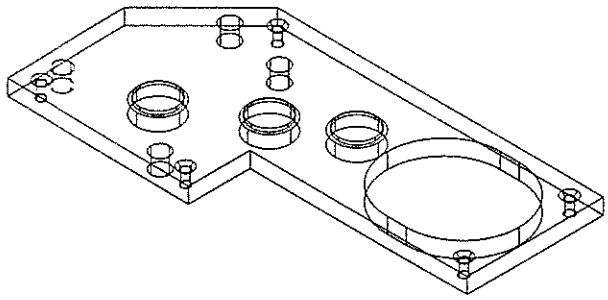


Figure 3.336 Left front mounting

Start a new multi-part drawing. As shown in Figure 3.337, construct a profile, fully constrain it, and extrude it a distance of 5 units. This is the base solid feature.

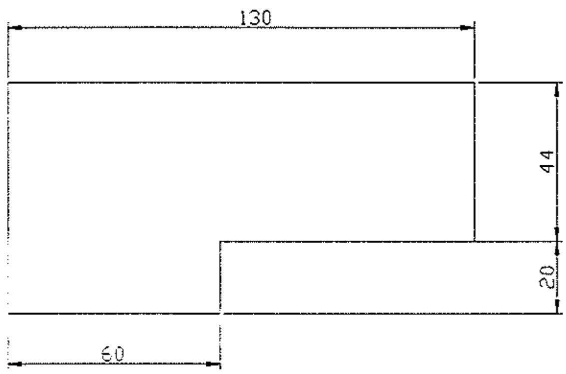


Figure 3.337 Dimensions of the base feature

Construct a circle and fully constrain it as shown in Figure 3.338. Then extrude the profile to cut through the base solid. This extrude feature will be used as a datum for other solid features.

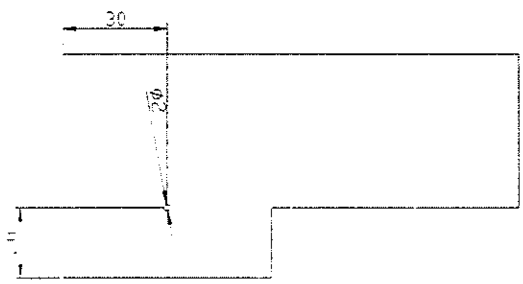


Figure 3.338 Base feature constructed, circular profile constructed

Using the last extrude solid as a reference, construct a profile as shown in Figure 3.339. After that, extrude the profile to cut the base solid. This extrude solid will also be used as a datum.

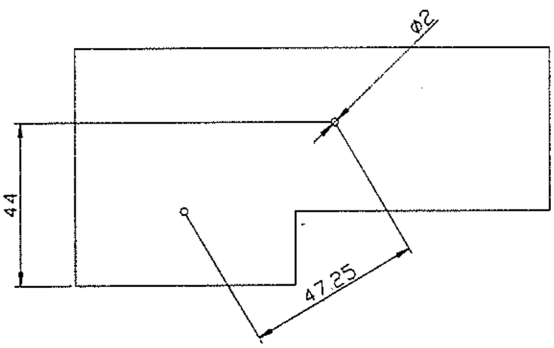
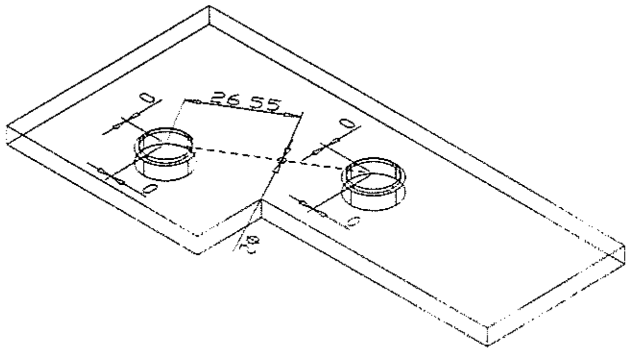


Figure 3.339 Circular profile extruded to cut the base solid, second circular profile constructed

Set the display to an isometric view. Then place two counterbore (C'bore) through holes that are concentric to the last two extrude solid features. The drill size is 12 units, the counterbore depth (C'Depth) is 4 units, and the counterbore diameter (C'Dia) is 14 units. After placing the holes, construct a sketch as shown in Figure 3.340.



3.340 C'bore holes placed concentric to the extrude solids, and sketch constructed and constrained

Extrude the circle. Then place a concentric counterbore (C'bore) hole with drill size of 12 units, counterbore depth (C'Depth) of 8 units, and counterbore diameter (C'Dia) of 14 units. After that, chamfer an edge a distance of 25 units times 25 units. (See Figure 3.341.)

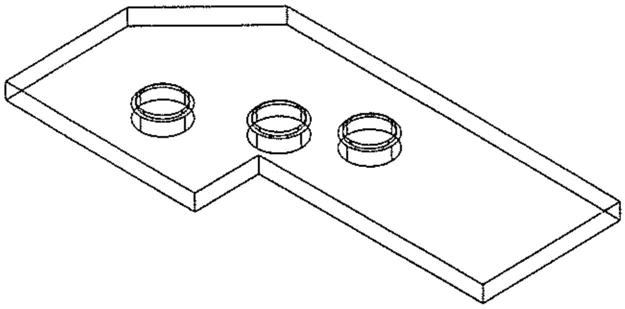


Figure 3.341 Circle extruded, concentric C'bore hole placed, and chamfer feature placed

As shown in Figure 3.342, place five countersink (C'sink) holes with diameter 3 units, countersink diameter (C'Dia) 5 units, and countersink angle (C'Angle) 90°.

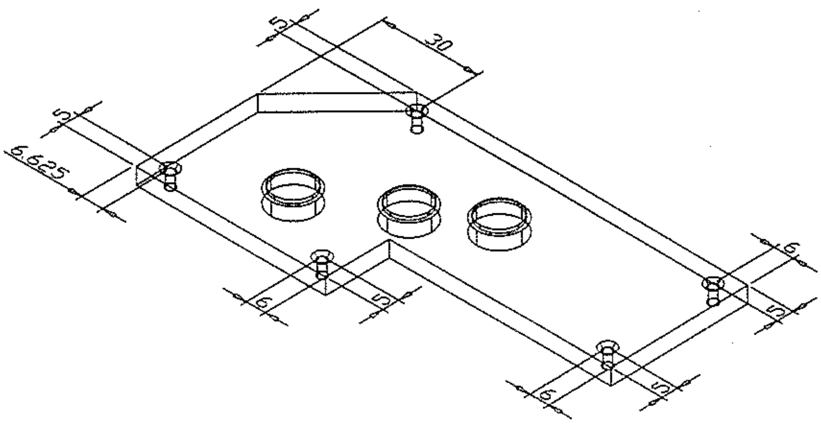


Figure 3.342 C'sink holes placed

As shown in Figure 3.343, construct a profile consisting of a circle and two hidden lines. The two hidden lines should be constrained to be perpendicular to each other. After constraining and adding parametric dimensions, extrude the profile to cut the base feature.

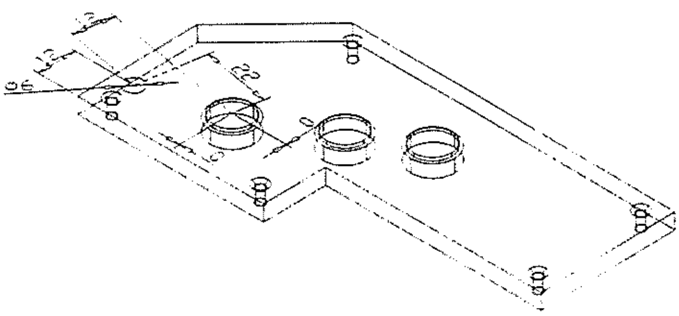


Figure 3.343 Profile constructed

As shown in Figure 3.344, construct a profile that consists of a circle and two hidden lines. One hidden line is drawn from the center of the last extrude hole to the newly constructed circle. The other line is drawn from a point on the first hidden line to the center of the C'bore hole. The two hidden lines should be perpendicular to each other. After resolving and constraining, extrude the profile to cut the base solid.

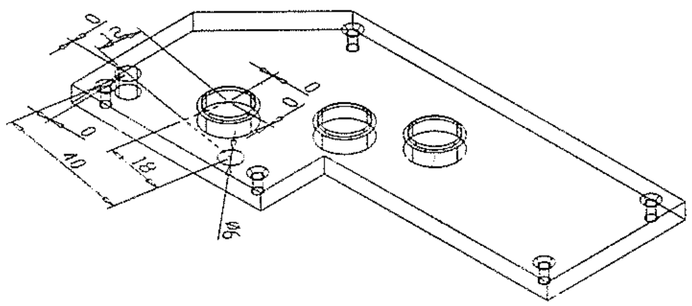


Figure 3.344 Profile extruded to cut base solid, another profile constructed

Construct another profile as shown in Figure 3.345. Then extrude it to cut the base solid.

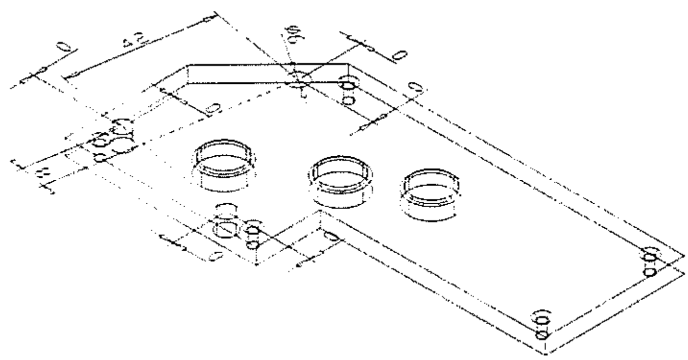


Figure 3.345 Profile extruded to cut base solid, next profile constructed

As shown in Figure 3.346, construct another profile. Extrude it to cut the base solid.

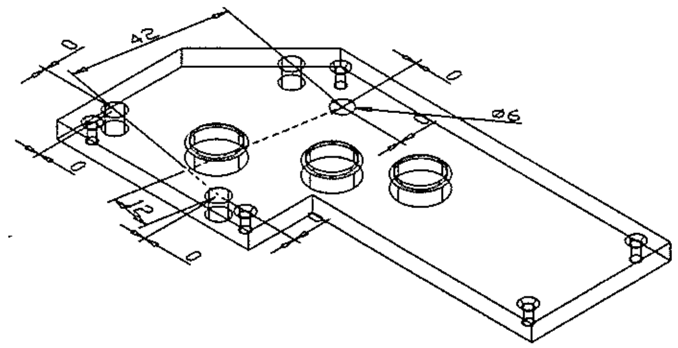


Figure 3.346 Profile extruded to cut, next profile constructed

Construct a profile as shown in Figure 3.347. Extrude it to cut the base solid.

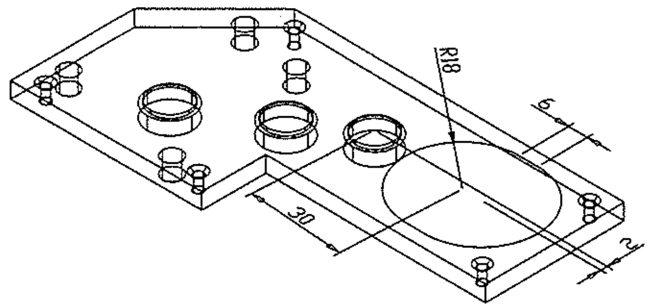


Figure 3.347 Profile extruded to cut, oval-shaped profile constructed

After cutting, the model is complete. (See Figure 3.336.) Save your drawing.

<File> **<Save>**

File name: **Boxf.dwg**

Exercise 3.9

Figure 3.348 shows the right front mounting of a scale model car. Basically, it is the mirror copy of the solid you constructed in Exercise 3.8, except that it has two additional extrude features.

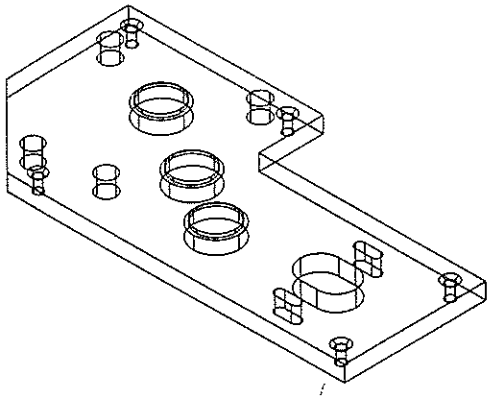


Figure 3.348 Right front mounting

Open the drawing Boxf.dwg if you have closed the file. You will construct this model in the same drawing file as the last one. Use the AMMIRROR command to make a mirror copy of the solid part. When mirroring, create a new part, not an instance. (See Figure 3.349.)

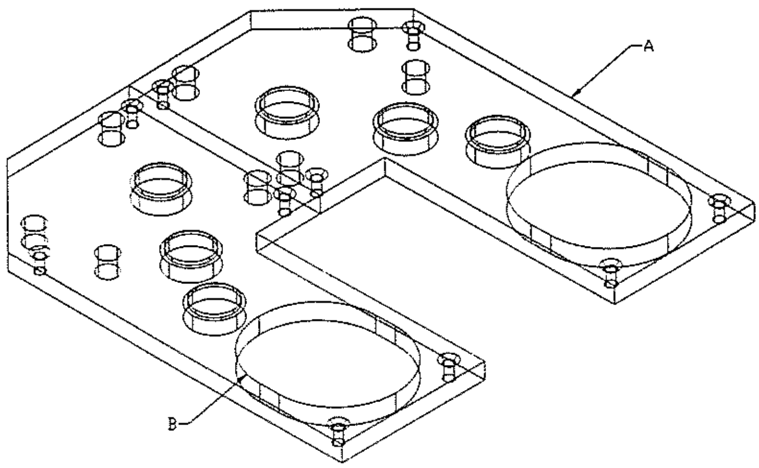


Figure 3.349 Solid part definition mirrored

Select the original solid definition A (Figure 3.349) in the Desktop Browser. Press the right mouse button. Then select the Visible item to hide it. After that, edit feature B (Figure 3.349) to change a dimension. Update the solid. (See Figure 3.350.)

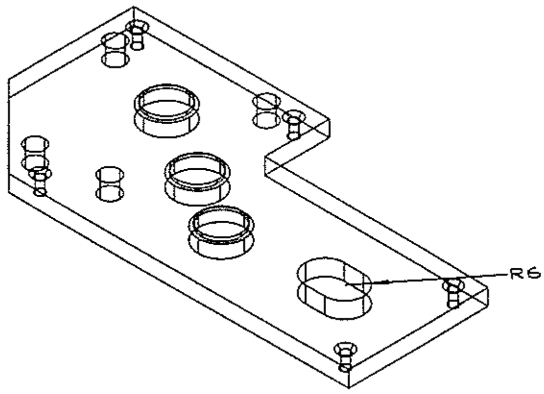


Figure 3.350 Original solid hidden, dimension of mirrored solid modified, and solid updated

Using the dimensions shown in Figure 3.351, copy two extrude features and modify the dimensions.

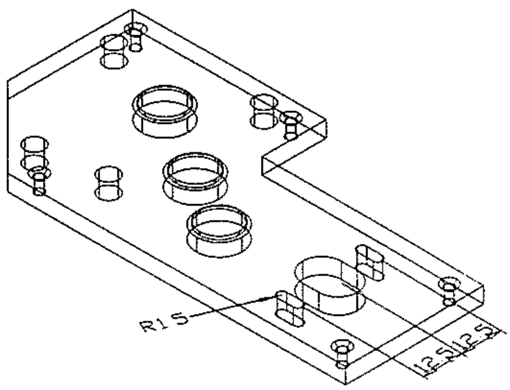


Figure 3.351 Dimensions of two copied features

The model is complete. Unhide the hidden solid. There are two solid parts in this drawing. Save your drawing.

<File> **<Save>**

Exercise 3.10

Figure 3.352 shows the right rear mounting of a scale model car. Unlike those in the front mounting, the drilled holes and C'sink holes are placed regularly from the edges of the base feature. To construct this model, you can make an extrude solid and then place the holes.

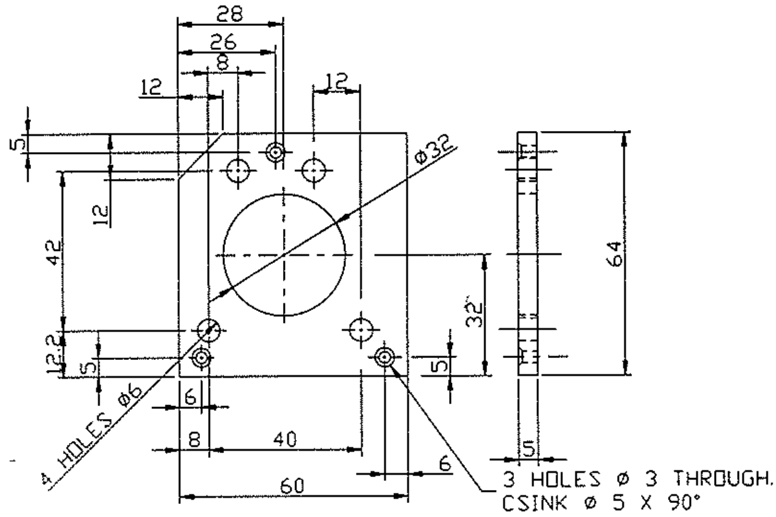


Figure 3.352 Right rear mounting

Start a new multi-part drawing. Using the outer dimensions, make an extrude solid. Then place the holes. Use the AMMIRROR command to make a mirror copy. When mirroring, create a new part. (See Figure 3.353.)

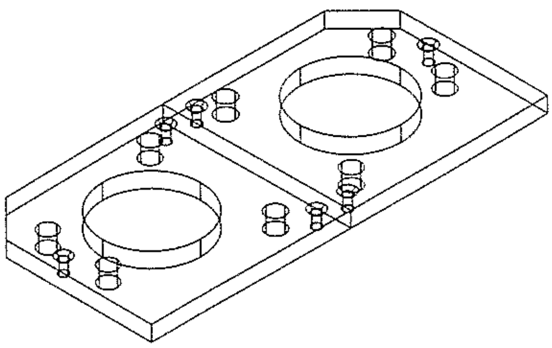


Figure 3.353 Solid part mirrored

Save the drawing again.

<File> **<Save>**

File name: **Boxr.dwg**

Exercise 3.11

Figure 3.354 shows a drawing of the mounting block of a scale model car.

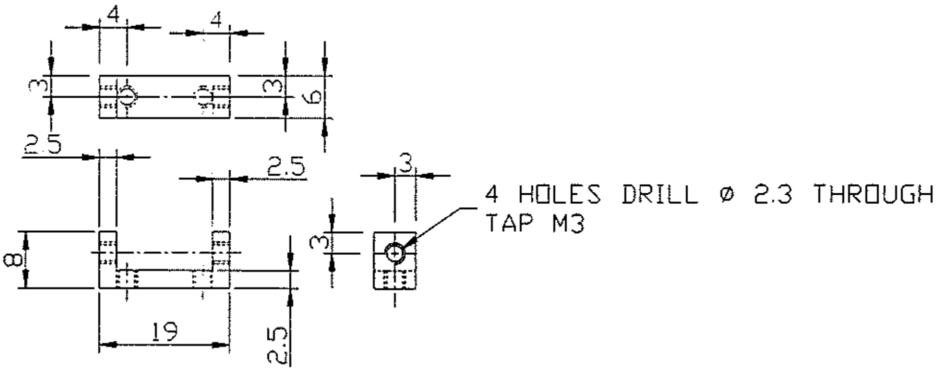


Figure 3.354 Mounting block

Using the dimensions and shape of the front view in Figure 3.354, construct a sketch, resolve it to a profile, constrain the profile, and extrude the profile to create an extrude solid. Then place four drilled through tapped holes. On completion, save your drawing.

<File> **<Save>**

File name: **Mountg.dwg**

Exercise 3.12

Figure 3.355 shows the drawing of the front axle of a scale model car.

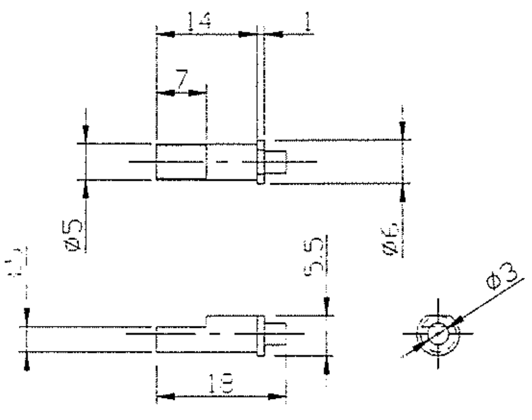


Figure 3.355 Front axle

As shown in Figure 3.356, construct a sketch, resolve it to a profile, fully constrain the profile, and revolve the profile to create a revolve solid. When revolving, select the lower horizontal line as the revolution axis.

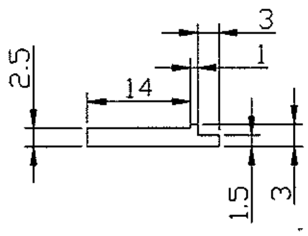


Figure 3.356 First sketch

Set the display to an isometric view. Then set the sketch plane to one end of the revolve solid and construct a profile. (See Figure 3.357.)

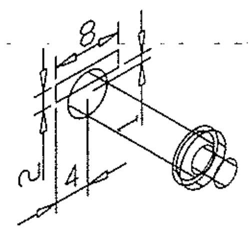


Figure 3.357 First sketch revolved, display set, and second sketch constructed

Extrude the profile a distance of 7 units to cut the base solid. Then construct the third sketch as shown in Figure 3.358.

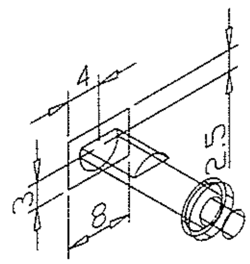


Figure 3.358 Second sketch extruded to cut, third sketch constructed

Extrude the third profile to intersect the base solid. The model is now complete.

<File> **<Save>**

File name: **Axlefd.dwg**

Exercise 3.13

Figure 3.359 shows a drawing of a U-shaped mounting bracket for a scale model car.

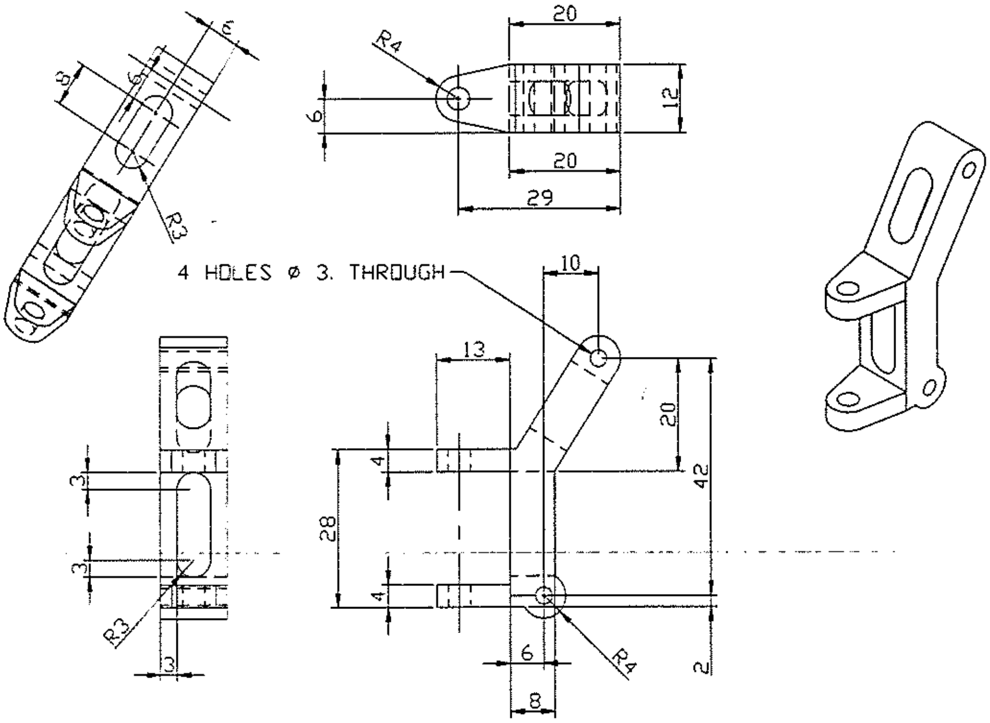


Figure 3.359 U-shaped mounting bracket

This solid part consists of four extrude solid features and four placed holes. Construct the first sketched feature as shown in Figure 3.360.

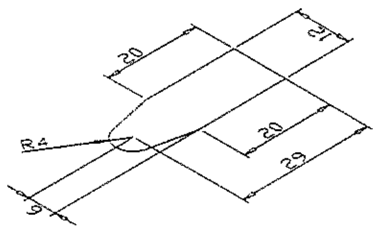


Figure 3.360 First sketched feature

Extrude the profile a distance of 50 units. Then set the sketch plane to a vertical face. After that, construct the second sketch as shown in Figure 3.361.

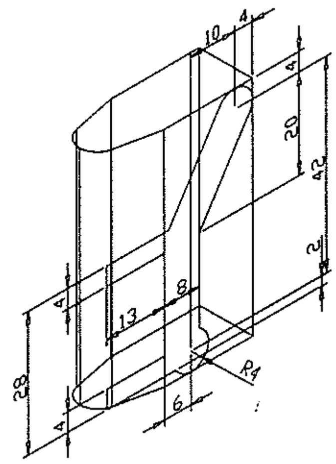


Figure 3.361 First sketch extruded, second sketch constructed

Extrude the second sketch to intersect the base sketch feature. Then set the sketch plane to another vertical face. After that, construct the third sketch as shown in Figure 3.362.

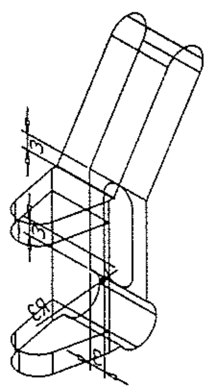


Figure 3.362 Second sketch extruded to intersect, third sketch constructed

Extrude the third sketch to cut through the base solid. Then set the sketch plane to an inclined face. After that, construct the fourth sketch. (See Figure 3.363.)

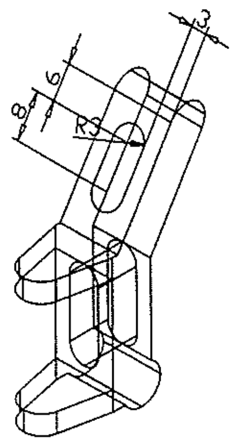


Figure 3.363 Third sketch extruded to cut, fourth sketch constructed

Extrude the fourth sketch to cut through the base solid. Then place four through holes as shown in Figure 3.359. The solid part is complete. Save your drawing.

<File> **<Save>**

File name: **Ubracket.dwg**

Exercise 3.14

Figure 3.364 shows the drawing of a hexagonal block. It consists of a circular extrude base feature, a C'sink hole, and a hexagonal extrude solid.

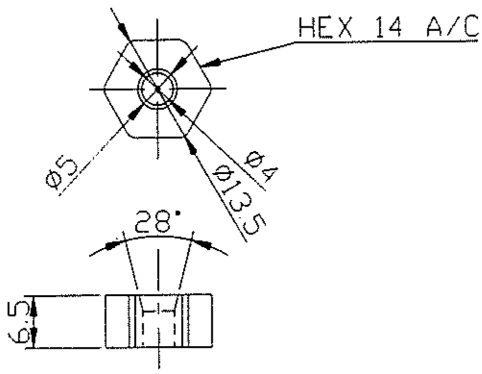


Figure 3.364 Hexagonal block

Construct a circle, resolve it, and add a parametric dimension. (See Figure 3.365.)

Figure 3.365 First sketch

Extrude the profile a distance of 6.5 units. Then place a concentric C'sink hole. (See Figure 3.366.)

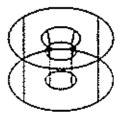


Figure 3.366 First sketch extruded, C'sink hole placed

As shown in Figure 3.367, construct a hexagon, resolve it to a profile, and add parametric dimensions.

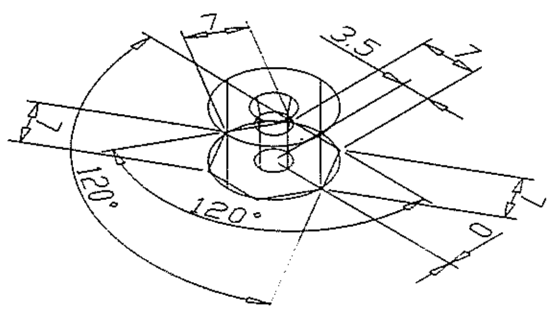


Figure 3.367 Hexagon constructed and constrained

Extrude the hexagon to intersect with the base solid feature. The model is complete. Save your drawing.

<File> **<Save>**

File name: **Hexblk.dwg**

Exercise 3.15

Figure 3.368 shows a drawing of the differential gear set casing for a scale model car. You will use the table-driven solid part Gear20t.dwg as the basis for making this solid part.

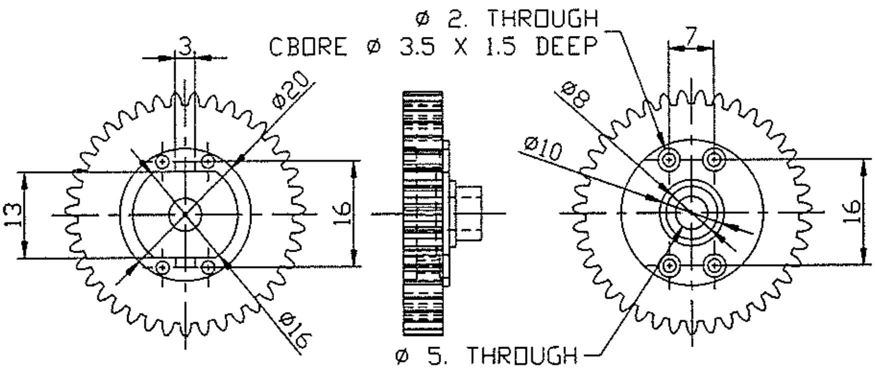


Figure 3.368 Differential gear set casing

Open the drawing Gear20t.dwg that you saved earlier in this chapter. Then save it to a new file name, Dif39t.dwg.

<File> **<Open...>**

File name: **Gear20t.dwg**

<File> **<Save As...>**

File name: **Dif39t.dwg**

As shown in Figure 3.369, select the 39teeth09 item in the Excel table and update the solid part. Then delete the last extrude solid feature. (See Figure 3.370.)

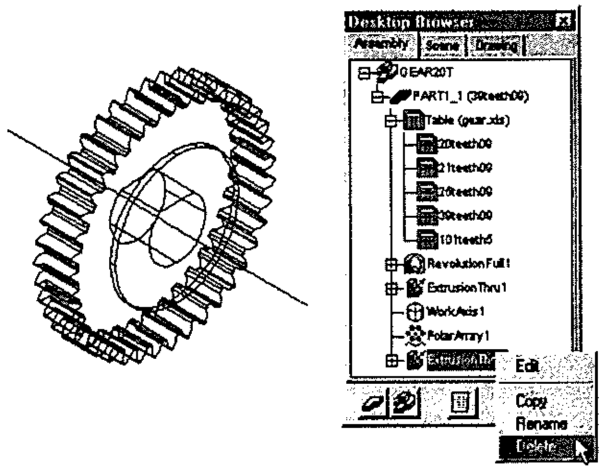


Figure 3.369 Thirty-nine teeth selected and updated

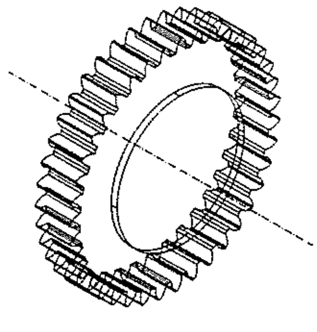


Figure 3.370 Extrude feature deleted

Set the sketch plane to face A (Figure 3.371). Construct a concentric circular profile D (Figure 3.371) of diameter 8 units and extrude it a distance of 12 units from face A to join the base solid. Then construct another concentric circular profile C (Figure 3.371) of diameter 10 units and extrude it a distance of 8 units from face A to join the base solid. After that, construct a concentric circular profile B (Figure 3.371) with diameter 20 units and extrude it to cut a depth of 2 units.

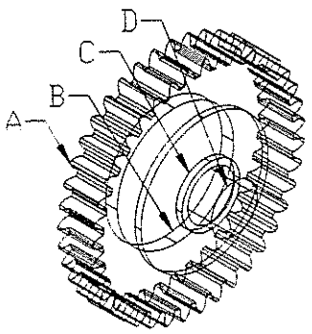


Figure 3.371 Three extrude features cut

Set the display to left isometric. Then place a concentric drilled through hole of diameter 5 units. (See Figure 3.372.)

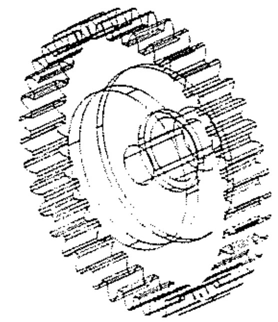


Figure 3.372 Concentric through hole placed

As shown in Figure 3.373, set the sketch plane to face A (Figure 3.373), construct a rectangle, resolve it to a profile, and constrain the profile.

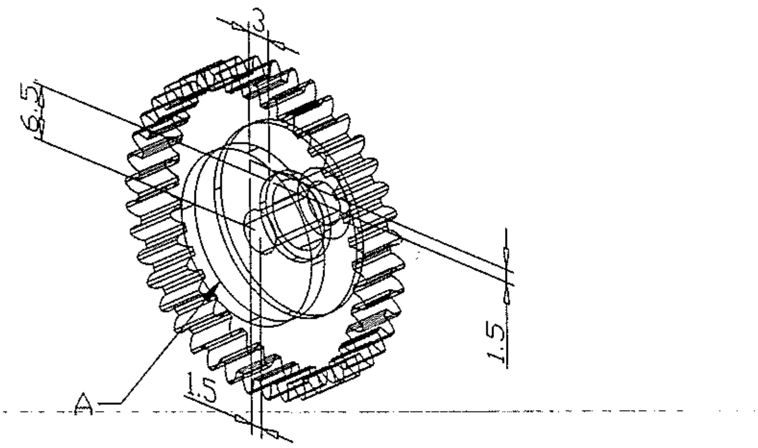


Figure 3.373 Rectangle constructed, resolved, and constrained

Extrude the profile a distance of 2 units to join the base solid. Then construct a profile as shown in Figure 3.374.

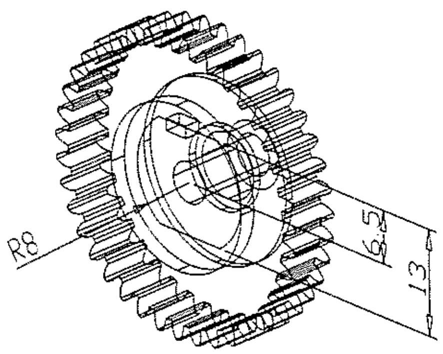


Figure 3.374 Rectangle extruded and joined, profile constructed

Extrude the profile a distance of 3 units to cut the base solid. Then polar-array feature A (Figure 3.375) to make it feature B (Figure 3.375).

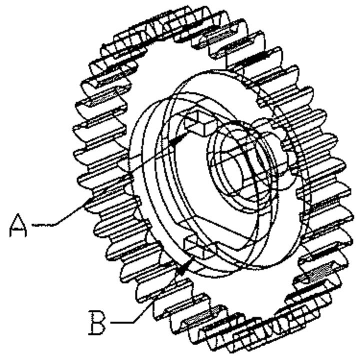


Figure 3.375 Profile extruded and cut, feature arrayed

To complete the model, place four C'bore through holes of diameter 2 units, C'depth 1.5, and C'Dia 3.5 as shown in Figure 3.368. Save your drawing.

<File> **<Save>**

Exercise 3.16

Figure 3.376 shows a drawing of the differential casing for a scale model car.

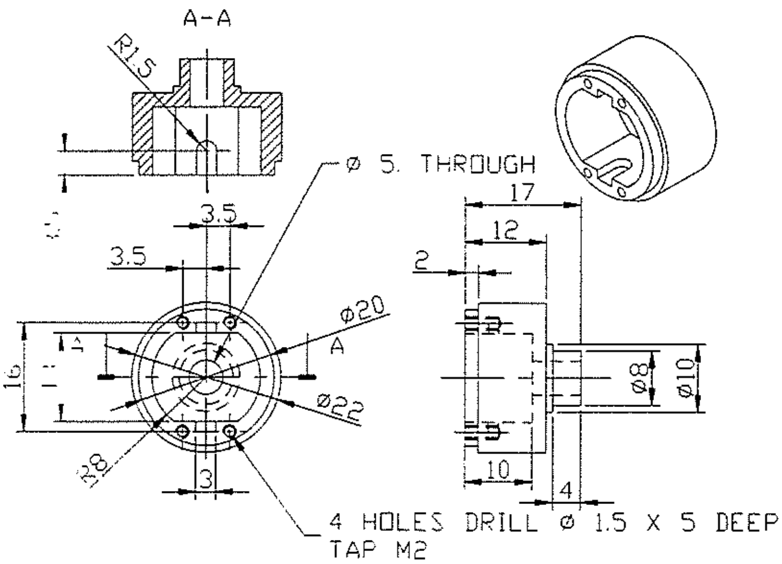


Figure 3.376 Differential casing

Start a new drawing and construct a profile as shown in Figure 3.377.

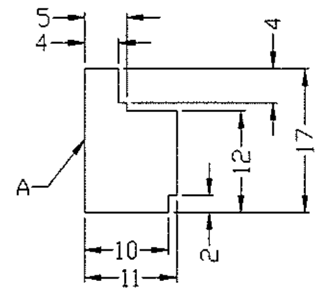


Figure 3.377 Profile constructed on default XY plane

Revolve the profile about A (Figure 3.377). Then set the display to an isometric view. After that, set the sketch plane to the end face and construct a profile as shown in Figure 3.378.

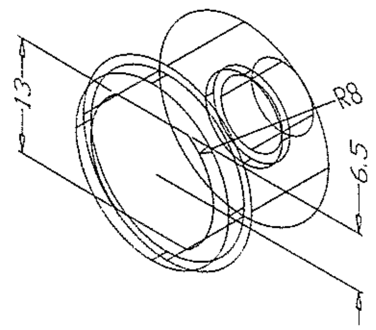


Figure 3.378 Profile revolved, display set, sketch plane set, and second profile constructed

Extrude the profile a distance of 10 units to cut the base solid. Then place a concentric through hole with a diameter of 5 units. (See Figure 3.379.)

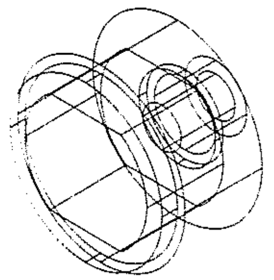


Figure 3.379 Second profile extruded and cut, hole placed

Set the sketch plane to worldxy. Then construct a profile as shown in Figure 3.380.

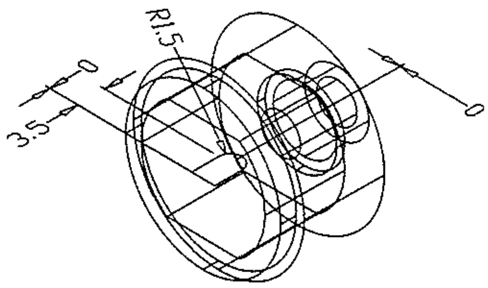


Figure 3.380 Sketch plane set, profile constructed

Extrude the profile from midplane for a distance of 16 units to cut the base solid. (See Figure 3.381.)

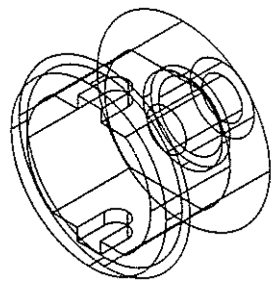


Figure 3.381 Third profile extruded midplane to cut the base solid

To complete the model, place four tapped blind holes, using the dimensions shown in Figure 3.376. (See Figure 3.382.)

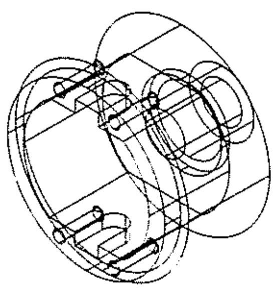


Figure 3.382 C'bore holes placed

Save your drawing.

<File> **<Save>**

File name: **Difcase.dwg**

Exercise 3.17

Figure 3.383 shows a drawing of the drive shaft for a scale model car.

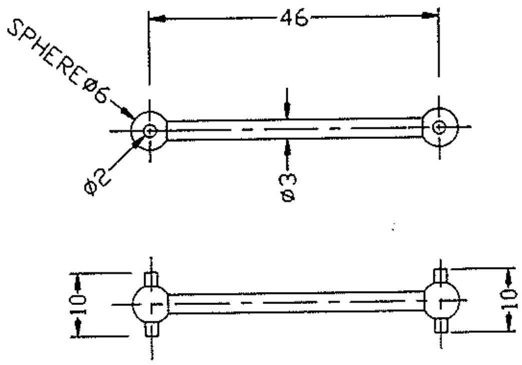


Figure 3.383 Drive shaft

As shown in Figure 3.384, construct a circular profile of diameter 2 units and extrude it a distance of 10 units from mid-plane. Then construct another profile.

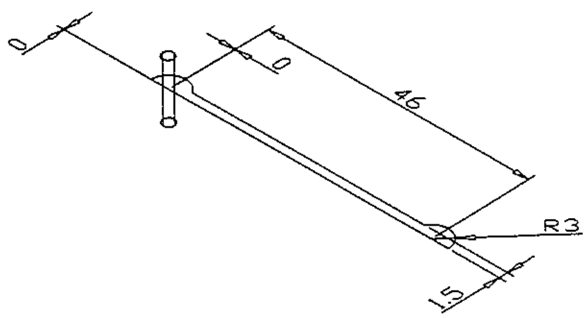


Figure 3.384 Circular profile extruded, second profile constructed

Revolve the profile and join the result with the base solid. Then construct the third profile. (See Figure 3.385.)

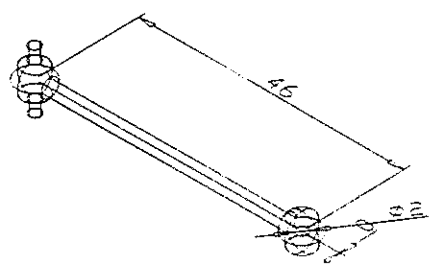


Figure 3.385 Profile constructed

Extrude the third profile a distance of 10 units from midplane to join. The model is complete. Save your drawing.

<File> **<Save>**

File name: **Dshaft.dwg**

Exercise 3.18

Figure 3.386 shows a drawing of the suspension hinge pin for a scale model car.

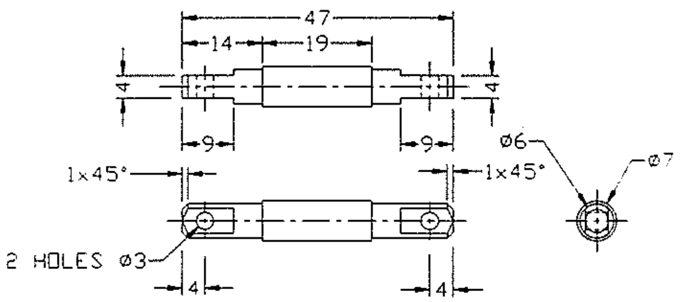


Figure 3.386 Hinge pin

Start a new drawing and construct a profile as shown in Figure 3.387.

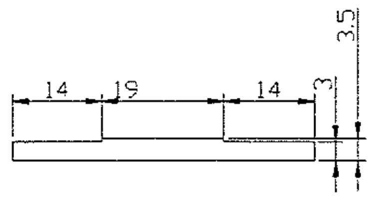


Figure 3.387 Profile constructed

Set the display to an isometric view. Then revolve the profile into a solid. After that, chamfer two edges, set the sketch plane to the end face, and construct a profile. (See Figure 3.388.)

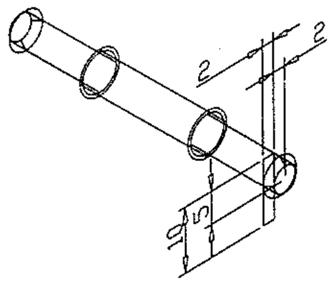


Figure 3.388 Profile revolved, chamfers placed, sketch plane set, and profile constructed

Extrude the profile a distance of 9 units to cut the base solid. (See Figure 3.389.)

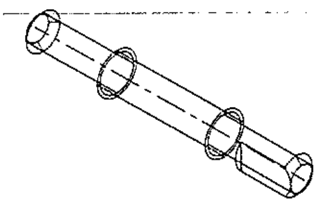


Figure 3.389 Second profile extruded and cut

Set the sketch plane to worldxy. Then array the second sketched solid feature. After that, set the sketch plane to the vertical face and construct a profile. (See Figure 3.390.)

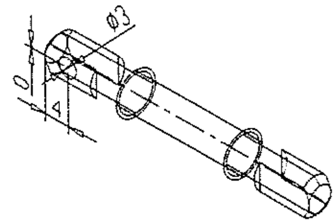


Figure 3.390 Solid feature arrayed, circular profile constructed

Extrude the circular profile. Then make an array. The model is complete.

<File> **<Save>**

File name: **Hinge.dwg**

Exercise 3.19

Figure 3.391 shows a drawing of the front hub for a scale model car.

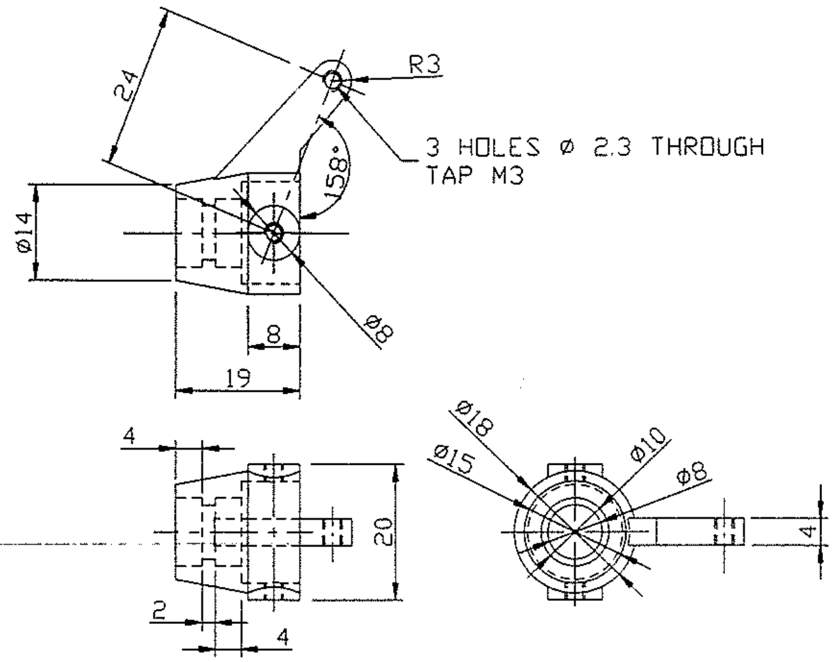


Figure 3.391 Front hub

Start a new drawing and construct a profile as shown in Figure 3.392.

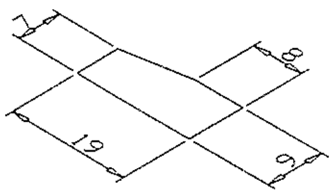


Figure 3.392 Profile constructed

Revolve the profile to create a solid. (See Figure 3.393.)

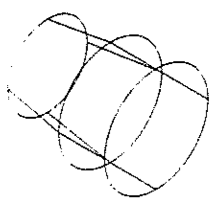


Figure 3.393 Profile revolved

Construct the second profile as shown in Figure 3.394.

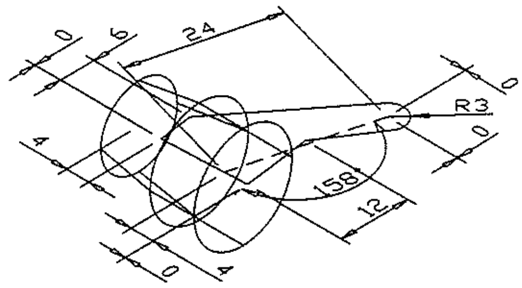


Figure 3.394 Second profile constructed

Extrude the profile a distance of 4 units from midplane to join. (See Figure 3.395.)

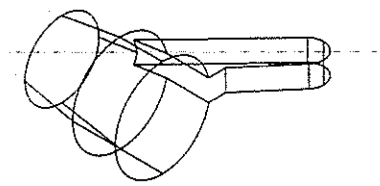


Figure 3.395 Profile extruded and joined

Construct a circular profile. (See Figure 3.396.)

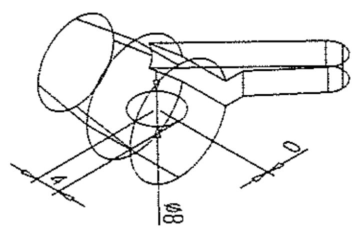


Figure 3.396 Circular profile constructed

Extrude the profile a distance of 20 units from midplane to join. (See Figure 3.397.)

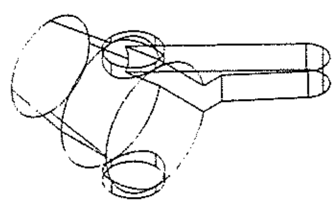


Figure 3.397 Circular profile extruded from midplane

Refer to the dimensions shown in Figure 3.391 and construct a profile as shown in Figure 3.398.

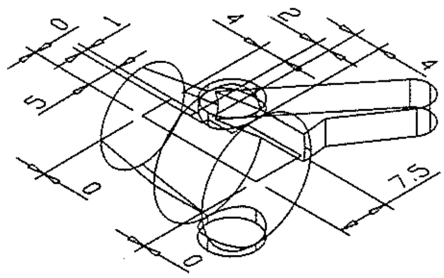


Figure 3.398 Profile constructed

Revolve the profile to cut the solid. (See Figure 3.399.)

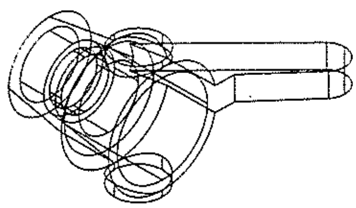


Figure 3.399 Profile revolved and cut

Place three tapped holes as shown in Figure 3.391. Save your drawing.

<File> **<Save>**

File name: **Hub_f.dwg**

Exercise 3.20

Figure 3.400 shows a drawing of the rear hub for a scale model car.

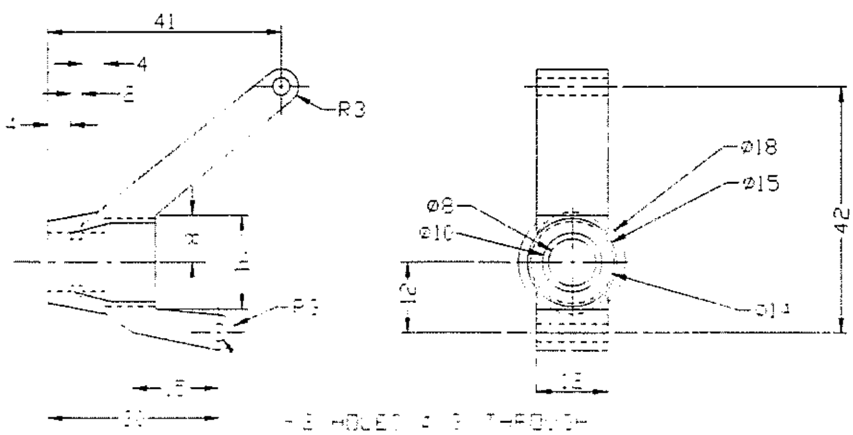


Figure 3.400 Rear hub

Start a new drawing and construct a profile as shown in Figure 3.401.

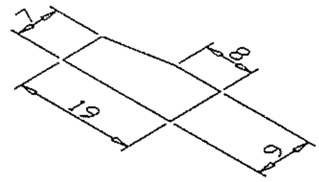


Figure 3.401 Profile constructed

Revolve the profile to create a solid. (See Figure 3.402.)

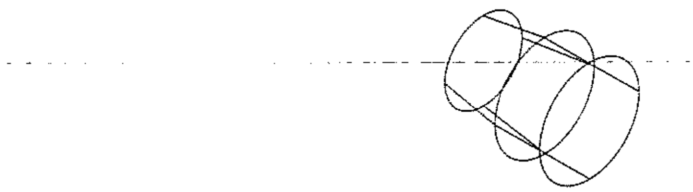


Figure 3.402 Profile revolved

Construct a profile as shown in Figure 3.403.

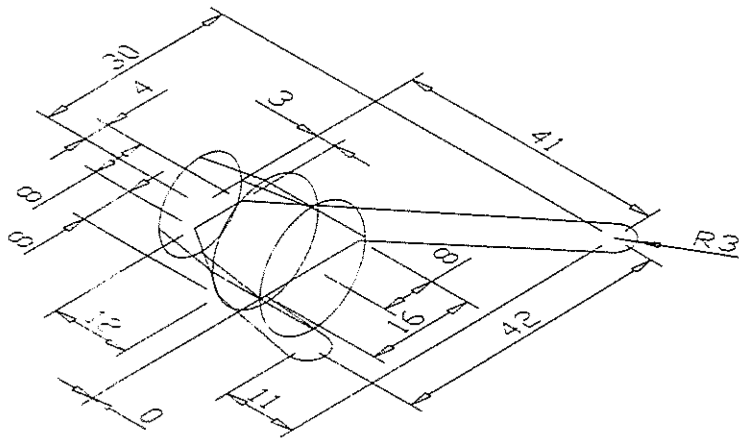


Figure 3.403 Second profile constructed

Extrude the profile a distance of 12 units from midplane. (See Figure 3.404.)

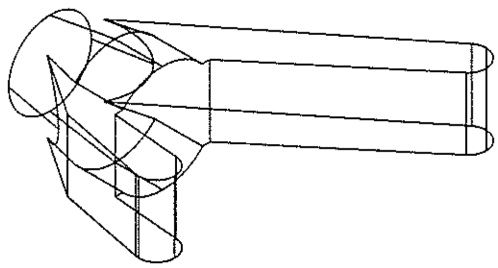
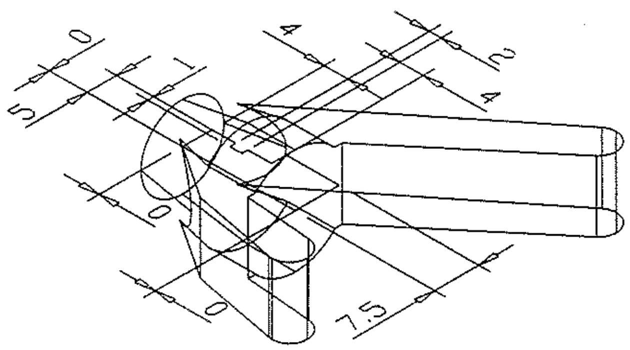


Figure 3.404 Second profile extruded and joined

Refer to the dimensions shown in Figure 3.400 and construct the third profile as shown in Figure 3.405.



Third profile constructed

Revolve the profile to cut the base solid. (See Figure 3.406.)

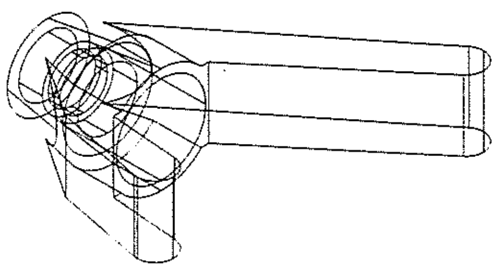


Figure 3.406 Profile revolved and cut

To complete the model, add two drilled through holes. After that, save your drawing.

<File> **<Save>**

File name: **Hub_r.dwg**

Exercise 3.21

Figure 3.407 shows a drawing of the first intermediate shaft for a scale model car. This solid part has two features, a revolve solid and an extrude solid.

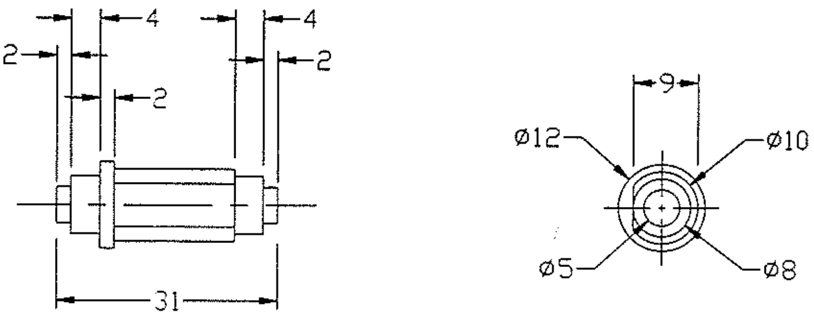


Figure 3.407 First intermediate shaft

Start a new drawing and construct a profile as shown in Figure 3.408.

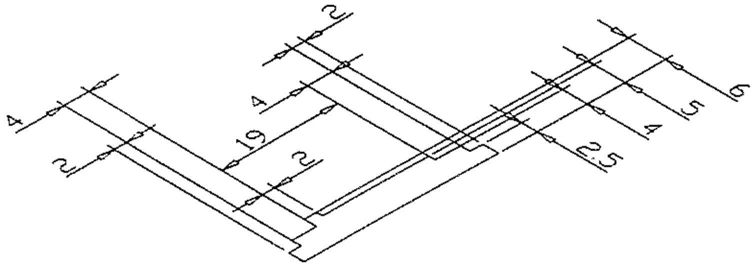


Figure 3.408 Profile constructed

Revolve the profile. Then set the sketch plane to a vertical face and construct a profile as shown in Figure 3.409.

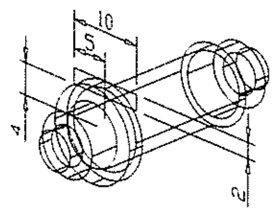


Figure 3.409 Profile revolved, sketch plane set, second profile constructed

Extrude the second profile to cut through the base solid. Save your drawing.

<File> **<Save>**

File name: **Intersa.dwg**

Exercise 3.22

Figure 3.410 shows a drawing of the second intermediate shaft for a scale model car.

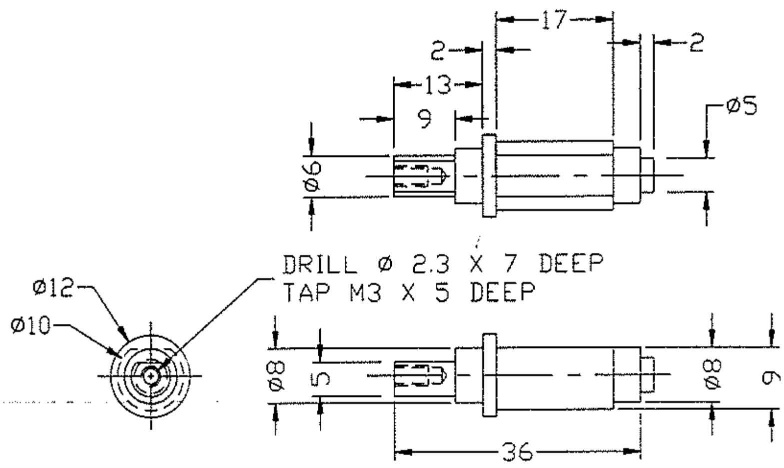


Figure 3.410 Second intermediate shaft

Start a new drawing and construct a profile as shown in Figure 3.411.

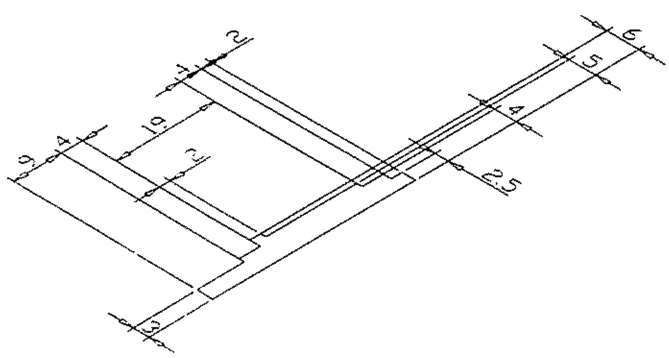


Figure 3.411 Profile constructed

Revolve the profile. Then place a concentric tapped hole. (See Figure 3.412.)

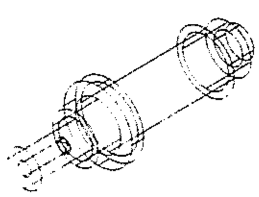


Figure 3.412 Profile extruded, tapped hole placed

As shown in Figure 3.413, construct a profile.

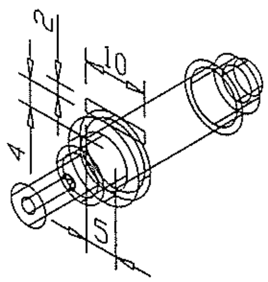


Figure 3.413 Second profile constructed

Extrude the profile to cut the base solid. Then construct another profile. (See Figure 3.414.)

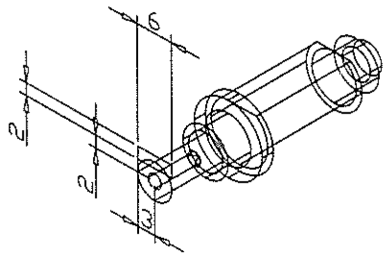


Figure 3.414 Profile extruded and cut, third profile constructed

Extrude and cut the base solid. The model is complete. Save your drawing.

<File> **<Save>**

File name: **Intersm.dwg**

Exercise 3.23

Figure 3.415 shows a drawing of the shaft joint for a scale model car.

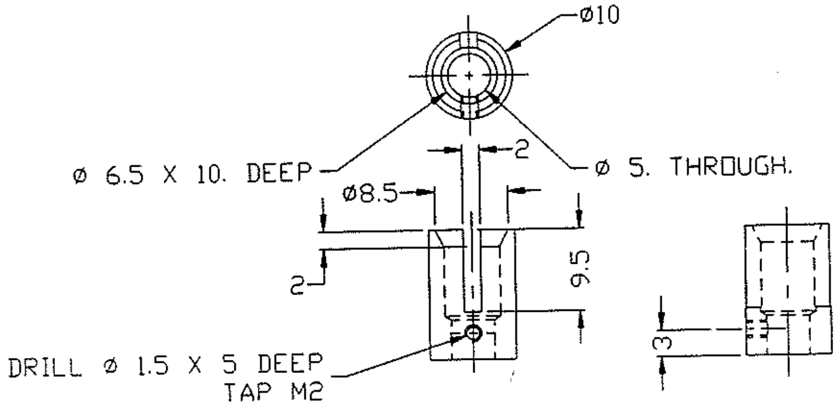


Figure 3.415 Shaft joint

Start a new drawing and construct a rectangle as shown in Figure 3.416.

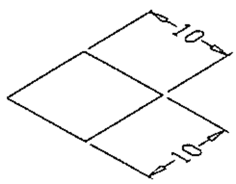


Figure 3.416 Rectangle constructed

Extrude the profile a distance of 15 units. Then place a blind tapped hole of size M2 with a depth of 5 units. The hole is 5 units from the vertical edge and 3 units from the base. (See Figure 3.417.)

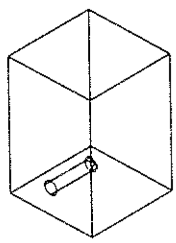


Figure 3.417 Profile extruded, tapped hole placed

Construct a circular profile as shown in Figure 3.418.

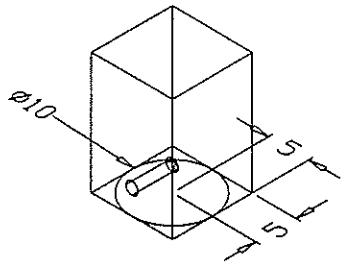


Figure 3.418 Circular profile constructed

Extrude the circular profile to intersect with the base solid. Then place a concentric through hole (diameter 5 units) and a concentric blind hole (diameter 6.5 units and depth 10 units). (See Figure 3.419.)

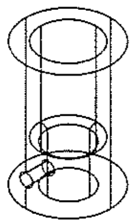


Figure 3.419 Circular profile extruded and intersected, holes placed

Place a chamfer feature of 1 unit by 2 units. Then construct a profile. (See Figure 3.420.)

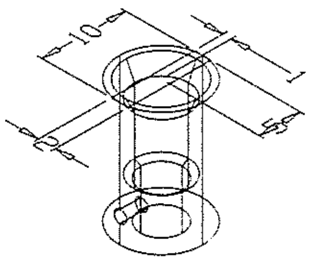


Figure 3.420 Chamfer feature placed, profile constructed

Extrude the profile a distance of 9.5 units to cut the solid. Then save your drawing.

<File> <Save>

File name: **Joint.dwg**

Exercise 3.24

Figures 3.421 and 3.422 show drawings of the tire and the wheel.

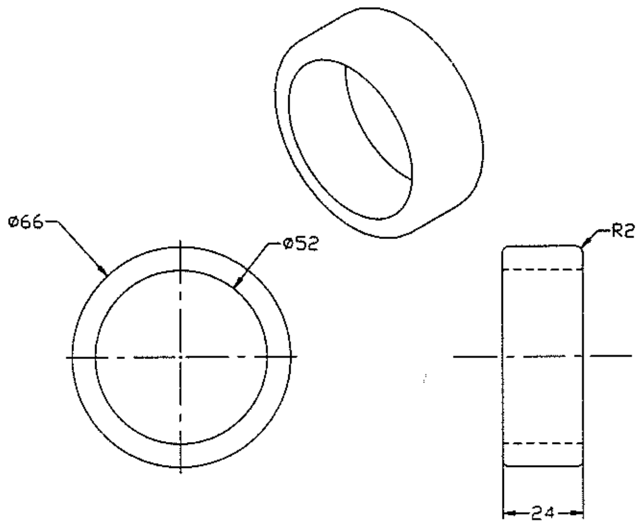


Figure 3.421 Tire

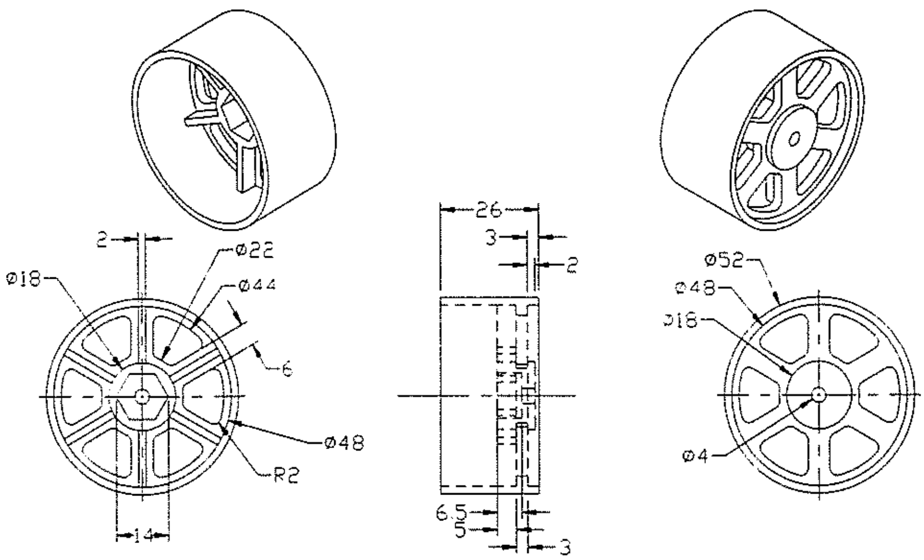


Figure 3.422 Wheel

Start a multi-part drawing and construct the solid parts using the dimensions shown in Figures 3.421 and 3.422.

<File> **<Save>**

File name: **Whl_tr.dwg**

Exercise 3.25

Figure 3.423 shows the solid parts of the shock absorber of a scale model car. It has eleven parts. Two of them, the helical spring and the shock tube, were constructed earlier in this chapter. They are saved as Spring.dwg and Shock.dwg. The drawings for the remaining parts are shown in Figures 3.424 through 3.432. Start a new multi-part drawing and construct all the solid parts. Then save the drawing.

<File> **<Save>**

File name: **Shocks.dwg**

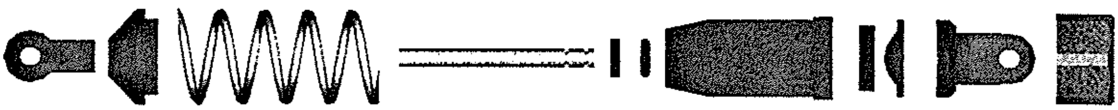


Figure 3.423 Shock absorber

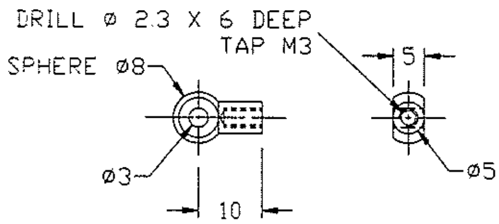


Figure 3.424 Ball end

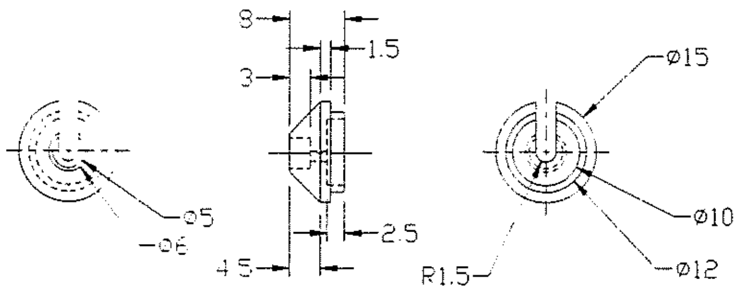


Figure 3.425 Spring seat

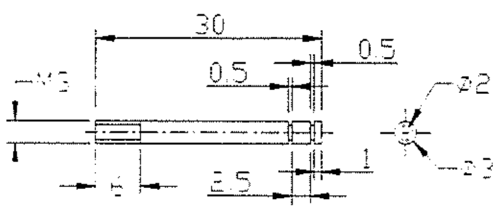


Figure 3.426 Rod

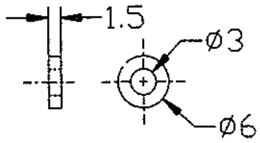


Figure 3.427 Bushing

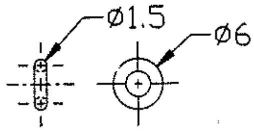


Figure 3.428 O-ring

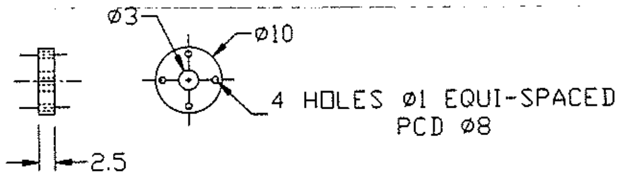


Figure 3.429 Restrictor

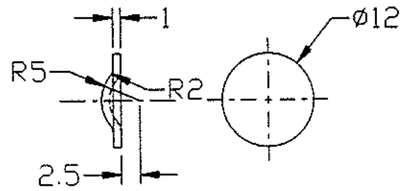


Figure 3.430 Rubber diaphragm

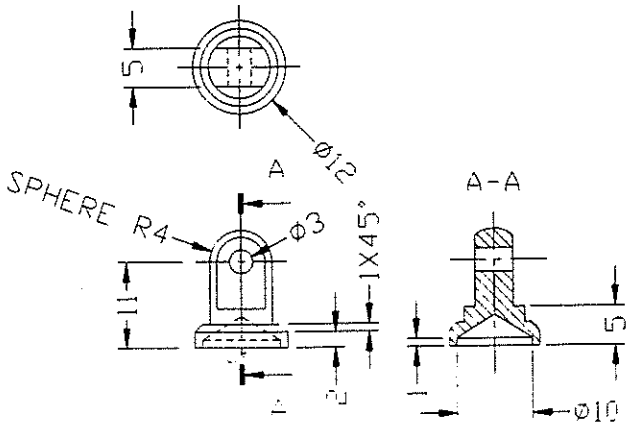


Figure 3.431 End piece

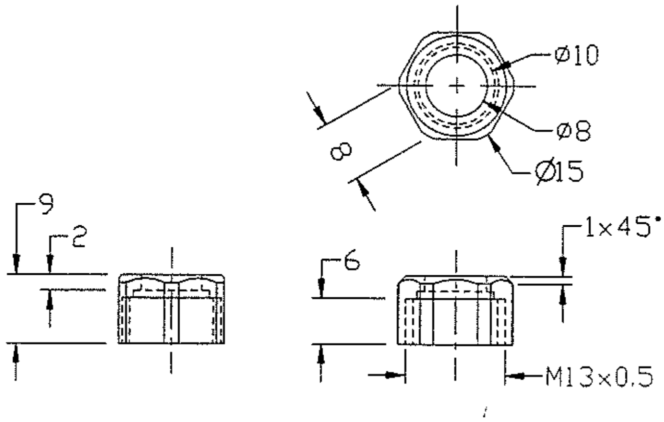


Figure 3.432 Screw cap

Exercise 3.26

Figure 3.433 shows the parts for the turning arm of a hand press. It has four solid parts: the central boss, the hand lever, the extension arm, and the weight support. Figures 3.434 through 3.437 show their dimensions. Start a new multi-part drawing and construct the four solid parts. Then save the drawing.

<File> **<Save>**

File name: **Tarm.dwg**

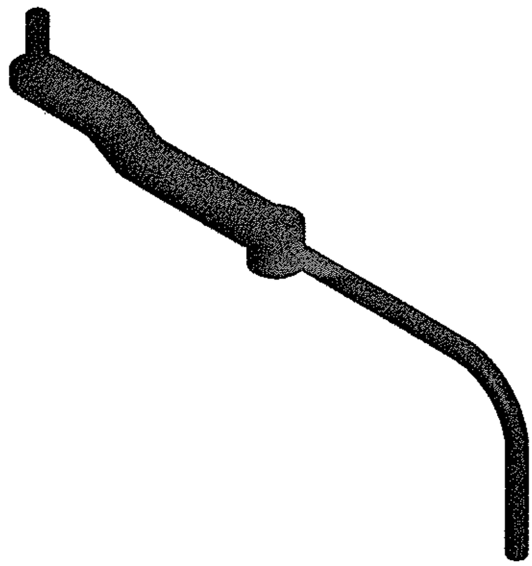


Figure 3.433 Parts of the hand press turning arm

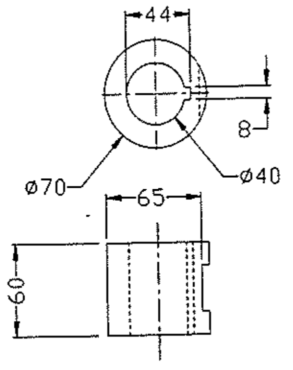


Figure 3.434 Central boss

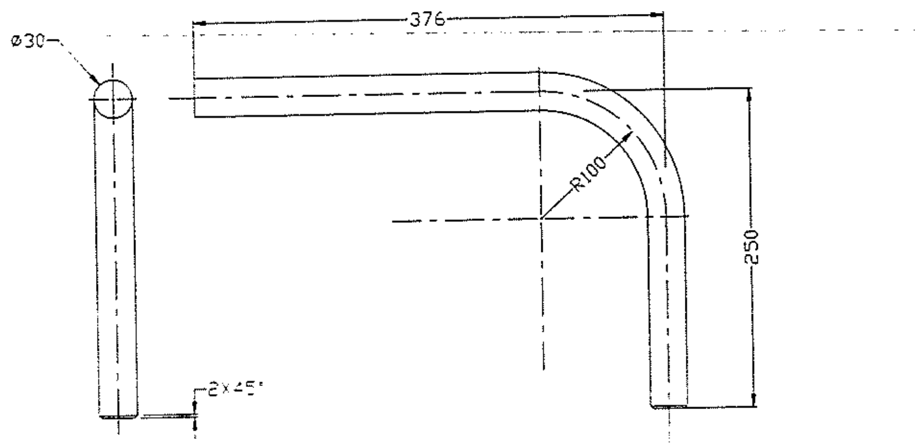


Figure 3.435 Hand lever

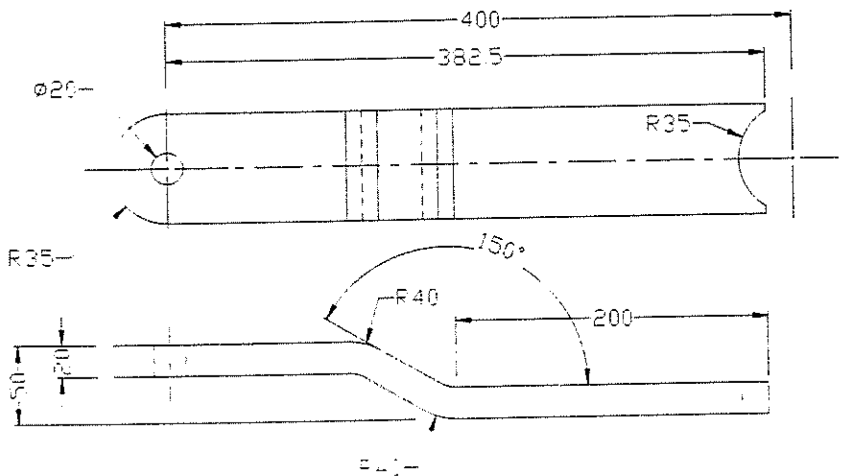


Figure 3.436 Extension arm

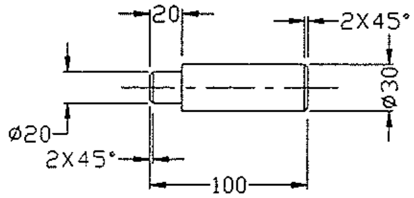


Figure 3.437 Weight support

Exercise 3.27

Figure 3.438 shows the rendered image of the lead screw of a hand press. This is a revolve solid with a polar array of two helical grooves. Start a new drawing and construct a profile as shown in Figure 3.439.

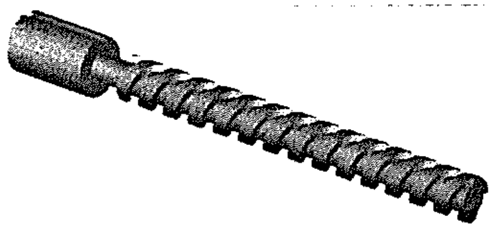


Figure 3.438 Lead screw

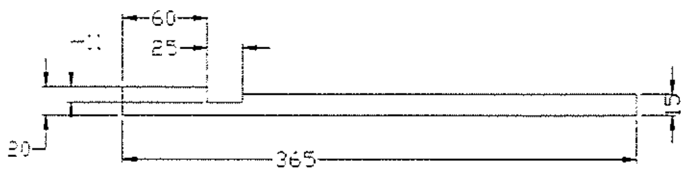


Figure 3.439 Profile constructed

Set the display to isometric and revolve the profile to create a solid. (See Figure 3.440.)

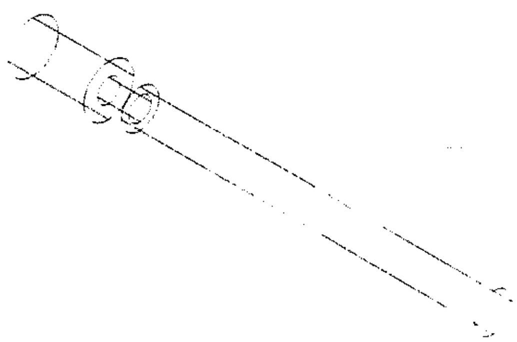


Figure 3.440 Profile revolved

Construct a rectangular profile of width 10 units and height 5 units, as shown in Figure 3.441. Then set the sketch plane to the end face. After that, use the AM3DPATH command to construct a helical path (Revolution: 7.5, Pitch: 40, Diameter: 30, Start Angle: 90). (See Figure 3.441.)

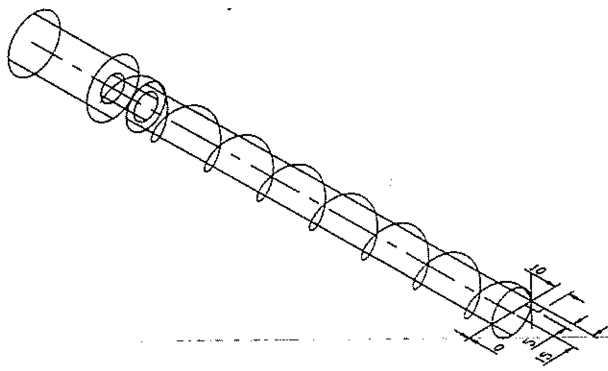


Figure 3.441 Rectangular profile and 3D helical path constructed

Sweep the rectangular profile along the 3D helical path to cut a helical groove. (See Figure 3.442.)

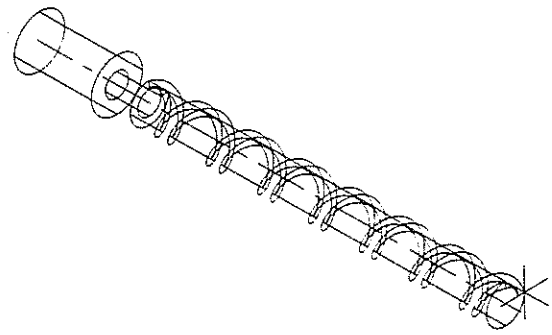


Figure 3.442 Helical groove cut

Use the AMARRAY command to construct a polar array of two instances of the helical groove. (See Figure 3.443.)

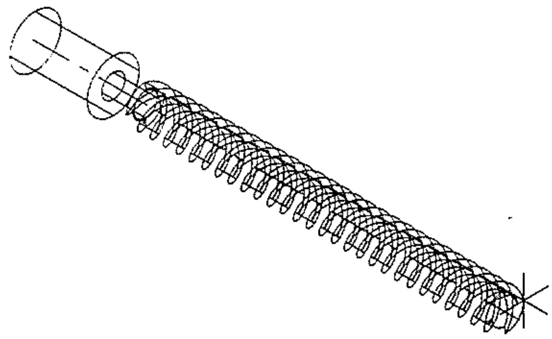


Figure 3.443 Helical groove polar-arrayed

Set the display to the left isometric view. Then set the sketch plane to the other end. After that, construct a profile as shown in Figure 3.444.

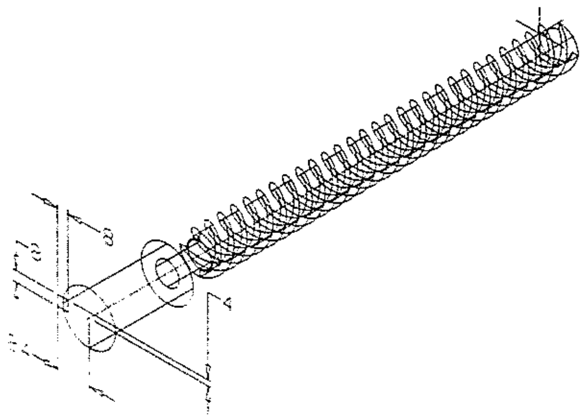


Figure 3.444 Display set, profile constructed

Extrude the profile a distance of 60 units to cut the base solid. Then save your drawing.

<File> **<Save>**

File name: **Ldscrew.dwg**

Exercise 3.28

Figure 3.445 shows a drawing for the front panel of a personal computer casing. Start a new drawing and construct a profile as shown in Figure 3.446.

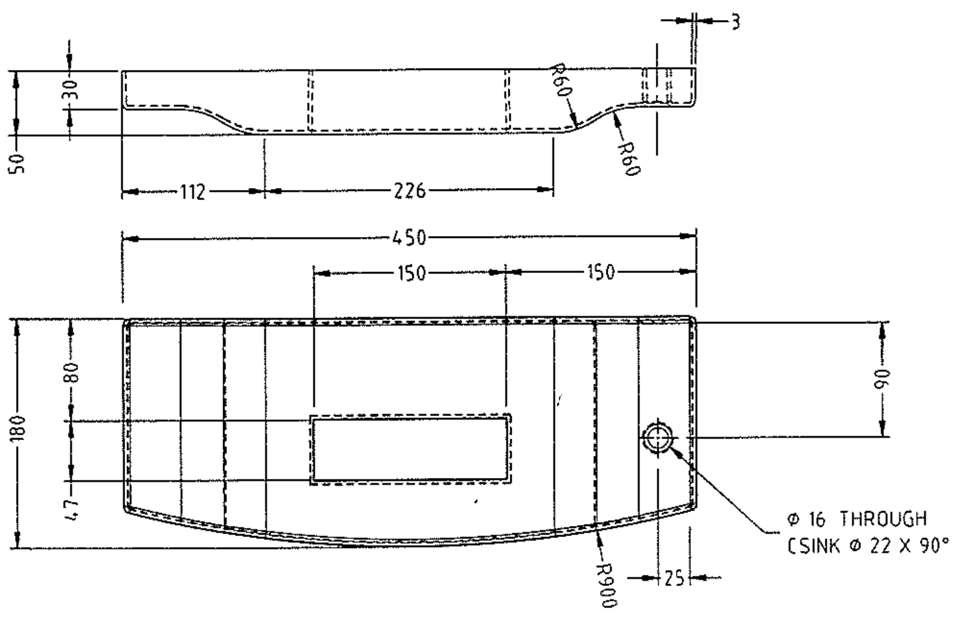


Figure 3.445 Front panel of a personal computer

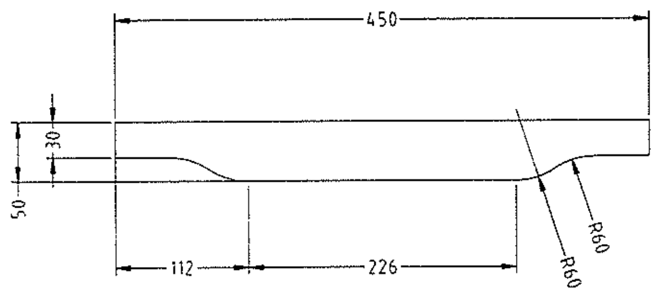


Figure 3.446 Profile constructed

Extrude the profile a distance of 180 units. Then, in accordance with the outer profile of the front view in Figure 3.445, construct a profile, then extrude it a distance of 50 units and a draft angle of −1° to intersect with the base solid.

Fillet the edges a radius of 5 units. Then construct a rectangular profile to cut a rectangular slot, and place a C'sink hole.

To complete the model, use the AMSHELL command to cut a thin shell with a thickness of 3 units. Exclude the back vertical face. After that, save your drawing.

<File> **<Save>**

File name: **Panel.dwg**

Chapter 4

Assembly Modeling

Aims and Objectives

The aims of this chapter are to introduce the concepts of assembly modeling, to outline different design approaches to constructing a set of solid parts for an assembly, to show ways to apply assembly constraints to a set of solid parts to assemble them together, to illustrate techniques for setting up exploded assembly scenes, and to familiarize you with the uses of various assembly modeling utilities. After studying this chapter, you should be able to

- Describe the key concepts of assembly modeling
- Know different design approaches to constructing a set of solid parts
- Apply assembly constraints to a set of solid parts in an assembly
- Set up assembly scenes and exploded views
- Use Mechanical Desktop as a tool to construct 3D assemblies of 3D solid parts

Overview

Take a look around your desk. With the exception of very simple objects, such as a ruler, most objects have more than one component part. For example, a pencil has two parts: the wooden body and the graphite core. When you are designing a set of component parts, the relative dimensions and positions of the parts, and how they fit together, are crucial. You need to know whether there is any interference among the mating parts. If there is any interference, you need to find out where it occurs. Then you can eliminate it. To shorten the design lead time, you can construct virtual assemblies in the computer to validate the integrity of a set of solid parts.

Making virtual assemblies in a computer is far beyond translating solid parts together in 3D space. It involves constraining the features of a solid part in relation to the features of another solid part. After constructing virtual assemblies, you can check interference among the solid parts and generate exploded assembly views by setting up assembly scenes.

In Chapter 3, you learned how to construct parametric solid models from sketches, and you constructed a number of solid parts. In this chapter, you will learn about assembly modeling, the techniques of managing and assembling a set of solid parts, and ways to set up assembly scenes in which you can explode assembled solids apart.

In the next chapter, you will learn how to generate associative engineering drawings from 3D surface models, 3D solid parts, and 3D assemblies.

4.1 Assembly Modeling Concepts

A virtual assembly has to do with a collection of solid parts in 3D space (in the computer) that are associated together by a set of assembly constraints. While the geometric constraints we studied in the last chapter restrict the forms and shapes of the sketches of a solid feature, the assembly constraints here concern the way the features of one solid part are associated with the features of another.

Assembling solid parts in a virtual assembly is a process of applying assembly constraints to paired solid parts. (See Figure 4.1.)

Figure 4.1 Solid parts associated together by applying assembly constraints

Assembly Constraints

There are four kinds of assembly constraints: mate, flush, angle, and insert. (See Figures 4.2 through 4.7.)

1. The mate constraint causes the points, lines/axes, or planes of a pair of solid parts to align with each other.
2. The flush constraint causes the planes of a pair of solid parts to align in the same direction.
3. The angle constraint causes the lines/axes or planes of a pair of solid parts to align angularly.
4. The insert constraint causes the circular edges of a pair of solid parts to be concentric and the planes defined by the circular edges to be coplanar.

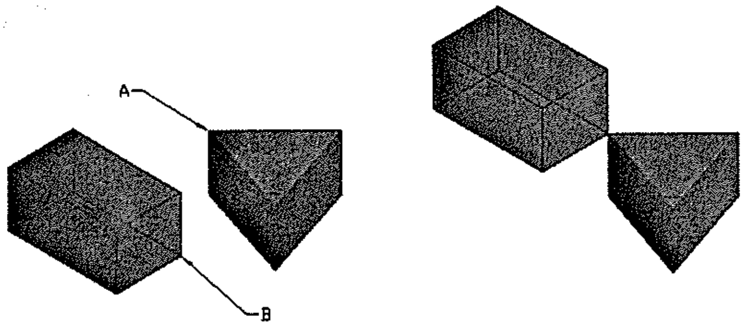


Figure 4.2 Mate constraint — point B is mated to point A

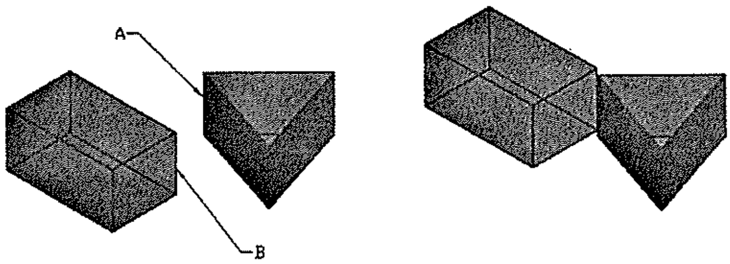


Figure 4.3 Mate constraint — line B is mated to line A

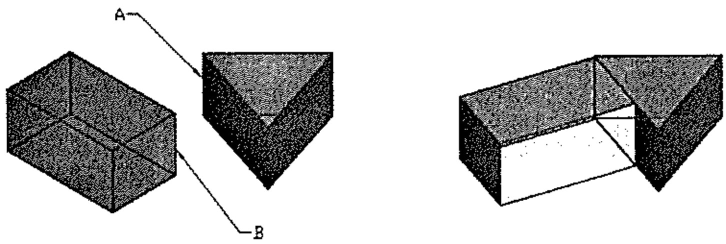


Figure 4.4 Mate constraint — vertical plane B is mated to vertical plane A

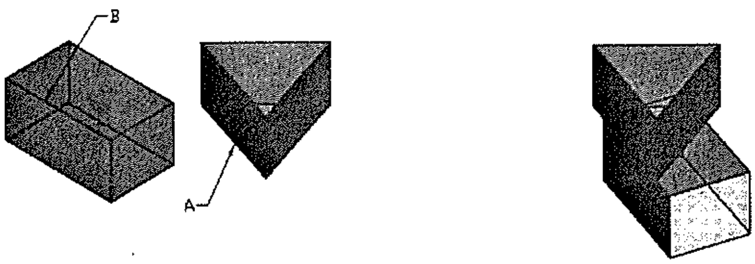


Figure 4.5 Flush constraint — vertical plane B is flush with vertical plane A

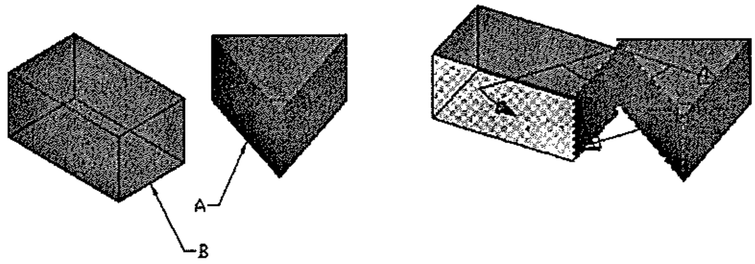


Figure 4.6 Angle constraint — plane B is set at an angle with plane A

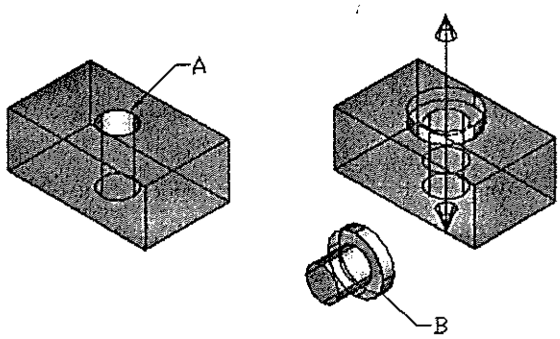


Figure 4.7 Insert constraint — circular edge B is inserted into circular edge A

Degrees of Freedom

In 3D space, a free object has six degrees of freedom. It can translate linearly in three directions and rotate about three axes. To find out the number of degrees of freedom of a solid part, you can display the degrees of freedom (DOF) symbol. (See Figure 4.8.)

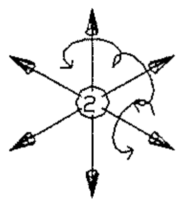


Figure 4.8 DOF symbol

The DOF symbol has three linear arrows, three rotational arrows, a circle, and a number. The linear arrows depict the three degrees of linear translation freedom. The rotational arrows show the three degrees of rotational freedom. The circle indicates the center of the geometry of the solid part. The number shows the order of the solid part in the hierarchy. The first solid part of the drawing is number 1.

With two or more solid parts in a drawing file, a hierarchy forms. The first solid part is said to be grounded. It has no degrees of freedom. Figure 4.9 shows the DOF symbol for the grounded solid part. It has a circle and a number.

Figure 4.9 DOF symbol for the grounded solid part

When you apply an assembly constraint to a pair of solids, the second solid of the drawing file translates toward the first solid of the drawing file. If there is a third solid part in the drawing, it translates toward the second solid or the first solid.

As you apply assembly constraints to a pair of solids, the number of degrees of freedom decreases. In Figure 4.10, a face of object A is mated to a face of object B. As a result, object A can no longer translate along the Y axis or rotate about the X and Z axes. However, it can still translate linearly along Y and Z, and rotate about Y. Thus, it has three degrees of freedom left.

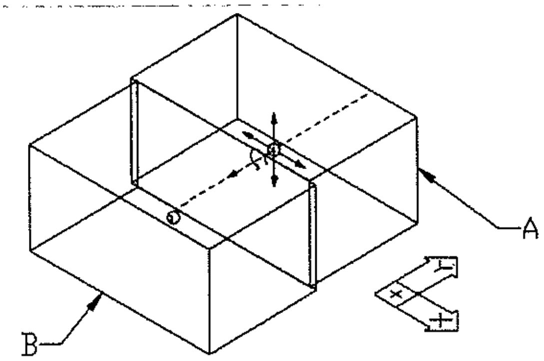


Figure 4.10 Three degrees of freedom left

Figure 4.11 shows another example. Here a circular edge of an object is inserted into a circular edge of another object. As a result, the inserted object can only rotate about the Z axis. It has one degree of freedom left.

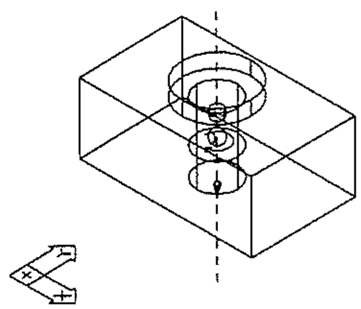


Figure 4.11 One degree of freedom left

External and Local Solid Parts

As we explained in the last chapter, there are two kinds of drawing files: single-part and multi-part. Naturally, an assembly drawing is a multi-part drawing. You can manage an assembly of solid parts in three ways: You can keep all of them in the multi-part drawing file, you can keep them in separate single-part or multi-part drawing files, and you can keep some solid parts in the multi-part drawing file and some in separate drawing files. (See Figure 4.12.)

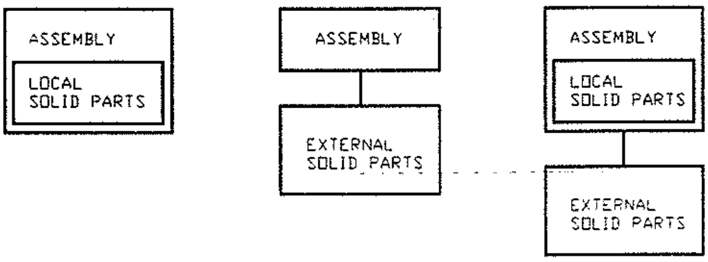


Figure 4.12 Three ways to manage an assembly of solid parts

In the first method, all the solids are stored in the same drawing file. They are called local solid parts. In the second method, all the solids are attached to the assembly drawing file. They are called external solid parts. The third method is a combination of the first and second methods. Here, some solid parts are local and some are external.

Local and external solid parts can be swapped. You can externalize a local solid part to make it an individual drawing that is attached to the current drawing as an external solid part. On the other hand, you can localize an external solid part by copying the solid definition to the current drawing; this makes it a local solid part. (See Figure 4.13.)

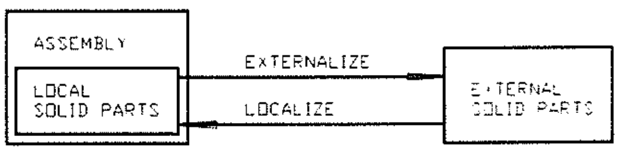


Figure 4.13 Localizing and externalizing the solid parts

Instance and Solid Parts

In an assembly environment, the objects are instances of solid parts that are either local or external.

Scene and Explode Views

After a set of solid parts is properly assembled together, you can set up a number of scenes. This way, you can explode the solid parts apart in different ways.

Assembly and Subassemblies

To manage a large number of solid parts, you can first form subassemblies of small collections of solid parts and then put the subassemblies into a single assembly. Therefore, an assembly can be a collection of solid parts and/or a collection of subassemblies.

Utilities

With an assembly, you can check the solid parts for fitness and interference. If there is any interference, you can determine its location. Thus, you can focus on the interfering portions of the solid parts and make necessary modifications.

Apart from interference checking, you can evaluate the mass properties and generate a bill of materials for production purposes.

4.2 Assembly Modeling Preferences

Before working on the projects in this chapter, set the system preferences that are related to assembly modeling by selecting the Assembly Preferences... item of the Assembly pull-down menu. See Figure 4.14, the Assemblies tab of the Desktop Preferences dialog box. It has three major areas: Automatic, Attach and Insert Parts, and Naming Prefixes.

The Automatic area has to do with view restoration and translation of parts. Checking the View Restore with Assembly Activation box restores the last display view when you change from one subassembly to another. Checking the Update Assembly as Constrained box causes the parts to update as the assembly constraints are applied.

In the Attach and Insert Parts area, you can control the insertion base point of an instance so that it is either the center of the geometry of the solid part or the absolute insertion point of the solid part.

<Assembly> **<Assembly Preferences...>**

Command: **AMPREFS**

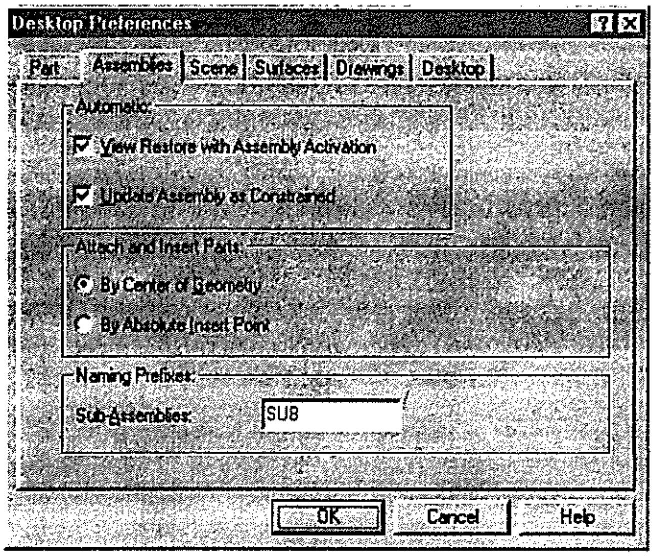


Figure 4.14 Assemblies tab of the Desktop Preferences dialog box

4.3 Design Approaches

As we have said, there are three ways to manage an assembly of solid parts. These ways come from three design approaches.

With the first approach, you construct all the solid parts in a single drawing file. The solid parts, being resident in the assembly drawing file, are called local solid parts. In terms of manufacturing, we need individual drawing files to produce the components of a product. Practically, you should externalize the local solid parts to individual drawing files because of manufacturing requirements. (See Figure 4.15.)

Therefore, this approach is called a top-down approach. By constructing all the solid parts in a single drawing file, you can have a better perception of their relative shapes and sizes. However, it may not be feasible to use this approach to construct an assembly that has a large number of parts because the drawing file will become very large.

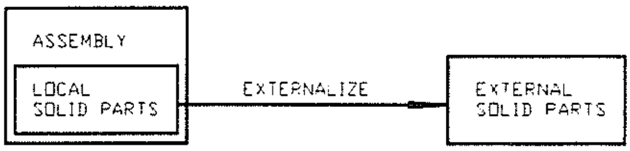


Figure 4.15 Top-down approach

With the second approach, you construct all the solid parts in separate drawing files. Then you start a new multi-part drawing and attach all the solid parts to it. Because the solid parts of the assembly are external to the assembly drawing file, they are called external solid parts. Here you work from external drawings and then move up to the assembly drawing This approach is called down-top approach. Because the solid parts are external to the assembly drawing, you cannot edit them while working on the assembly drawing. To overcome this limitation, you can localize the external drawing. This way, the external solid part becomes local, and you can modify it without leaving the assembly drawing. After editing and modifying, you can externalize the solid part again. (See Figure 4.16.)

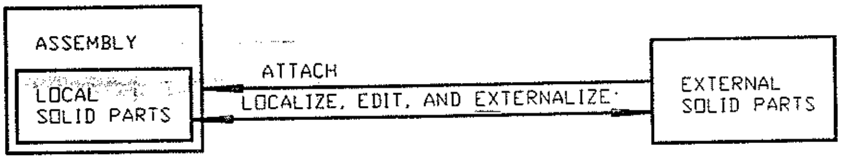


Figure 4.16 Down-top approach

With the third approach, some solid parts are constructed individually in separate drawing files and some are constructed in the assembly drawing. After that, you attach the external solid parts and assemble them together with the local solid parts. This is called the middle-way-out approach. As in the top-down approach, it is practical to externalize the local solid parts after completing the assembly. (See Figure 4.17.)

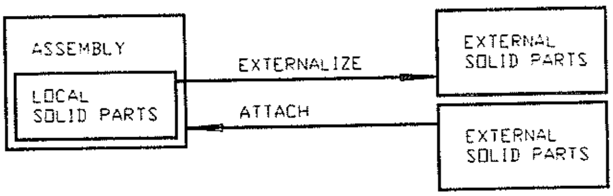


Figure 4.17 Middle-way-out approach

The choice of the approach you will use to handle a project depends on a number of factors. Here are a few:

- The number of solid parts involved in the assembly
- The use of standard engineering parts in the assembly
- The number of engineering designers involved in the project

No matter which approach you use, it is common engineering practice to have an assembly drawing and a set of individual solid parts attached to the assembly drawing.

4.4 Assemblies

Now you will work on two assembly projects: a remote control and a four-bar linkage. For the remote control project, you will use a down-top approach. For the four-bar linkage project, you will use a top-down approach.

Remote Control Project (Continued)

The remote control has two solid parts. You constructed them in separate single-part drawing files in the last chapter. Here you will start a new multi-part drawing file. Then you will attach the solid parts and assemble them together by applying assembly constraints. After assembling, you will learn how to localize the solid parts in order to edit them locally without having to open the original solid part files. After modifying, you will update the assembly. Because it is normal engineering practice to keep the solid part files in individual drawing files, you will externalize the solid parts to overwrite the original drawing files. On completion of this project, you will have an assembly drawing with attached external solid parts residing in individual drawing files.

Start a new multi-part drawing and use Solid.dwt as the template.

<File> **<New...>**

Template File: **Solid.dwt**

Select the Attach Part/Assembly item of the Desktop Browser (Figure 4.18) or select the Catalog item of the Assembly pull-down menu to use the AMCATALOG command. (See Figure 4.19.)

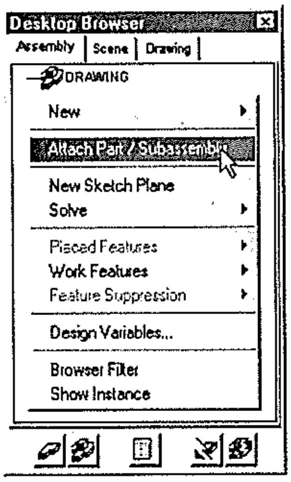


Figure 4.18 AMCATALOG command activated in the Desktop Browser

<Assembly> **<Catalog...>**

Command: **AMCATALOG**

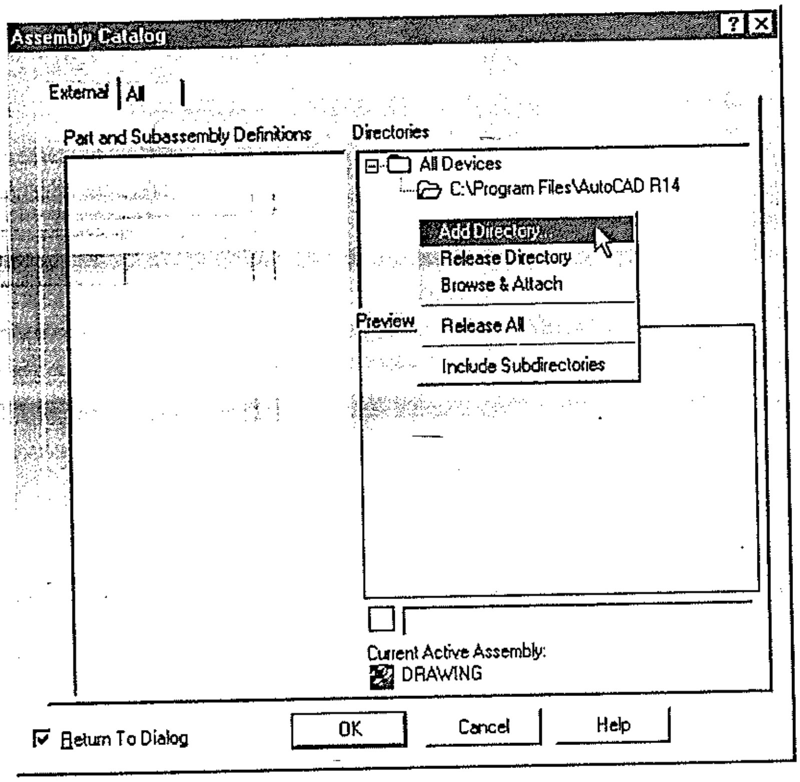


Figure 4.19 Add Directory... item in the Assembly Catalog dialog box

In the Assembly Catalog dialog box, select the External tab. Initially, the Part and Subassembly Definitions box is empty because the directories in which you saved the solid part drawing files are not included in the Directories box.

To add a directory, place the mouse cursor on the background of the Directories box and press the right mouse button. After that, select the Add Directory... item to bring up the Browse for Folder dialog box. (See Figure 4.20.)

Let us assume that the two solid part files are saved in the directory C:\Projects. Open this directory in the Browse for Folder dialog box. Then select the [OK] button.

When you return to the Assembly Catalog dialog box, the directory C:\Projects is added and the drawing files saved in that directory are displayed in the Part and Subassembly Definitions box. (See Figure 4.21.) In the Preview box, you will see a preview of a selected drawing. Now select the file UCASE and press the right mouse button.

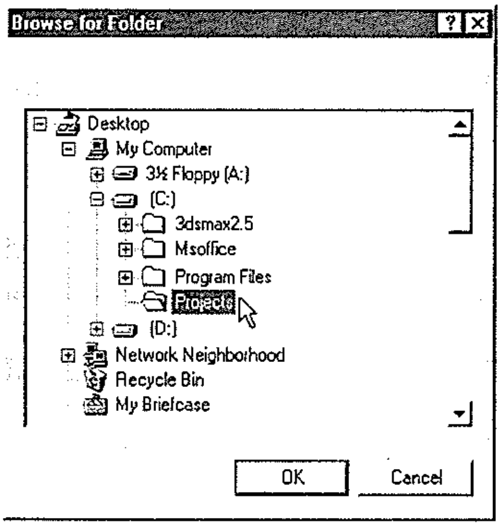


Figure 4.20 Browse for Folder dialog box

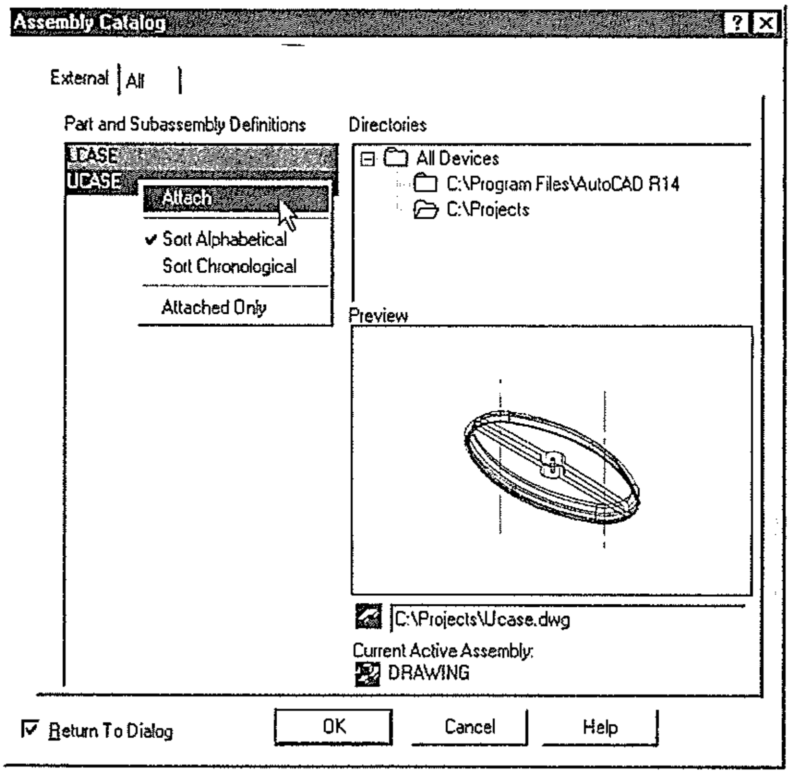


Figure 4.21 Directory added

Select the Attach item (Figure 4.21) to insert the selected solid part. Then select point A (Figure 4.22) on your screen. This will put an instance of the solid part into your drawing.

Select insertion point: **[Select A (Figure 4.22).]**
Select insertion point: **[Enter]**

On returning to the Assembly Catalog dialog box, select the file LCASE, then double-click. After that, select point B (Figure 4.22).

Select insertion point: **[Select B (Figure 4.22).]**
Select insertion point: **[Enter]**

After attaching two external solid parts, select the [OK] button to exit.

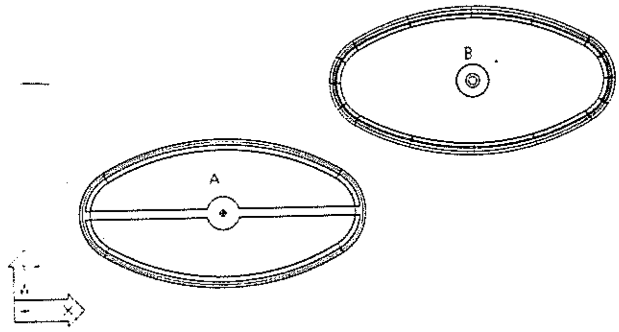


Figure 4.22 Solid parts attached

Set the display to an isometric view. (See Figure 4.23.)

Command: **8**

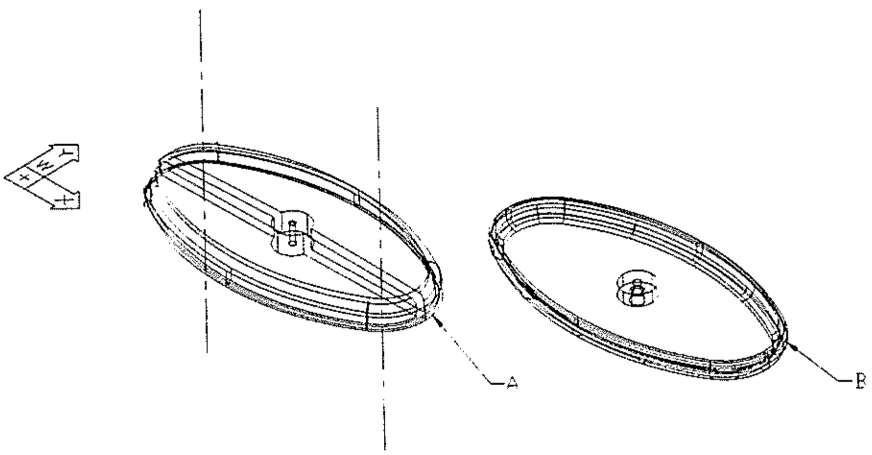


Figure 4.23 Display set to an isometric view

Now you will apply assembly constraints to assemble the two solid parts together. As we have explained, there are four kinds of assembly constraints: mate, flush, angle, and insert. To run these commands, you can select the Create... item of the Constraints cascading menu of the Assembly pull-down menu to use the AMCONSTRAIN command or select the Mate, Flush, Angle, or Insert item of the Constraints cascading menu to use the AMMATE, AMFLUSH, AMANGLE, or AMINSERT command.

The AMCONSTRAIN command is a collective assembly application command that controls the AMMATE, AMFLUSH, AMANGLE, and AMINSERT commands.

Use the AMCONSTRAIN command. From left to right, Figure 4.24 shows the controls on the four kinds of assembly constraints: mate, flush, angle, and insert. Use the mate constraint button. Then select A (Figure 4.23).

Because the solid part is displayed in wireframe mode and you are selecting an edge of it, the selected edge can represent a point, a line, or a plane. Therefore, a blinking mouse icon (Figure 4.25) appears on your screen after you select a point of the solid part. By pressing the left mouse button, you can toggle among three kinds of mating mode: point, line, or plane.

<Assembly> **<Constraints>** **<Create...>**

Command: **AMCONSTRAIN**

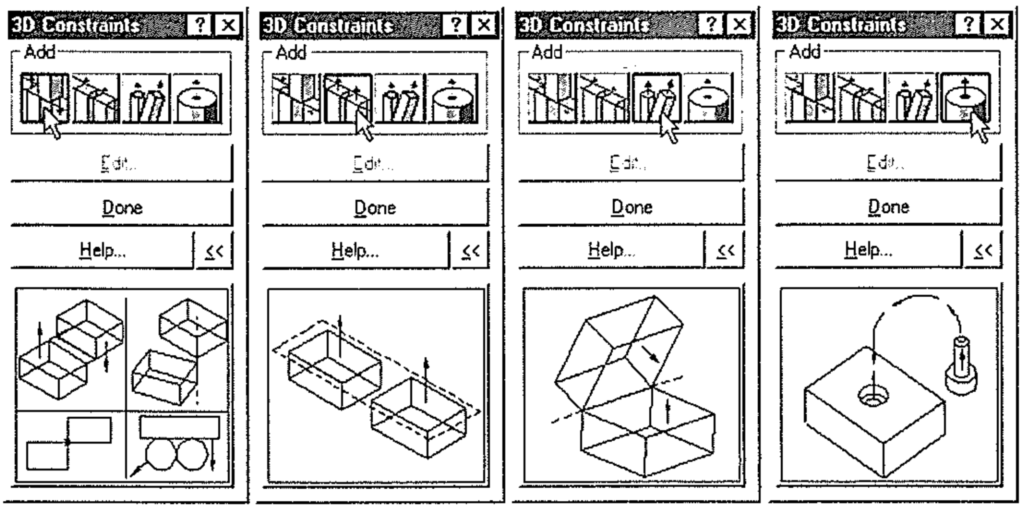


Figure 4.24 Four kinds of assembly constraints controlled by the 3D Constraints dialog box

Select first set of geometry: **[Select A (Figure 4.23).]**
(First set = Axis, (arc), RETURN to Accept)
Clear/Face/Point/cYcle/<Select first set>: **[Press the left mouse button to toggle.]**
(First set = Point, (arc), RETURN to Accept)
Clear/aXis/Face/cYcle/<Select first set>: **[Press the right mouse button to accept if the center point of the selected edge is highlighted (Figure 4.25).]**

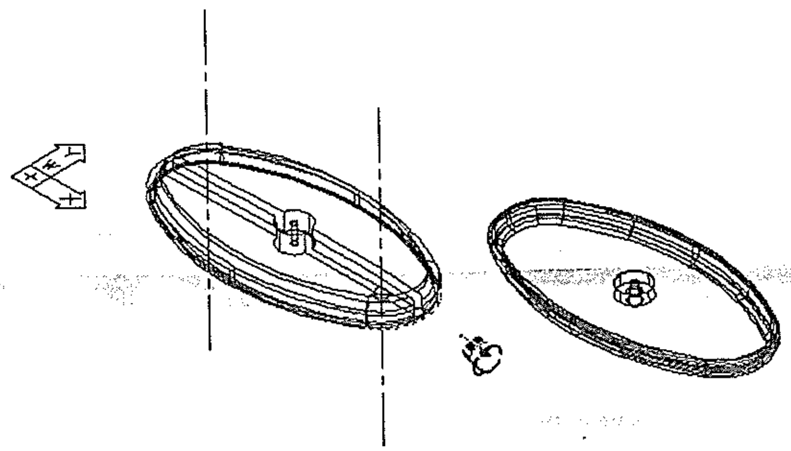


Figure 4.25 Toggle among point, line, and plane by using the blinking mouse icon

The center point of an edge of the upper casing is selected. Now select a point on the second set of geometry, the lower casing. When selecting, you may have to zoom in to select the edge of the mating face. Again, the blinking mouse icon appears. Press the left mouse button until the center point of the selected edge is highlighted. (See Figure 4.26.) The center point of the selected edge is selected. Now you have to specify an offset distance. Because we want the two points to coincide, set the offset value to zero. (See Figure 4.27.)

Select second set of geometry: **[Select B (Figure 4.23).]**
Clear/Face/Point/cYcle/<Select second set>: **[Press the left mouse button to toggle.]**
(Second set = Point, (arc), RETURN to Accept)
Clear/aXis/Face/cYcle/<Select second set>: **[Press the right mouse button to accept if the center point of the selected edge is highlighted (Figure 4.26).]**
Offset: **0**

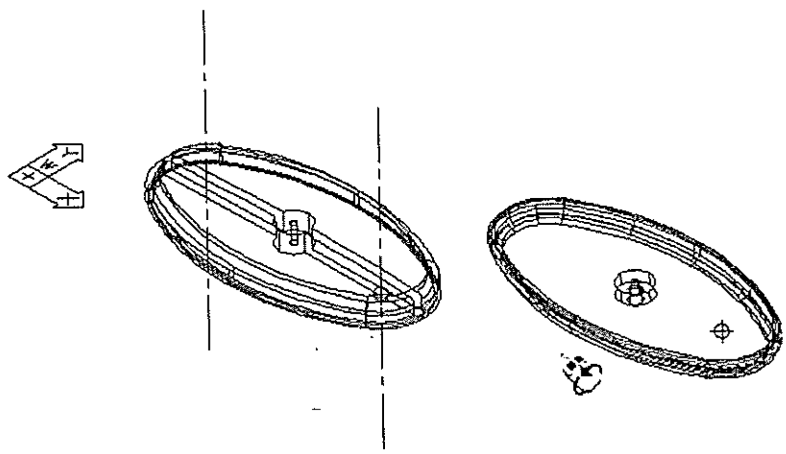


Figure 4.26 Toggle until the center point is highlighted

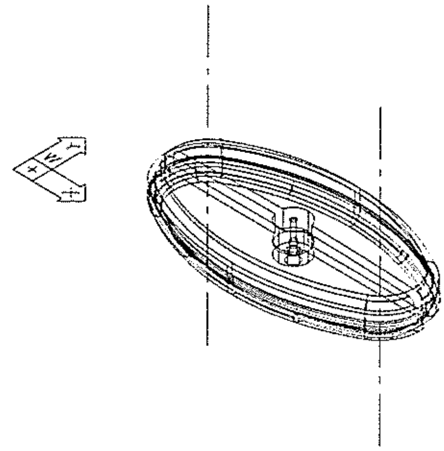


Figure 4.27 Lower casing translated toward the grounded upper casing

Now the two solid parts assemble together. As you can see on your screen, the lower casing translates toward the upper casing because the upper casing is the first solid part and is grounded. Looking at Figure 4.27, you may think you have already properly assembled the two solid parts together. However, this may not be so. Turn on the DOF symbol to find out how many degrees of freedom are left. Select UCASE1 in the browser, then press the right mouse button. Then select the DOF symbol. (See Figure 4.28.)

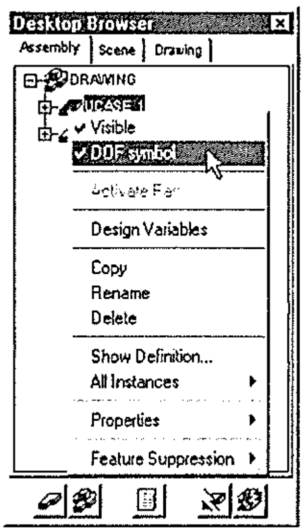


Figure 4.28 Turning on the DOF symbol in the browser

The DOF symbol for the Ucase is turned on. Now turn on the DOF symbol for the Lcase. (See Figure 4.29.)

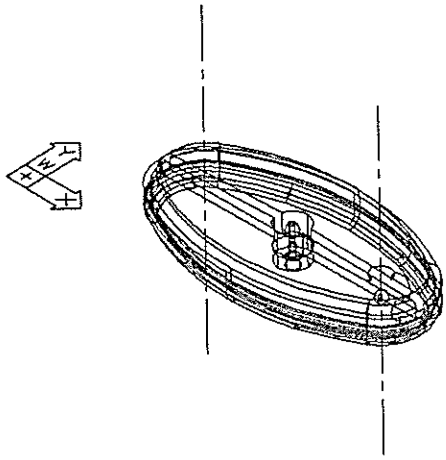


Figure 4.29 DOF symbols turned on

In Figure 4.29, the DOF symbol of the upper casing shows no freedom at all because this solid part is the first solid and is grounded. The DOF symbol for the lower casing shows three degrees of rotational movement, because a point of it is assembled to a point of the upper casing. As a result, the lower casing cannot translate linearly in all directions. However, it can rotate freely about a point. This means that there are three degrees of rotational freedom left.

To fully constrain the lower casing with respect to the grounded upper casing, you need to apply further constraints to it. However, you may find it difficult to select a point on the upper casing and select a corresponding point on the lower casing, because the two solid parts are already put together.

In order to prevent the solid parts from translating and assembling while the assembly constraints are applied, you can unselect the Update Assembly as Constrained box in the Assemblies tab of the Desktop Preferences dialog box. (See Figure 4.14.)

To see how this works, use the UNDO command to undo the AMCONSTRAIN command. After undoing, your screen should resemble Figure 4.23.

<Edit> **<Undo>**

Now apply the AMPREFS command. Then uncheck the Update Assembly as Constrained box (Figure 4.14). After that, apply the AMCONSTRAIN command and go through the steps as shown in Figures 4.24 through 4.26.

<Assembly> **<Assembly Preferences...>**

<Assembly> **<Constraints>** **<Create...>**

After you turn on the DOF symbols, the drawing should resemble Figure 4.30.

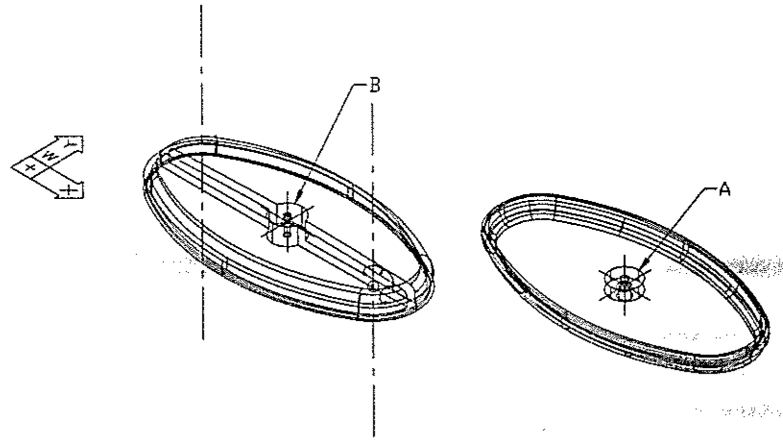


Figure 4.30 Assembly not updated automatically after application of assembly constraint

Note in Figure 4.30 that the center of geometry has moved toward the grounded solid part, but updating is deferred. Now use the AMMATE command to add a mate constraint to an axis of the lower casing and an axis of the upper casing. (See Figures 4.31 and 4.32.)

<Assembly> **<Constraints>** **<Mate>**

Command: **AMMATE**
Select first set of geometry: **[Select A (Figure 4.30).]**
(First set = Axis, (arc), RETURN to Accept)
Clear/Face/Point/cYcle/<Select first set>: **[Toggle until an axis is highlighted.]**
Select second set of geometry: **[Select B (Figure 4.30).]**
(Second set = Point, (spline), RETURN to Accept)
Clear/aXis/Next/cYcle/<Select second set>: **[Press the left mouse button to toggle among point, line, and plane selection.]**
(Second set = Point, (spline), RETURN to Accept)
Clear/aXis/Next/cYcle/<Select second set>: **[Toggle until an axis is highlighted.]**
(Second set = Axis, (cylinder), RETURN to Accept)
Clear/Point/cYcle/<Select second set>: **[Enter to accept if the axis is highlighted (Figure 4.31).]**
Offset: **0**

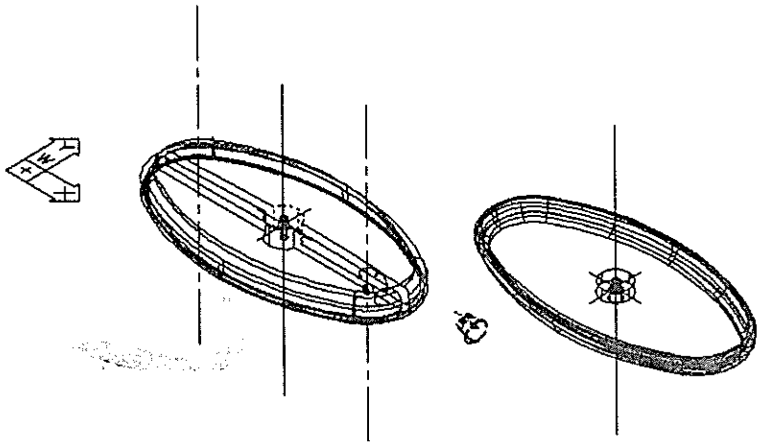


Figure 4.31 Assembly fully constrained but not updated

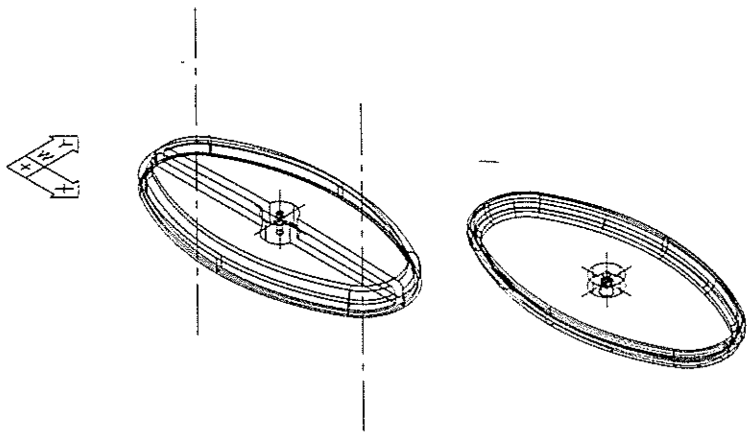


Figure 4.32 DOF symbols show a fully constrained scenario

To update an assembly, you can select the Assembly Update item of the Assembly pull-down menu or use the update assembly button to use the AMASSEMBLE command. (See Figures 4.33 and 4.34.)

<Assembly> **<Assembly Update>**

Command: **AMASSEMBLE**

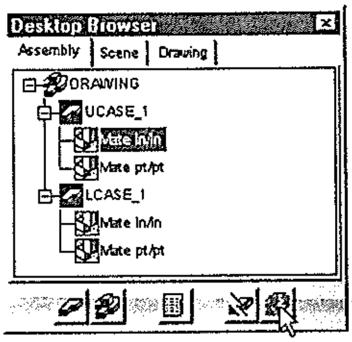


Figure 4.33 Update Assembly button of the Desktop Browser

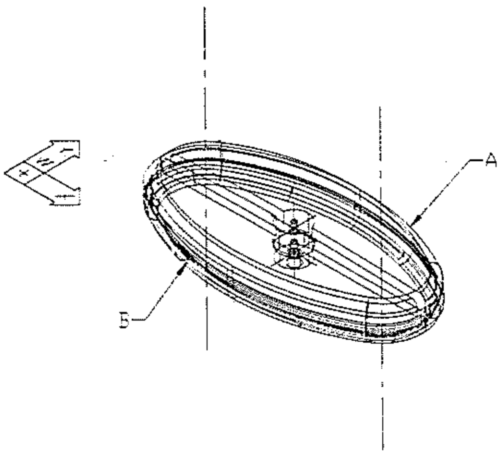


Figure 4.34 Assembly updated

Now the solid parts are assembled and the assembly is updated. To verify the assembly, you can use the AMLISTASSM command.

<Assembly>   <Assembly>   <Query>

Command: **AMLISTASSM**
Select parts/subassemblies Name/<Select>: **[Enter]**
Select part: **[Select B (Figure 4.34).]**
Select part: **[Enter]**
Part/Subassembly name: LCASE_1
Definition name: LCASE External file: C:\projects\lcase.dwg
Degrees of freedom
----
0 Rotational degrees of freedom
0 Translational degrees of freedom
Attributes on definition

In Figure 4.34 or on your screen, it is difficult to tell whether there is any interference between the solid parts. To inspect the assembly visually, you can use the AVEDGES command to render the solid parts. (See Figure 4.35.)

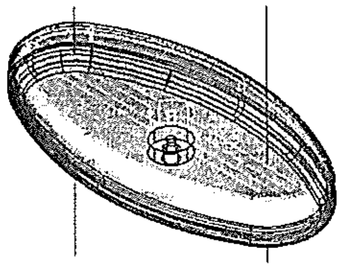


Figure 4.35 Rendered solid parts with edges displayed

By using the AVROTATE command to rotate the solid parts, you can inspect the solid parts from various viewing directions. However, you should not rely only on visual inspection. To check interference, you can use the AMINTERFERE command.

<Assembly> **<Analysis>** **<Interference>**

Command: **AMINTERFERE**
Nested part/subassembly selection? Yes/<No>: [**Enter**]
Select the first set of parts/subassemblies.
Select part/subassembly: [**Select A (Figure 4.34).**]
Select part/subassembly: [**Enter**]
Select the second set of parts/subassemblies.
Select part/subassembly: [**Select B (Figure 4.34).**]
Select part/subassembly: [**Enter**]
Parts/Subassemblies do not interfere.

As shown in the command window interface, the solid parts do not interfere. Now you have an assembly drawing of two attached solid parts. Because the solid parts are external to the current drawing and they may be edited by someone else while you are doing the assembly, you can use the AMAUDIT command to determine whether all externally referenced part definitions are up to date in the current assembly model.

<Assembly> **<Analysis>** **<Audit...>**

Command: **AMAUDIT**

If the files are up to date, you will get the dialog box shown in Figure 4.36.

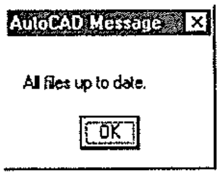


Figure 4.36 Dialog box shown after auditing

The solid parts, being external to the assembly drawing, cannot be edited in the current drawing. To edit and modify the solid parts, you can close the current drawing and open the solid parts one by one.

To edit the solid parts of an assembly without exiting the assembly drawing and going to the individual solid-part drawings, you can localize them by using the AMCATALOG command. Select the All tab. (See Figure 4.37.)

In the Referenced External Definitions box, select UCASE and then press the right mouse button. After that, select the Localize item to localize the solid part. Do the same for the other solid part. (See Figure 4.38.)

<Assembly> <Catalog>

Command: **AMCATALOG**

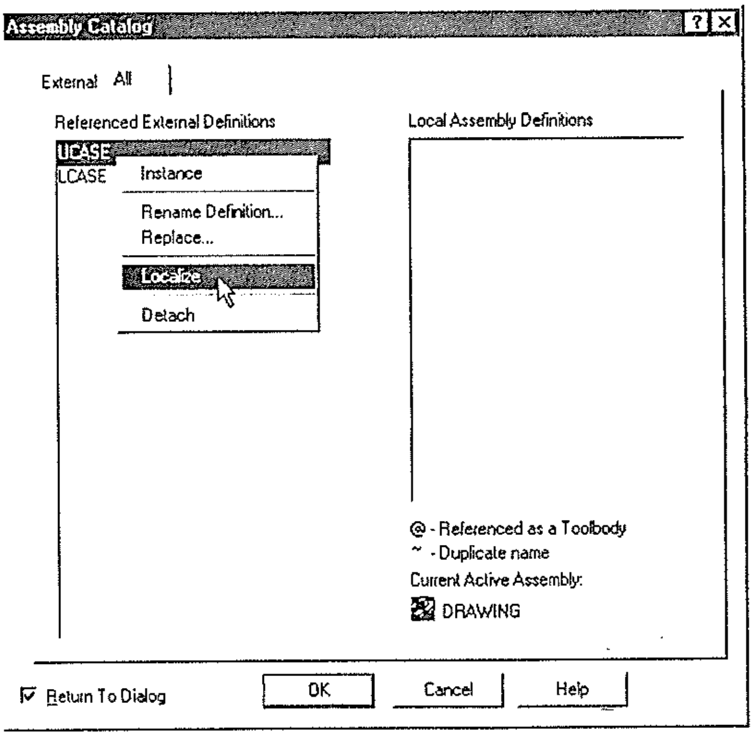


Figure 4.37 All tab of the Assembly Catalog dialog box

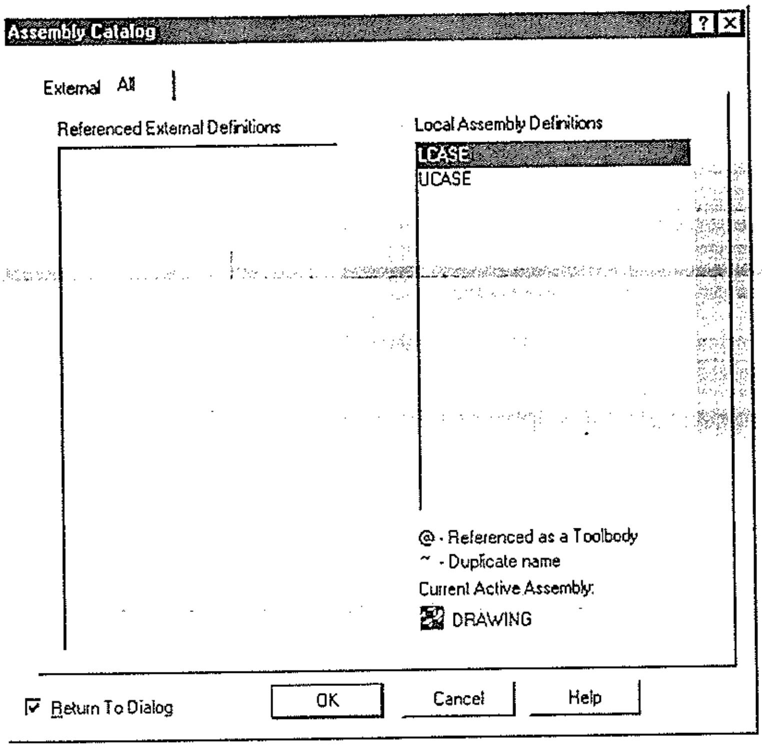


Figure 4.38 External solid parts localized

The solid parts are localized. Now you can edit the solid parts. In the Desktop Browser, double-click UCASE to set it as the active solid part. Then select LCASE and press the right mouse button. (See Figure 4.39.) Uncheck the Visible item to hide the lower casing. (See Figure 4.40.)

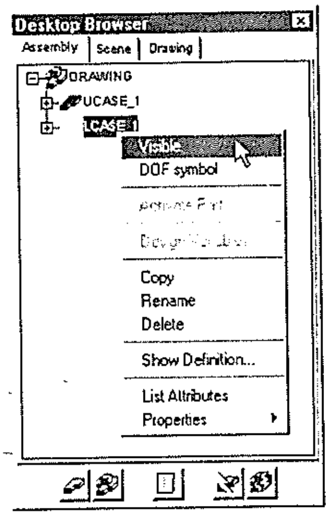


Figure 4.39 Activating the upper casing and hiding the lower casing

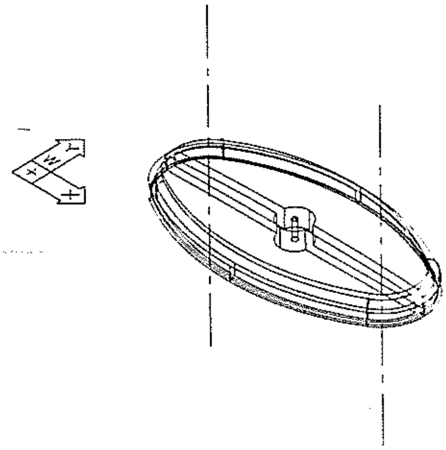


Figure 4.40 Lower casing hidden

Now you will edit the upper casing. Select the solid part in the browser to activate the solid. Then select the Hole1 item (Figure 4.41) in the browser. After that, drag it below Extrusion Blind 2 (Figure 4.42) to reorder the modeling sequence.

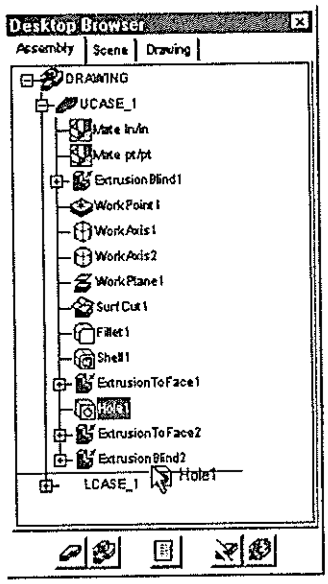


Figure 4.41 Reordering the features of a localized solid part

After reordering, select the Hole1 item and press the right mouse button. Then select the Edit item to edit the hole. (See Figure 4.42.) In the Hole Feature dialog box (Figure 4.43), change the hole to a C'Bore hole with C'Depth of 3 units and C'Dia of 4 units, then select the [OK] button.

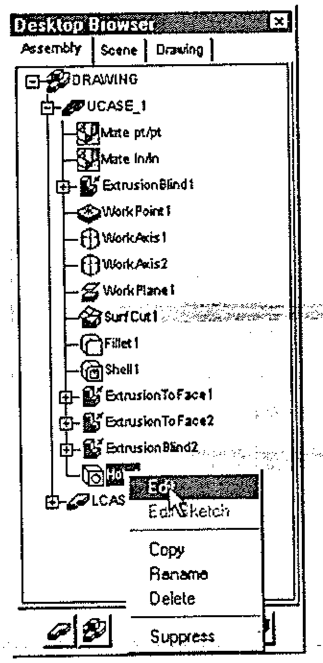


Figure 4.42 Editing the hole feature

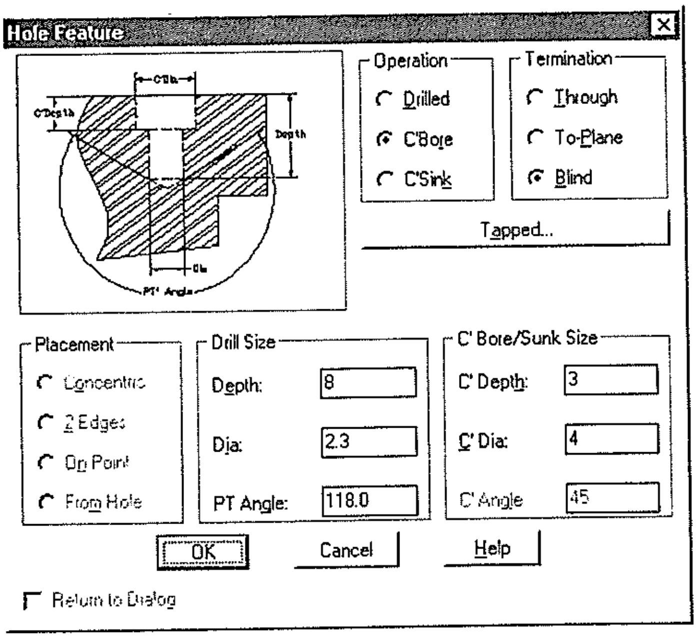


Figure 4.43 Hole feature changed to C'Bore hole

Select object: **[Enter]**

Now select the [Update Part] button of the browser to update the solid part. (See Figure 4.44.)

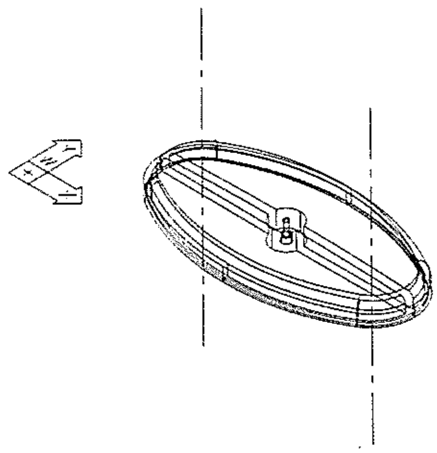


Figure 4.44 Hole feature edited and updated

By using the shortcut in the browser, hide the upper casing, unhide the lower casing, and set the lower casing as the active solid part. (See Figure 4.45.)

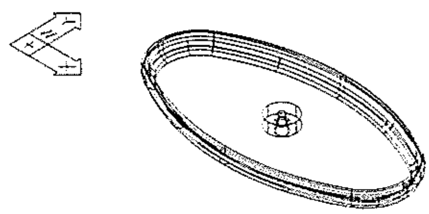


Figure 4.45 Upper casing hidden, lower casing unhidden and activated

Use the shortcut in the browser to edit the hole feature. (See Figure 4.46.) In the Hole Feature dialog box, change the C'Bore hole to a C'Sink hole. Then set the C'Dia to 5 units and the C'Angle to 90°. (See Figure 4.47.)

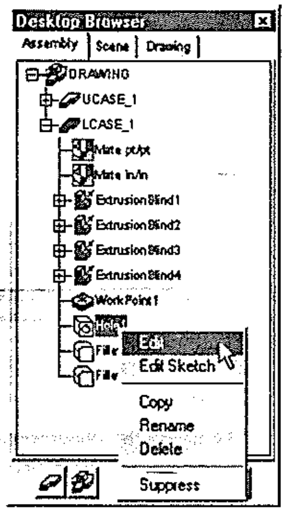


Figure 4.46 Hole feature selected in the browser

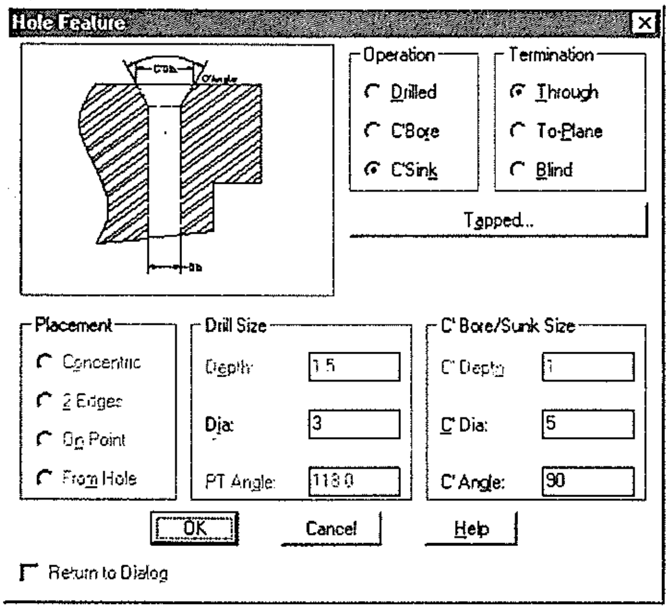


Figure 4.47 C'Bore hole changed to a C'Sink hole

Select the [OK] button.

Select object: **[Enter]**

Now use the [Update Part] button to update the solid part. (See Figure 4.48.)

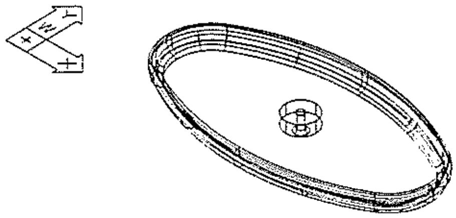


Figure 4.48 C'Bore hole changed to C'Sink hole, solid part updated

Now the two localized solid parts are modified. Save the drawing. Then open it again.

<File> **<Save>**

File name: **Remote.dwg**

<File> **<Open...>**

File name: **Remote.dwg**

As we have explained, it is normal engineering practice to keep the individual solid parts in separate drawings. Therefore, you will externalize the solid parts.

Select the [Catalog] button of the browser to use the AMCATALOG command. Select the All tab. Then select the local solid part and press the right mouse button. (See Figure 4.49.)

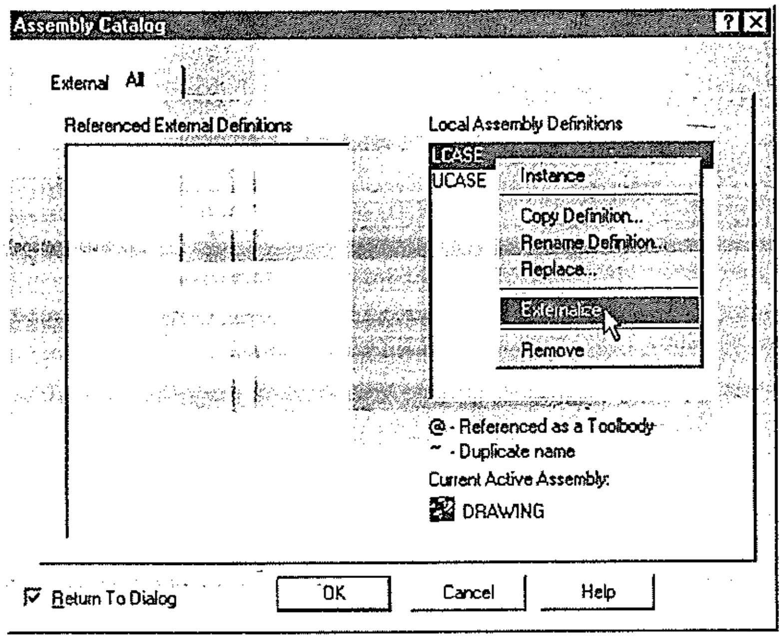


Figure 4.49 All tab of the Assembly Catalog dialog box

Select the Externalize item to externalize the solid part. Overwrite the original drawing file. After that, externalize the other solid part to overwrite the other file. (See Figure 4.50.)

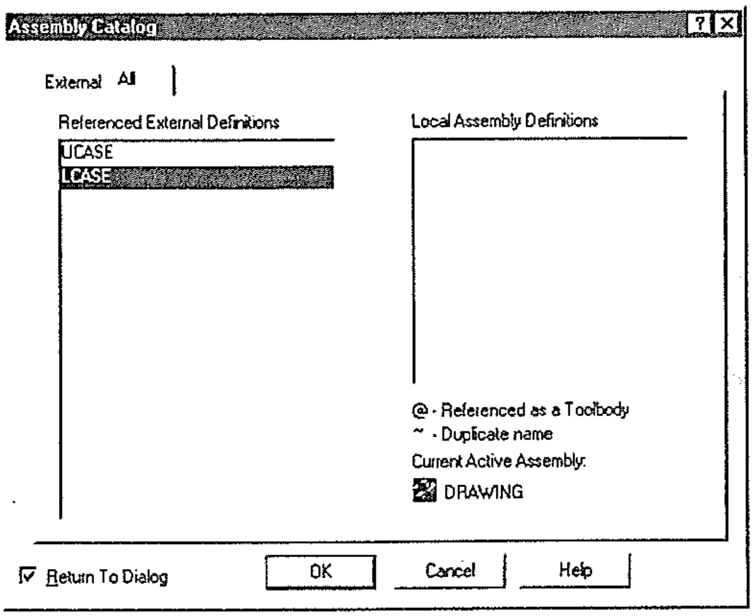


Figure 4.50 Local solid parts externalized

After externalizing the solid parts, you have a set of drawings that consists of an assembly drawing and two attached solid parts. Unhide the hidden part. Then save your drawing.

<File> **<Save>**

File name: **Remote.dwg**

In making this assembly, you used the AMCATALOG command to attach external drawing files. These files then became external solid parts of the assembly drawing. After that, you assembled the instances of the solid parts together by applying assembly constraints. There are four kinds of assembly constraints: mate, flush, angle, and insert. To apply the assembly constraints, there are five commands: AMCONSTRAIN, AMMATE, AMFLUSH, AMANGLE, and AMINSERT. The AMCONSTRAIN command is a collective command to apply the four kinds of assembly constraints. You also experienced updating and not updating when applying assembly constraints.

After assembling, you localized the external solid parts to make them local solid parts. Since they were local to the assembly drawing, you could edit and modify them without exiting the current assembly drawing. After editing, you externalized the modified local solid parts to make them external solid parts. Finally, you had an assembly drawing together with a set of external solid part drawings.

You will generate assembly scenes later in this chapter. In the scenes, you will set up exploded views.

Four-Bar Linkage Project (Continued)

This assembly consists of four linkage bars of different lengths and four identical pivot pins. In Chapter 3, you constructed the linkage bars and a pivot pin in a multi-part drawing. The solid definition of the linkage bar was constructed by using a set of global design variables. Open the file that you saved in Chapter 3. (See Figure 4.51.)

<File> **<Open... >**

File name: **4bar.dwg**

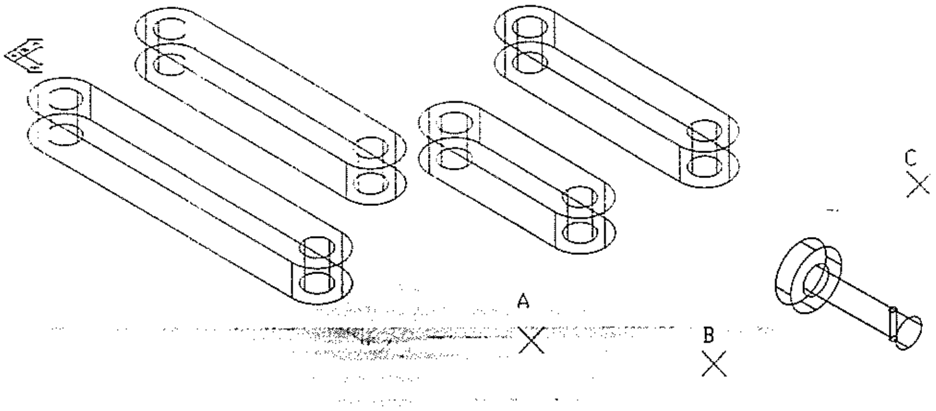


Figure 4.51 Solid parts for the four-bar linkage

Here you will make three more instances of the pivot pin and then assemble all the solid parts together. Use the AMCATALOG command to open the Assembly Catalog dialog box. In the All tab, rename the solid parts as shown in Figure 4.52. Then select the PIVOT item and press the right mouse button. After that, select the Instance item to insert it into the drawing as instances.

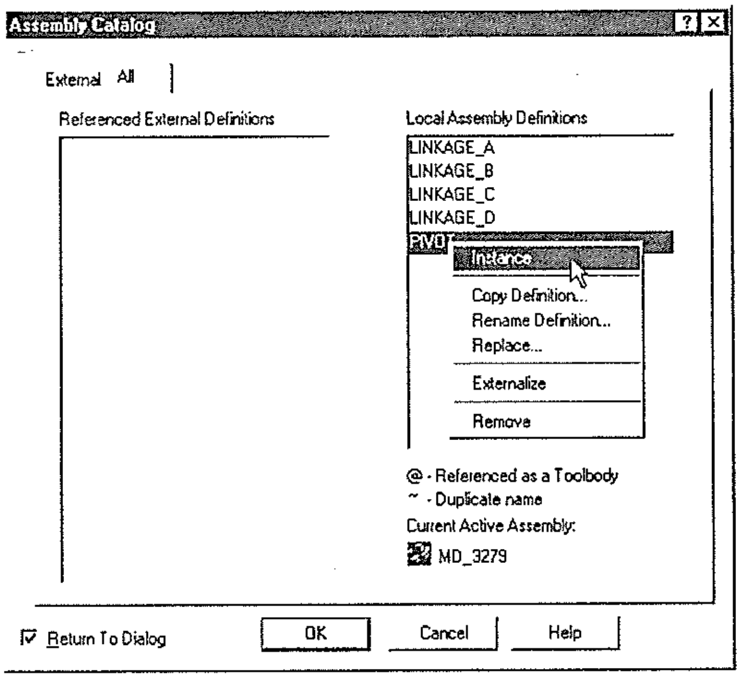


Figure 4.52 Assembly Catalog dialog box

Select insertion point: **[Select A, B, and C (Figure 4.51).]**
Select insertion point: **[Enter]**

On returning to the Assembly Catalog dialog box, select the [OK] button. (See Figure 4.53.)

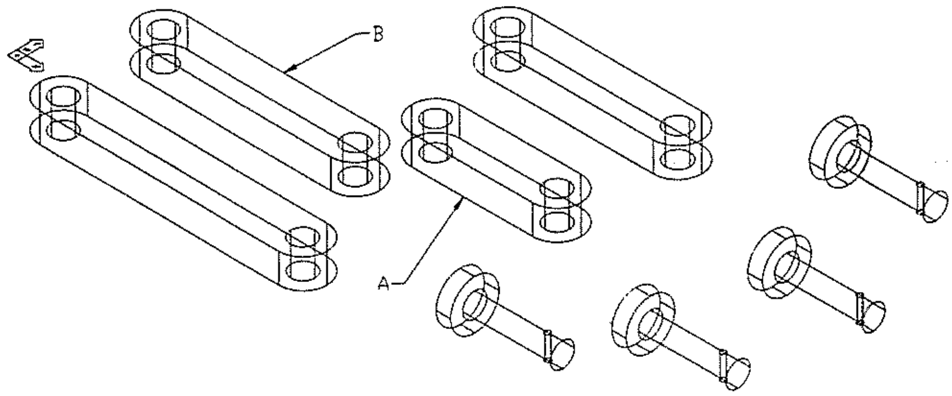


Figure 4.53 Instances of the pivot pin inserted

Now you have four solid definitions of the linkage bar and four instances of the pivot pin. To display the centers of geometry and the DOF symbols, use the AMVISIBLE command.

<Assembly> **<Assembly Visibility>**

Command: **AMVISIBLE**

[Assembly
Unhide **Center of Geometry** **Degrees of Freedom**
OK]

During work on the last project, you used the AMMATE command to apply the mate constraint to a pair of points and to a pair of lines. Here you will use the AMMATE command to apply a mate constraint to a pair of planes. (See Figures 4.54 and 4.55.)

<Assembly> **<Constraints>** **<Mate>**

Command: **AMMATE**
Select first set of geometry: **[Select A (Figure 4.53).]**
(First set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Press the left mouse button to toggle until a plane is highlighted and the direction of the arrow is the same as that shown in Figure 4.54.]**
Select second set of geometry: **[Select B (Figure 4.53).]**
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Press the left mouse button to toggle until a plane is highlighted and the direction of the arrow is the same as that shown in Figure 4.54.]**
(Second set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Enter]**

Offset: **0**

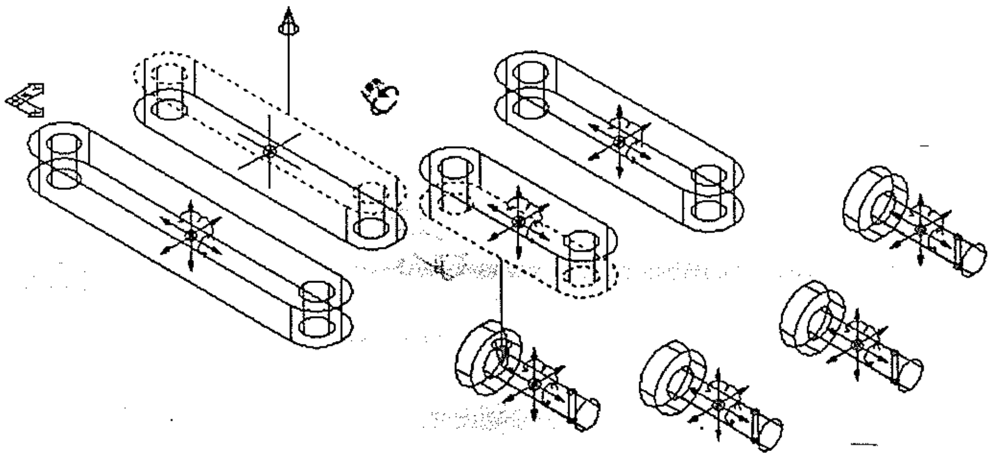


Figure 4.54 Application of mate constraint on a pair of planes

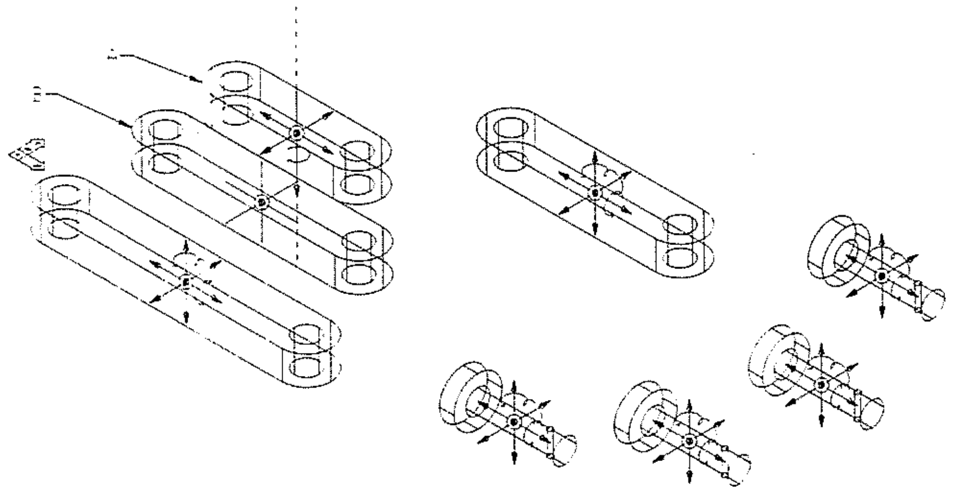


Figure 4.55 Mate constraint applied on a pair of planes

In Figure 4.55, the DOF symbol indicates that the mate constraint has removed three degrees of freedom and the linkage has three degrees of freedom left: translation along the X axis, translation along the Y axis, and rotation about the Z axis. Apply the AMMATE command on the axes of A and B (Figure 4.55). Toggle until the axes are highlighted (Figure 4.56). Then set the offset to 0. (See Figure 4.57.)

<Assembly> **<Constraints>** **<Mate>**

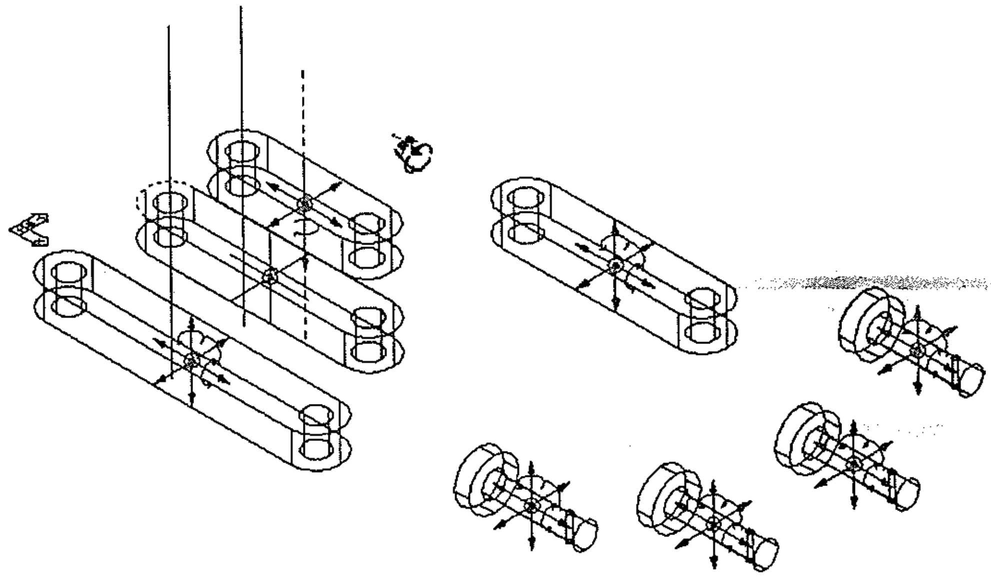


Figure 4.56 Application of mate constraint on a pair of axes

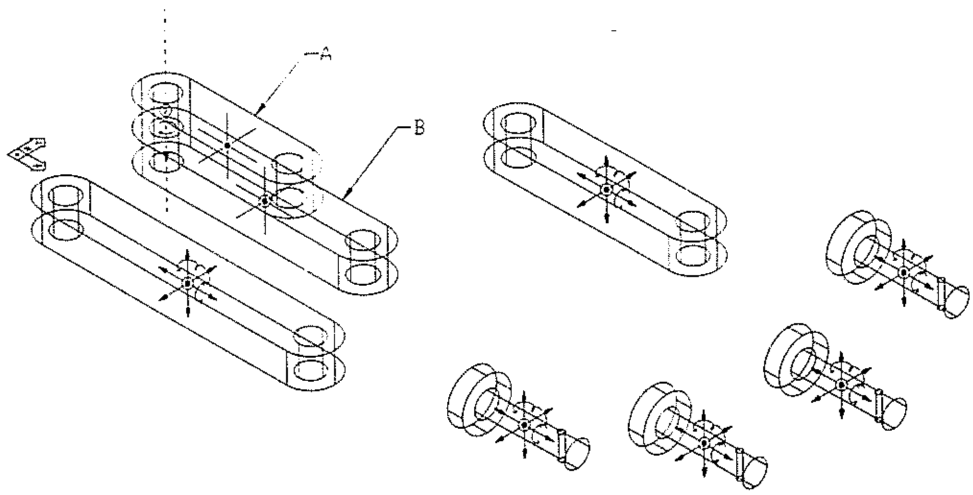


Figure 4.57 Mate constraint applied on a pair of axes

Now there is one degree of freedom left, rotation about the Z axis. Use the AMANGLE command to add an angle constraint. (See Figures 4.58 and 4.59.)

<Assembly> **<Constraints>** **<Angle>**

Command: **AMANGLE**
Select first set of geometry: **[Select A (Figure 4.57).]**
(First set = Plane/Vector)

Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Press the left mouse button to toggle until a plane is highlighted and the direction of the arrow is the same as that shown in Figure 4.58.]**
Select second set of geometry: **[Select B (Figure 4.57).]**
(Second set = Plane/Vector)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Press the left mouse button to toggle until a plane is highlighted and the direction of the arrow is the same as that shown in Figure 4.58.]**
Angle: **90**

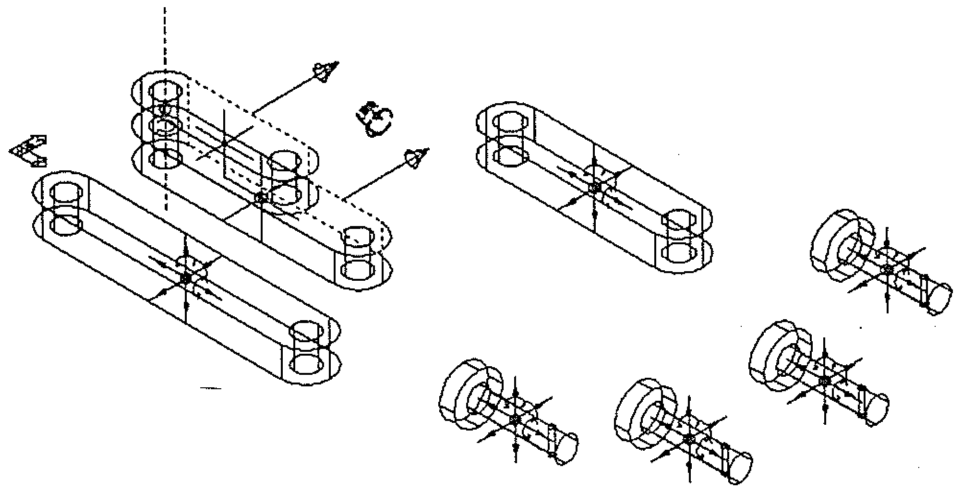


Figure 4.58 Application of angle constraint on a pair of planes

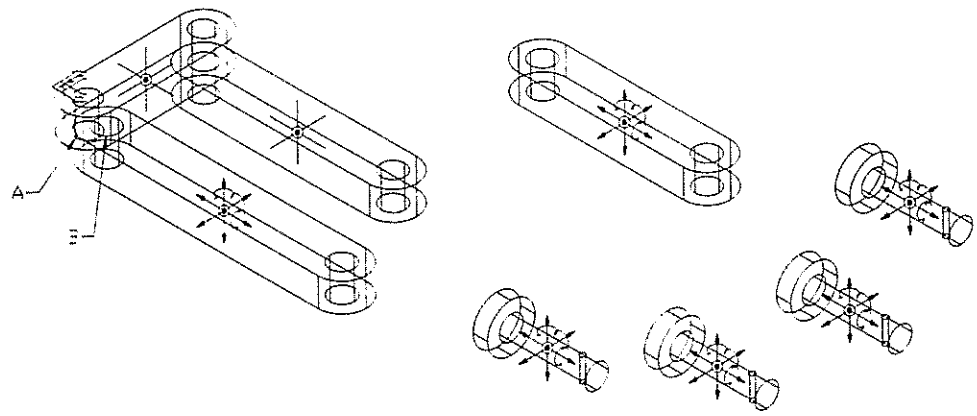


Figure 4.59 Angle constraint applied to a pair of planes

Use the AMINSERT command to constrain two linkages. The AMINSERT command applies assembly constraints to a pair of circular edges such that their center lines and the planes of their circular edges coincide. (See Figures 4.60 and 4.61.)

<Assembly> **<Constraints>** **<Insert>**

Command: **AMINSERT**
Select first circular edge: [**Select A (Figure 4.59).**]
(First set = Plane/Axis)
Clear/fLip/<Accept>: [**Accept if the direction of the arrow is the same as that shown in Figure 4.60.**]
Select second circular edge: [**Select B (Figure 4.59).**]
(Second set = Plane/Axis)
Clear/fLip/<Accept>: [**Accept if the direction of the arrow is the same as that shown in Figure 4.60.**]
Offset: **0**

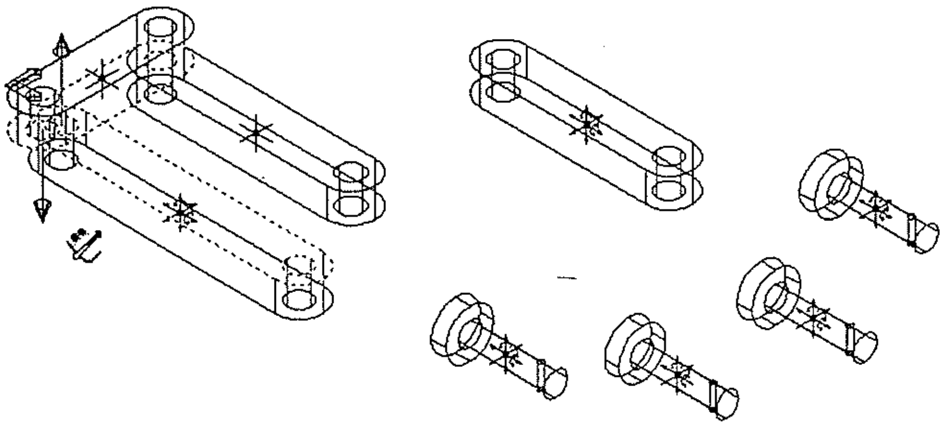


Figure 4.60 Application of insert constraint to a pair of circular edges

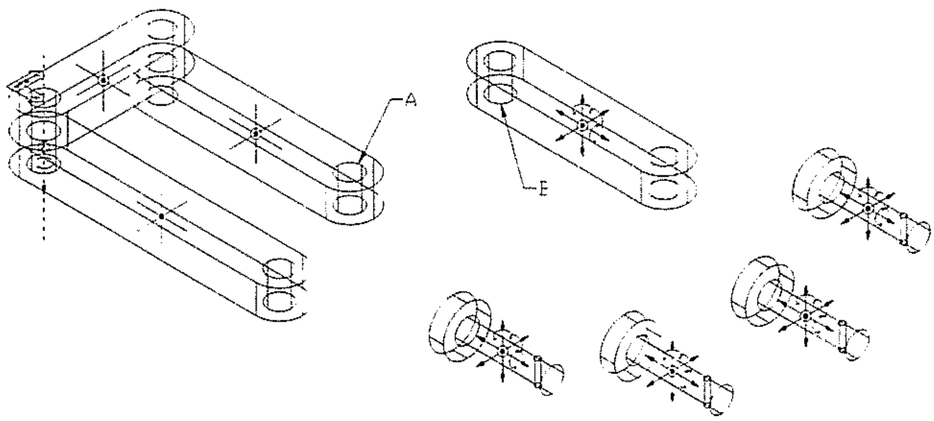


Figure 4.61 Insert constraint applied to a pair of circular edges

Repeat the AMINSERT command on another pair of circular edges. (See Figure 4.62.)

<Assembly> **<Constraints>** **<Insert>**

Command: **AMINSERT**
Select first circular edge: [**Select A (Figure 4.61).**]
(First set = Plane/Axis)
Clear/fLip/<Accept>: [**Accept if the arrow is pointing upward.**]
Select second circular edge: [**Select B (Figure 4.61).**]
(Second set = Plane/Axis)
Clear/fLip/<Accept>: >: [**Accept if the arrow is pointing downward.**]
Offset: **0**

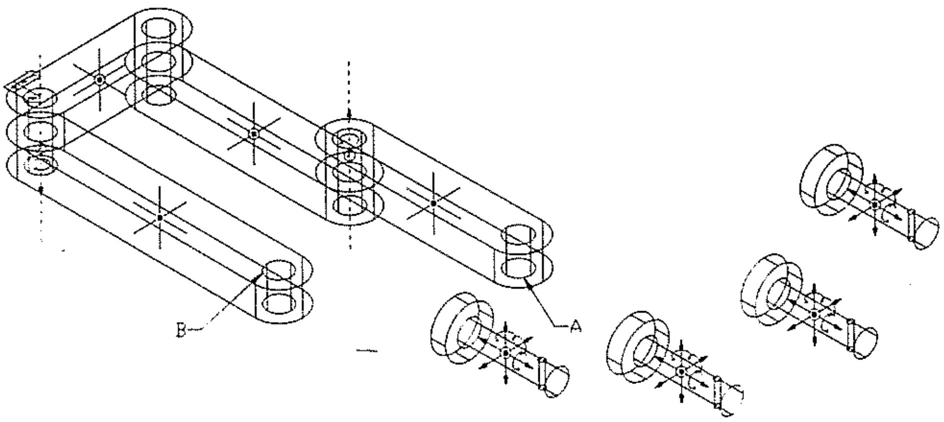


Figure 4.62 Inserted constraint applied

Use the AMINSERT command on a pair of circular edges A and B (Figure 4.62). (See Figure 4.63.) Note how this constraint applies a relational constraint to the other linkages.

<Assembly> **<Constraints>** **<Insert>**

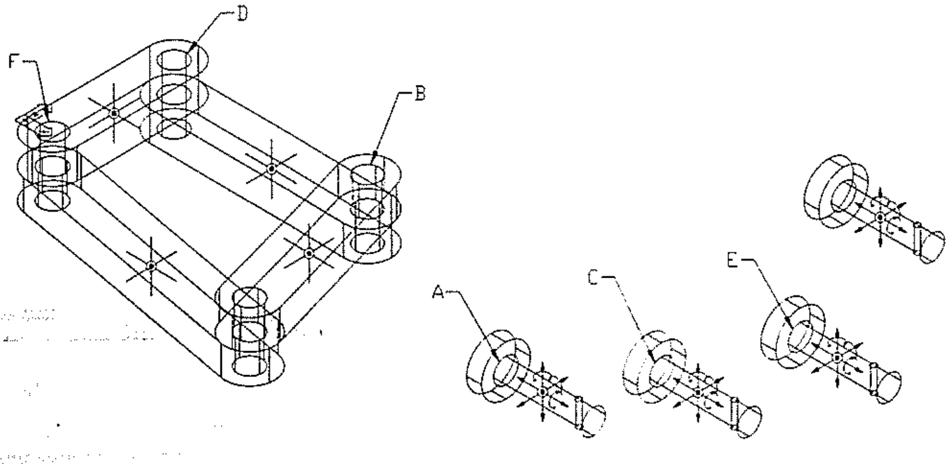


Figure 4.63 Inserted constraint applied

Continue to use the AMINSERT command three more times on paired circular edges A and B, C and D, and E and F (Figure 4.63). (See Figure 4.64.)

<Assembly>	**<Constraints>**	**<Insert>**
<Assembly>	**<Constraints>**	**<Insert>**
<Assembly>	**<Constraints>**	**<Insert>**

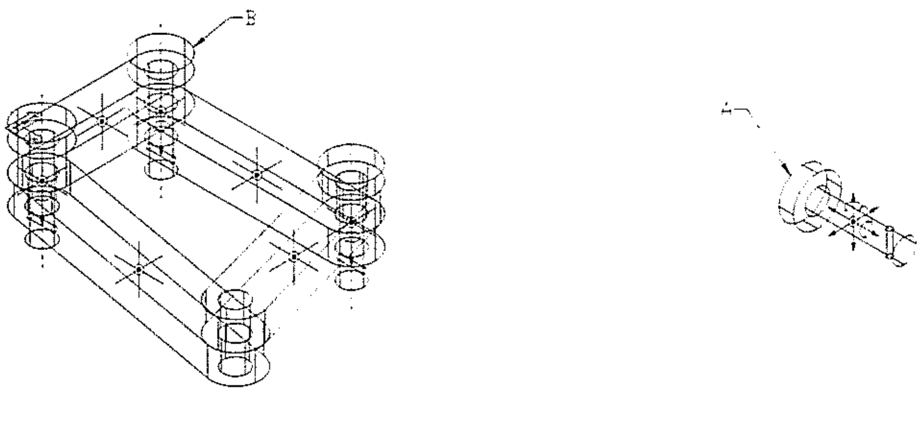


Figure 4.64 Pivot pins inserted

To illustrate the way the AMFLUSH command works, the last pivot pin will be assembled using a different approach. You will use the AMFLUSH command to add a flush constraint to a pair of planes and then use the AMMATE command to add a mate constraint to a pair of axes. Use the AMFLUSH command. (See Figures 4.65 and 4.66.)

<Assembly> **<Constraints>** **<Flush>**

Command: **AMFLUSH**
Select first set of geometry: **[Select A (Figure 4.64).]**
(First set = Plane)
Clear/aXis/Point/fLip/cYcle/<Accept>: **[Toggle until the selected face is highlighted and the arrow is pointing in the direction shown in Figure 4.65.]**
Select second set of geometry: **[Select B (Figure 4.64).]**
(Second set = Plane)
Clear/aXis/Point/fLip/cYcle/<Accept>: **[Toggle until the selected face is highlighted and the arrow is pointing in the direction shown in Figure 4.65.]**
Offset: **0**

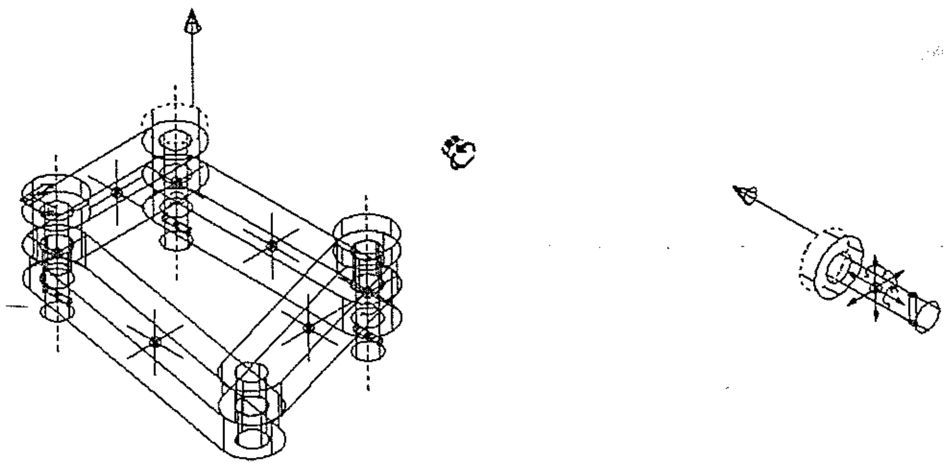


Figure 4.65 Application of flush constraint to a pair of planes

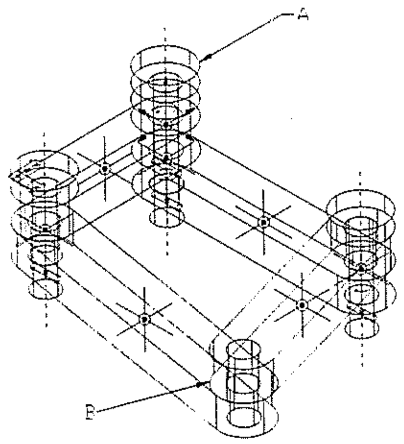


Figure 4.66 Flush constraint applied to a pair of planes

Now use the AMMATE command to add a mate constraint to a pair of axes A and B (Figure 4.66). (See Figure 4.67.)

<Assembly> <Constraints> <Mate>

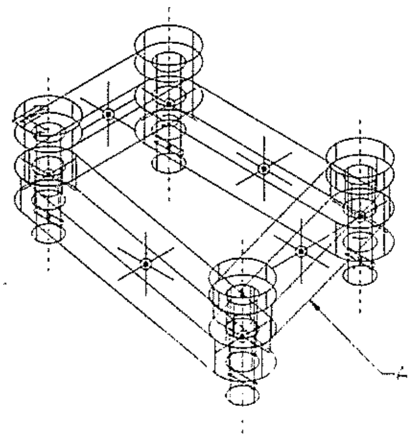


Figure 4.67 Mate constraint applied to a pair of axes

To appreciate how the flush constraint and the mate constraint differ from the insert constraint, you will edit the extrusion height of solid part A (Figure 4.67). Change it to 10 units. Then update the solid part and the assembly. (See Figure 4.68.)

<Part> <Part> <Make Active>

Command: **AMACTIVATE**
Select part to activate (or ?): **[Select A (Figure 4.67).]**

<Part> <Edit Feature>

Command: **AMEDITFEAT**
Independent array instance/Sketch/surfCut/Toolbody/<select Feature>: **[Select A (Figure 4.67).]**
Select object: **[Select the dimension THK.]**
Enter new value for dimension: **10**

<Part> <Update>

Command: **AMUPDATE**

<Assembly> <Assembly Update>

Command: **AMASSEMBLE**

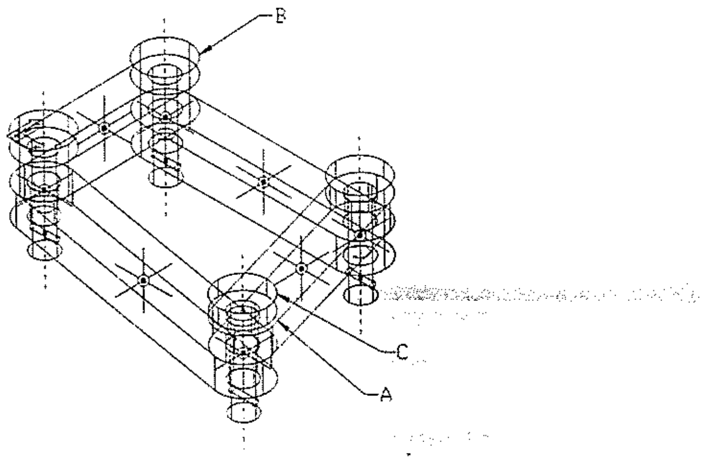


Figure 4.68 Feature edited, part updated, and assembly updated

In Figure 4.68, note that there is a gap at A because the plane C (Figure 4.68) is flush with plane B (Figure 4.68). To modify the assembly, you can delete the assembly constraints applied to A (Figure 4.69).

<Assembly> **<Constraints>** **<Edit...>**

Command: **AMEDITCONST**

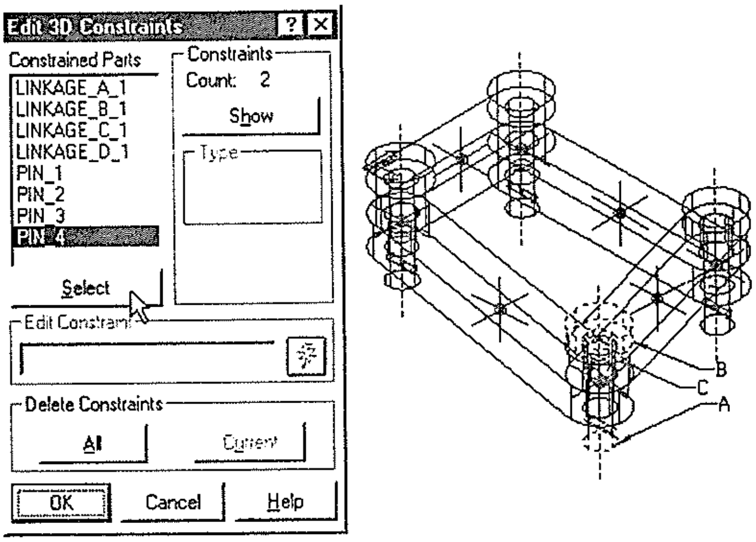


Figure 4.69 Edit 3D Constraints dialog box

In the Edit 3D Constraints dialog box, select the [Select] button and then select solid A (Figure 4.69). Note that the highlighted part name in the Edit 3D Constraints dialog box may not be the same as the one in yours.

After solid A (Figure 4.69) is highlighted, select the [All] button in the Delete Constraints box. Then select the [OK] button. This way, the assembly constraints applied to the selected solid part are removed.

Now use the AMINSERT command to apply an insert constraint to circular edges B and C (Figure 4.69). (See Figure 4.70.)

<Assembly> **<Constraints>** **<Insert>**

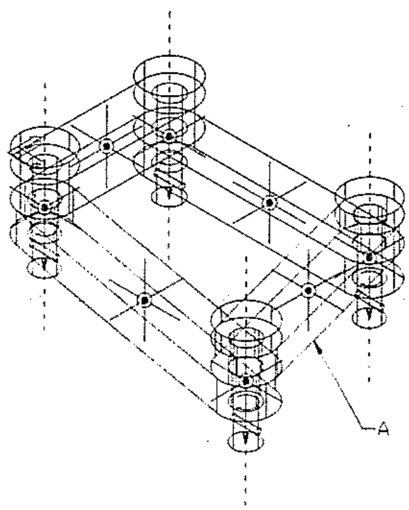


Figure 4.70 Insert constraint applied to a pair of circular edges

The assembly is constrained. To see the way a solid part is replaced by another solid part in the assembly, use the AMREPLACE command. (See Figure 4.71.)

<Assembly> **<Assembly>** **<Replace...>**

Command: **AMREPLACE**

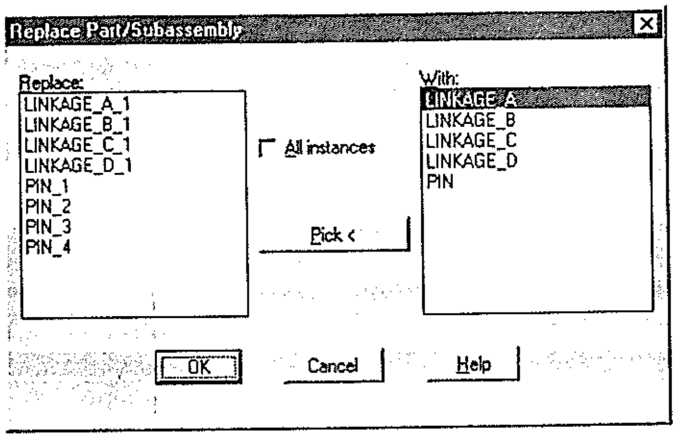


Figure 4.71 Replace Part/Subassembly dialog box

In the Replace Part/Subassembly dialog box, select the [Pick <] button. Then select A (Figure 4.70). Then select LINKAGE_A in the With box. After that, select the [OK] button. (See Figure 4.72.)

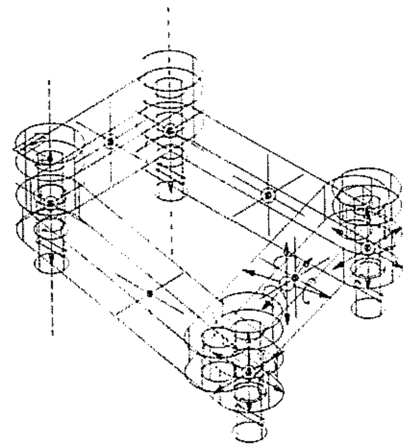


Figure 4.72 Solid part replaced in an assembly

As you can see, the assembly constraints applied to the removed solid part are deleted. To properly assemble the new solid part, you have to apply insert constraints to related pairs of circular edges. (See Figure 4.73.)

<Assembly> **<Constraints>** **<Insert>**

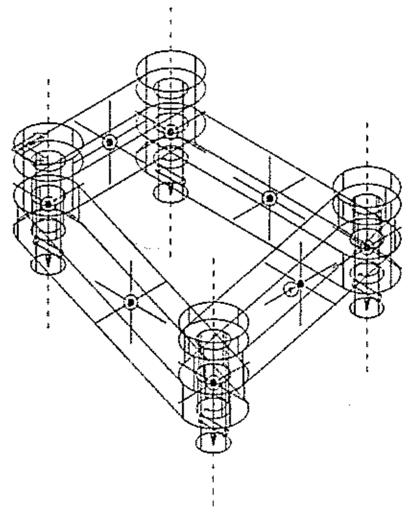


Figure 4.73 Assembly constraints applied

Now you have modified a solid part and performed a replacement. To find out where an instance of a solid part is used in the assembly, you can use the AMWHEREUSED command. (See Figure 4.74.)

<Assembly> **<Assembly>** **<Where Used...>**

Command: **AMWHEREUSED**

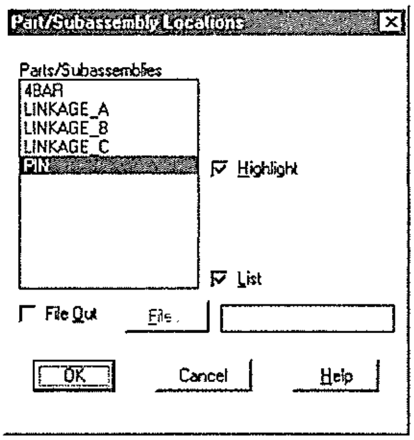


Figure 4.74 Part/Subassembly Locations

In the Part/Subassembly Locations dialog box, select the item PIN and then the [OK] button.

PIN is used in:
PIN_1|
Press RETURN to continue.
PIN_2|
Press RETURN to continue.
PIN_3|
Press RETURN to continue.
PIN_4|
Press RETURN to continue.

As we have said, it is normal engineering practice to place individual solid parts in separate drawing files. Use the AMCATALOG command to externalize the local solid parts. (See Figure 4.75.)

<Assembly> **<Catalog...>**

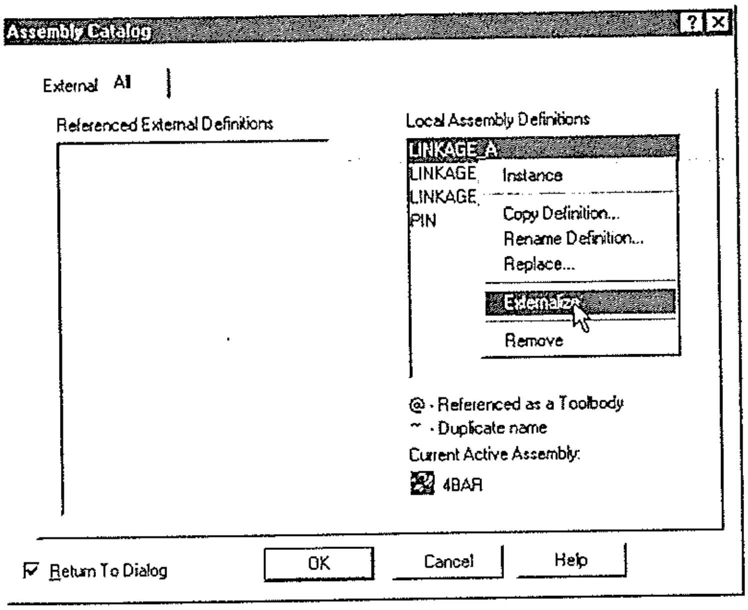


Figure 4.75 Externalizing local definitions

Select a local definition in the All tab and press the right mouse button. Then select the Externalize item to externalize the definition. After that, specify a file name.

File name: **Linkage1.dwg**

Repeat the process for the other three definitions. (See Figure 4.76.)

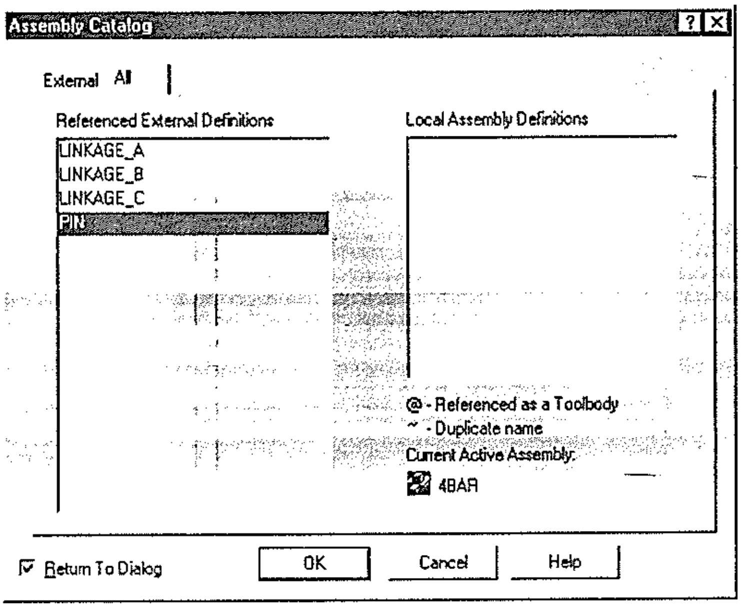


Figure 4.76 Local definitions externalized

Now the assembly and part drawings are complete. Save your drawing.

<File> <Save>

In constructing this assembly, you used a top-down approach in which you constructed all the solid parts in a multi-part drawing. This project consists of four linkage bars and four pivot pins. The four linkage bars are copied definitions and the four pivot pins are instances of a solid part.

With the solid parts residing in a multi-part drawing, you assembled them together by applying the assembly constraints: mate, flush, angle, and insert. After assembling, you edited a solid part and modified the assembly constraint parameters. You also learned how to replace a solid part in an assembly.

After assembling, you externalized the local solid parts to separate drawing files. Thus, you have an assembly drawing and a set of attached external solid parts.

To reiterate, it is normal engineering practice to organize and manage a set of drawings in separate drawing files and to have an assembly drawing in which the solid parts are externally referenced.

To handle large projects that involve large numbers of solid parts, you can organize the solid parts in a hierarchy of assemblies, subassemblies, and parts.

4.5 Combining

In the last chapter, we said that paired solid parts in a multi-part drawing can be combined. During combining, one of the paired solid parts is set to be the active solid and the other solid part is used as the tool solid. To maintain a proper parametric dimensional relationship between the paired solid parts, you have to apply assembly constraints. After constraining, you can combine them. Now you will complete the die set project from the last chapter.

Die Set for a Gear (Continued)

Open the drawing Dieset.dwg that you saved in the last chapter. (See Figure 4.77.)

<File> **<Open... >**

File name: **Dieset.dwg**

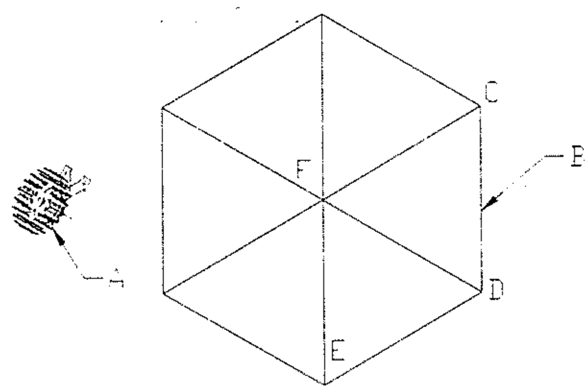


Figure 4.77 Two solid parts: gear and block

Use the AMVISIBLE command to turn on the CG and DOF symbols. Then use the AMMATE command to add a mate constraint to the vertical plane A (Figure 4.77) of the gear and the vertical plane CDEF (Figure 4.77) with an offset distance of 35 units. (See Figures 4.78 and 4.79.)

<Assembly> **<Assembly Visibility...>**

<Assembly> **<Constraints>** **<Mate>**

Command: **AMMATE**
Select first set of geometry: **[Select A (Figure 4.77).]**
(First set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Toggle until the vertical plane A (Figure 4.77) is highlighted and the direction of the arrow is the same as that shown in Figure 4.78.]**

Select second set of geometry: **[Select B (Figure 4.77).]**
(Second set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Toggle until the vertical plane CDEF (Figure 4.77) is highlighted and the direction of the arrow is the same as that shown in Figure 4.78.]**
Offset: **35**

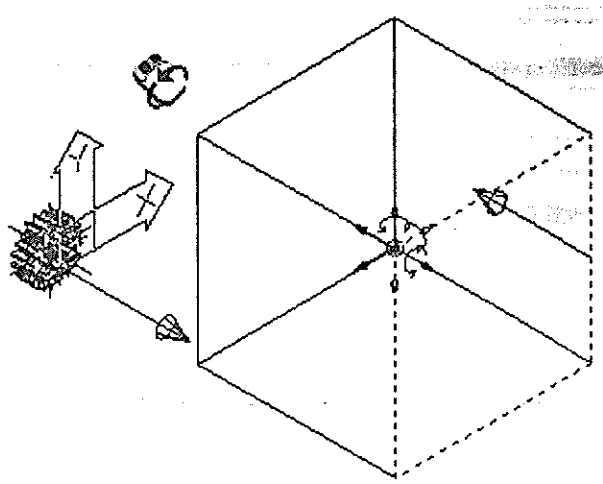


Figure 4.78 Application of mate constraint to a pair of planes

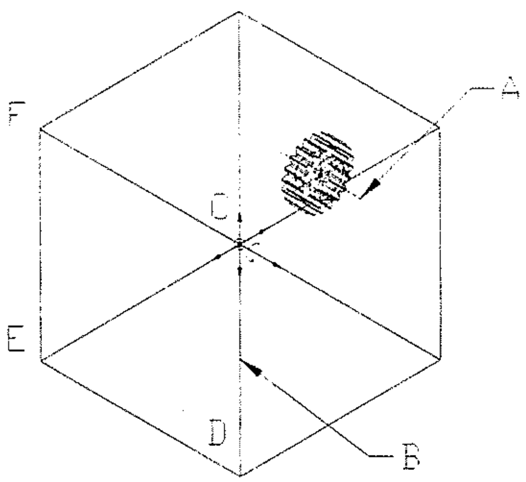


Figure 4.79 Mate constraint applied to a pair of planes

According to the DOF symbols shown in Figure 4.79, there are two translation degrees of freedom and one rotation degree of freedom left. Use the AMMATE command to apply a mate constraint to the work axis of the gear A (Figure 4.79) and the vertical plane CDEF (Figure 4.79) with an offset of 35 units. (See Figures 4.80 and 4.81.)

<Assembly> **<Constraints>** **<Mate>**

Command: **AMMATE**
Select first set of geometry: **[Select A (Figure 4.79).]**
(First set = Axis, (line), RETURN to Accept)
Clear/<Select first set>: **[Accept if the work axis is highlighted.]**
Select second set of geometry: **[Select B (Figure 4.79).]**
(Second set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: **[Accept if the vertical plane CDEF (Figure 4.79) is highlighted and the direction of the arrow is the same as that shown in Figure 4.80).]**
Offset: **35**

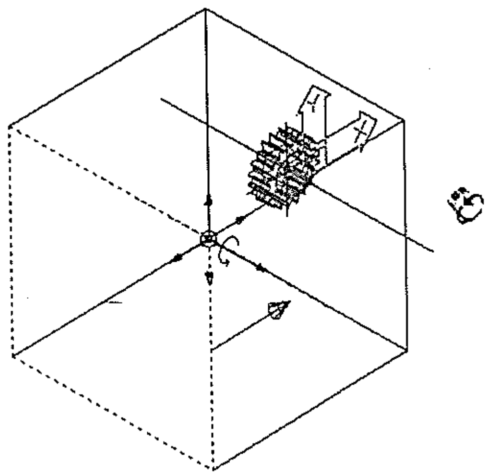


Figure 4.80 Application of mate constraint to a plane and an axis

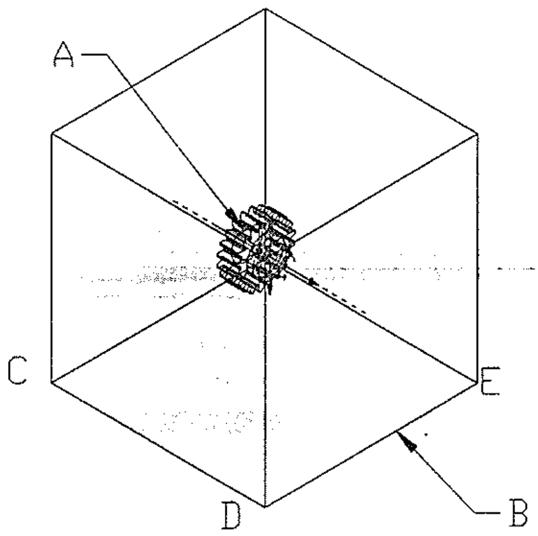


Figure 4.81 Mate constraint applied to a plane and an axis

Repeat the AMMATE command to apply a mate constraint to the horizontal plane CDE (Figure 4.81) and axis A (Figure 4.81). (See Figures 4.82 and 4.83.)

<Assembly> **<Constraints>** **<Mate>**

Command: **AMMATE**
Select first set of geometry: [**Select A (Figure 4.81).**]
(First set = Axis, (line), RETURN to Accept)
Clear/Face/Point/cYcle/<Select first set>: [**Accept if the edge of the gear is highlighted. See Figure 4.82).**]
Select second set of geometry: [**Select B (Figure 4.81).**]
(Second set = Plane)
Clear/aXis/Point/Next/fLip/cYcle/<Accept>: [**Toggle until the horizontal plane CDE (Figure 4.81) is highlighted and the direction of the arrow is the same as that shown in Figure 4.82.**]
Offset: **35**

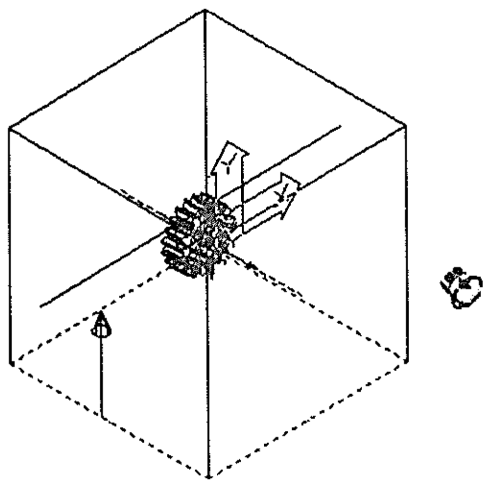


Figure 4.82 Application of mate constraint to a plane and an edge

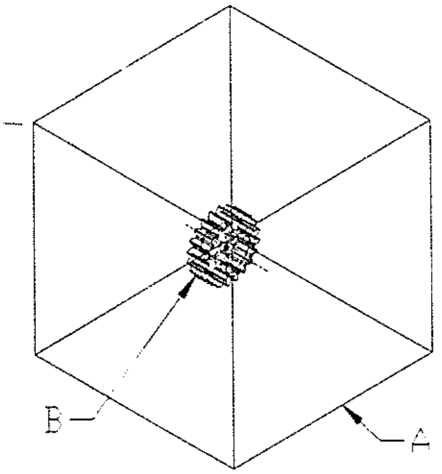


Figure 4.83 Mate constraint applied to a plane and an edge

Now the gear is fully constrained to the block. Set the block as the active solid part. Then use the AMCOMBINE command to combine the solid parts together. Use the cut option. (See Figure 4.84.)

<Part> **<Part>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **PART**
Select part to activate (or ?): **[Select A (Figure 4.83).]**

<Part> **<Placed Features>** **<Combine>**

Command: **AMCOMBINE**
Join/Intersect/<Cut>: **C**
Select toolbody: **[Select B (Figure 4.83).]**

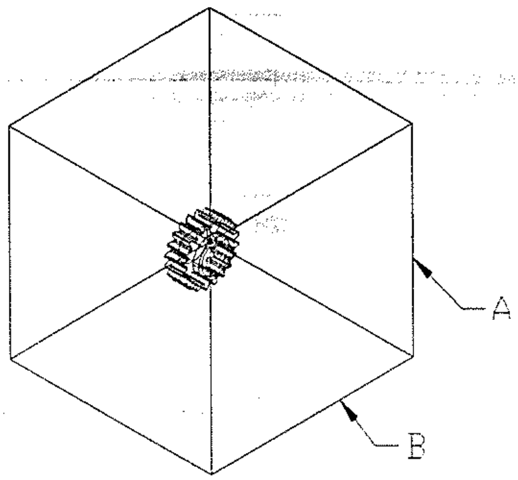


Figure 4.84 Solid parts combined

The solid parts are combined. Now construct a work plane for use as a split plane. (See Figures 4.85 and 4.86.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **Planar Parallel**
2nd Modifier **Offset** Offset **35**
OK]

worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select A (Figure 4.84).]**
Next/<Accept>: **[Accept if the vertical face AB (Figure 4.84) is highlighted.]**
Flip/<Accept>: **[Accept if the direction of the arrow is the same as that shown in Figure 4.85.]**
worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select B (Figure 4.84).]**
Rotate/Z-flip/<Accept>: **[Accept if the UCS orientation is the same as that shown in Figure 4.85).]**

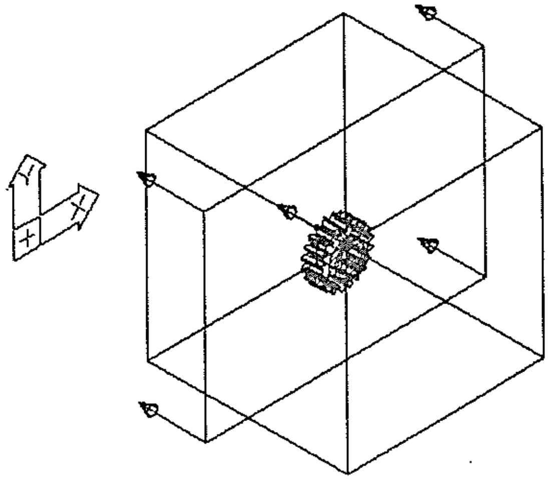


Figure 4.85 Work plane direction

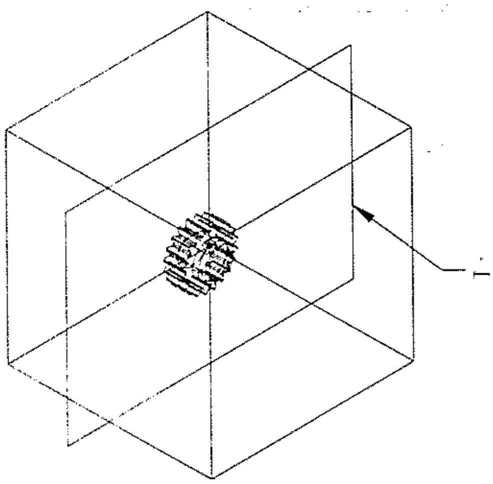


Figure 4.86 Work plane constructed

Now use the AMPARTSPLIT command to split the solid into two. (See Figures 4.87 and 4.88.)

<Part> **<Placed Features>** **<Part Split>**

Command: **AMPARTSPLIT**
Select planar face, work plane, or split line for split: **[Select A (Figure 4.86).]**
Define side for new part: Flip/<Accept>: **[Accept if the direction of the arrow is the same as that shown in Figure 4.87.]**
Enter name of the new part: **HALF**

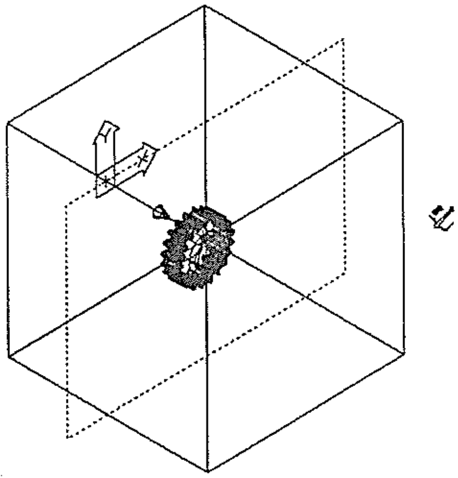


Figure 4.87 Split direction

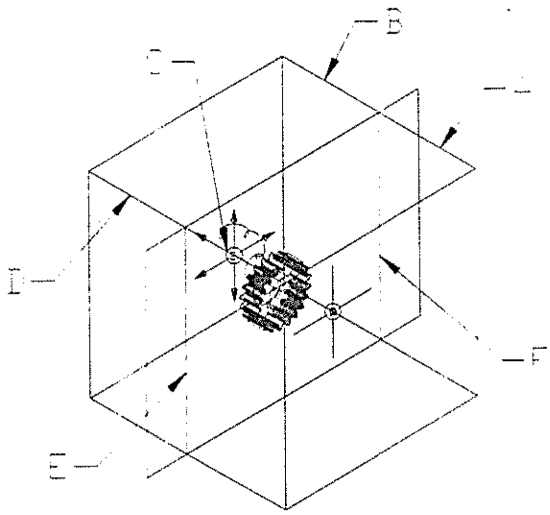


Figure 4.88 Solid part split into two

After splitting, you have two solid parts. Now use the AMMATE command to apply mate constraints to edges A and B, edges C and D, and vertical planes E and F (Figure 4.88). Planes E and F belong to the two solid parts, respectively. (See Figure 4.89.)

<Assembly>	<Constraints>	<Mate>
<Assembly>	<Constraints>	<Mate>
<Assembly>	<Constraints>	<Mate>

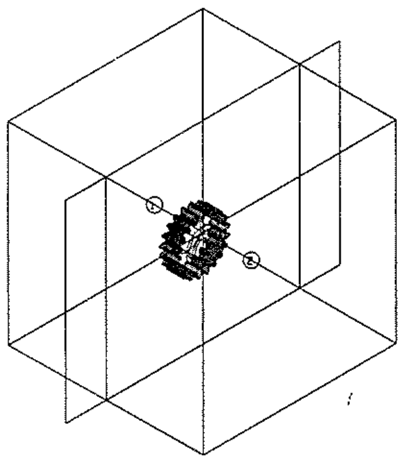


Figure 4.89 Mate constraints applied

Now the models are complete. Save your drawing.

<File> **<Save>**

To reiterate, two solid parts of a multi-part drawing can be combined together. One of the solids will be set as the active solid part and the other will be used as the tool solid. During combining, the tool solid will join, cut, or intersect the active solid. Before combining the solid parts, you need to assemble them together properly in order to establish a parametric relationship. In this project, you constructed two solid parts, applied assembly constraints, and combined them together. Finally, you split the solid into two solid parts.

4.6 Scenes, Exploded Views, and Scene Preferences

To illustrate how parts of an assembly are related to each other, it is a common engineering practice to set up exploded drawing views. To maintain the integrity of the assembled solid parts as a whole and yet obtain an exploded view, you can set up a number of scenes. In these scenes, you can explode or tweak the assembled parts apart to produce exploded views. This way, you can have a scene with no explosion or tweaking to show how the parts are assembled and also have one or more scenes to demonstrate how the parts are related to each other by exploding them.

Now you will continue to work on the three assemblies that you completed earlier in this chapter.

Remote Control Project (Continued)

Open the assembly drawing Remote.dwg that you saved earlier in this chapter.

<File> **<Open...>**

File name: **Remote.dwg**

To work on the scenes of an assembly, select the Scene tab of the Desktop Browser. Then select the background and press the right mouse button. (See Figure 4.90.)

Command: **AMMODE**
Model/<Drawing>: **MODEL**

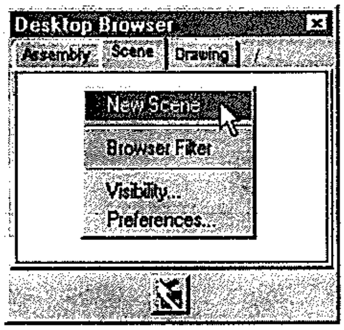


Figure 4.90 Constructing a new scene by using the Desktop Browser

Select the New Scene item to construct a new scene.

Command: **AMNEW**
Create a new Part/Scene/subAssembly/<Instance>: **SCENE**
Create a new Scene of Active Assembly (REMOTE) named: **REMOTE01**

There are two ways to construct an exploded view: by specifying an explosion factor for the solid parts collectively and by tweaking the solid parts in the scene individually. You will tweak the solid parts of this remote control project and use an explosion factor in the next project. To tweak the solid parts, use the AMTWEAK command. Then select the lower part of the assembly. The Tweak Part/Subassembly dialog box appears. (See Figure 4.91.) Select Transform in the Tweak Part/Subassembly dialog box and then the [OK] button. After that, use the Move option to translate the lower casing of the remote control assembly a distance of 60 units in the –Z direction. (See Figure 4.92.)

<Assembly> **<Exploded Views>** **<Add Tweaks...>**

Command: **AMTWEAK**
Select part/subassembly to tweak: **[Select the lower casing.]**

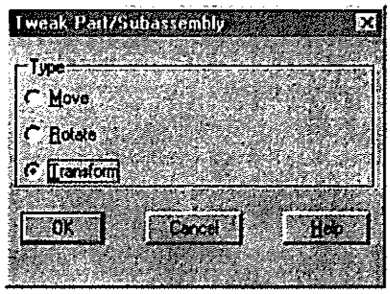


Figure 4.91 Tweak Part/Subassembly dialog box

eXit/Rotate/<Move>: **M**
Direction Viewdir/Wire/X/Y/Z/<Start point>: **Z**
Distance: **-60**
Flip/<Accept>: [**Accept if the arrow is pointing downward.**]
eXit/Rotate/<Move>: **X**

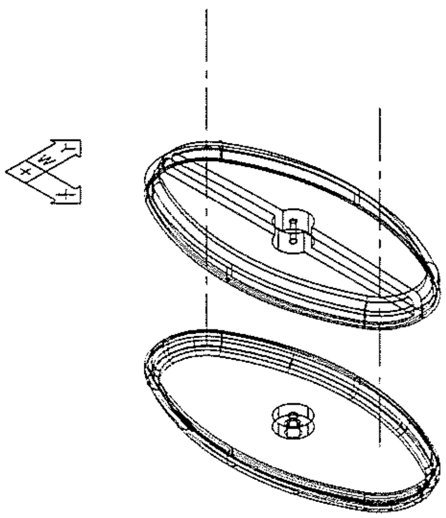


Figure 4.92 Solid parts tweaked apart

Now the lower casing is tweaked a distance of 60 units in the –Z direction. Note that the assembled solid parts are not moved apart, although they are tweaked in the scene. To find out whether the solid parts are still assembled properly, select the Assembly tab to go back to part modeling mode and you will find that the solid parts are still intact. If you want to remove the tweaking effect, you can switch back to scene mode and use the AMDELTWEAKS command.

<Assembly> **<Exploded Views>** **<Remove Tweaks>**

Now activate another new scene by selecting the New Scene item of the Scene cascading menu of the Assembly pull-down menu. (See Figure 4.93.)

<Assembly> **<Scene>** **<New Scene>**

Command: **AMNEW**
Create a new Scene of Active Assembly (REMOTE) named: **REMOTE02**

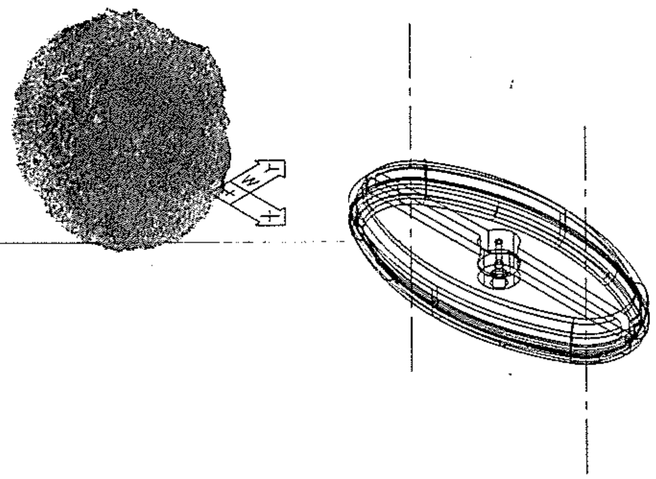


Figure 4.93 Second scene with no tweaking

A scene is an illustration of the assembly. Therefore, tweaking the parts in a scene does not affect the assembly or other new scenes. Now you have two scenes in the assembly, a scene with parts tweaked apart and a scene without any tweaking. To activate a scene, you can double-click the scene in the browser. (See Figure 4.94.)

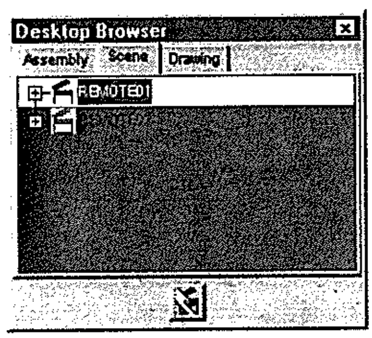


Figure 4.94 Scene tab showing two scenes in the assembly

Save the drawing.

Four-Bar Linkage Project (Continued)

Earlier in this chapter, you applied assembly constraints to four linkage bars and four pivot pins to assemble them. Now you will construct scenes of exploded views of this model. Open the file that you saved earlier.

<File> **<Open... >**

File name: **4bar.dwg**

In the browser, select the Scene tab. Place the mouse cursor on the background and then press the right mouse button to activate a new scene.

Create a new Part/Scene/subAssembly/<Instance>: **SCENE**
Create a new Scene of Active Assembly (4BAR) named: **4BAR01**

To set the explosion factor, select the name of the scene in the browser. Then press the right mouse button to bring up the pop-up menu. (See Figure 4.95.) After that, select the Explode Factor... item. (See Figure 4.96, the Explode Factor dialog box.) Set the explosion factor to 50. This will set all the assembled solid parts in the scene to a distance of 50 units apart. (See Figure 4.97.)

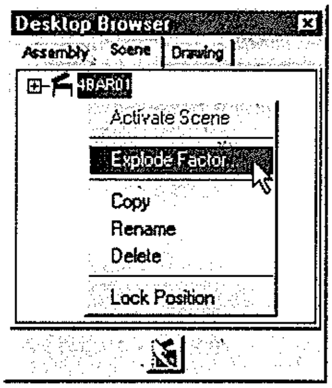


Figure 4.95 Activating the Explode Factor dialog box

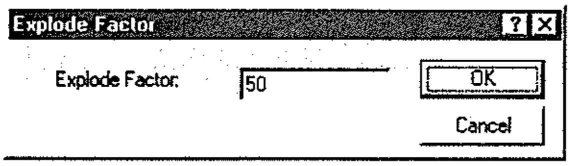


Figure 4.96 Explode Factor dialog box

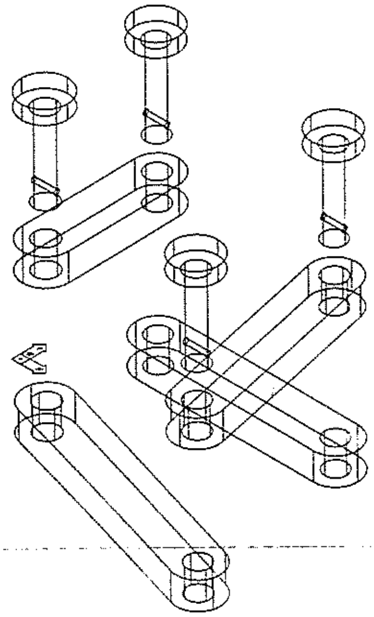


Figure 4.97 Explosion factor set in the scene

To change the explosion factor, you can select Scene Explosion Factor from the Exploded View cascading menu of the Assembly menu. Set it to 70. (See Figure 4.98.)

<Assembly> <Exploded Views> <Scene Explosion Factor>

Command: **AMXFACTOR**
Change Explosion factor: sCene/<Select part or subassembly>: **SCENE**
Scene to change (or ?): **4BAR01**
Reset/Enter new explosion factor for "4BAR01": **70**

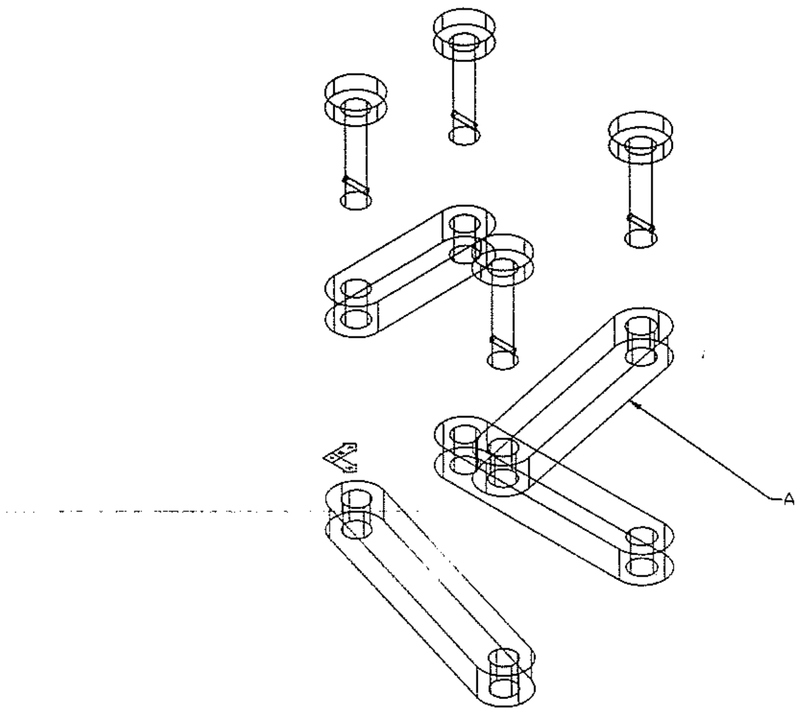


Figure 4.98 Explosion factor changed

In addition to changing the explosion factor for all the solid parts collectively, you can change the explosion factors individually. (See Figure 4.99.)

<Assembly> **<Exploded Views>** **<Part Explosion Factor>**

Command: **AMXFACTOR**
Change Explosion factor: sCene/<Select part or subassembly>: **SELECT**
Select part or subassembly: **[Select A (Figure 4.98).]**
Reset/Enter new explosion factor for "LINKAGE_A_2"<70>: **90**

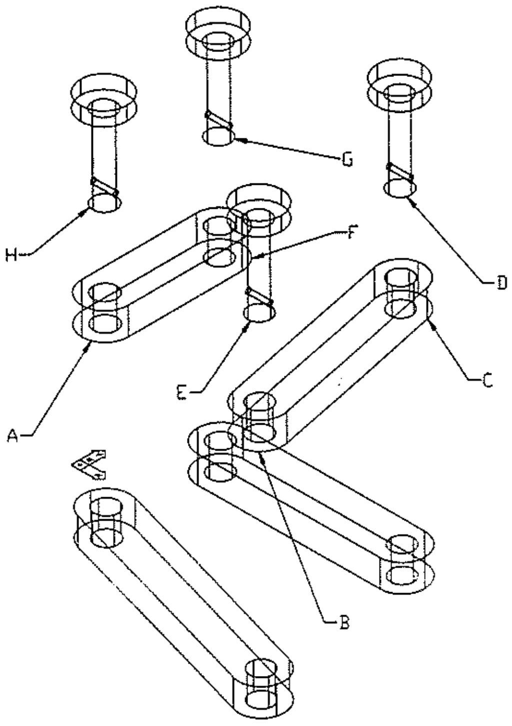


Figure 4.99 Explosion factor of an individual part changed

To see how the parts are related to each other while they are exploded, you can add trail lines by using the AMTRAIL command. (See Figure 4.100, the Trail Offsets dialog box.) Select Under Shoot and set the values to 5 units. Then select the [OK] button. This way, a trail line is added. Repeat the AMTRAIL command to add trail lines to B, C, D, E, F, G, and H (Figure 4.99). (See Figure 4.101.)

<Assembly> **<Exploded Views>** **<Create Trail...>**

Command: **AMTRAIL**
Select reference point on part/subassembly: **[Select A (Figure 4.99).]**

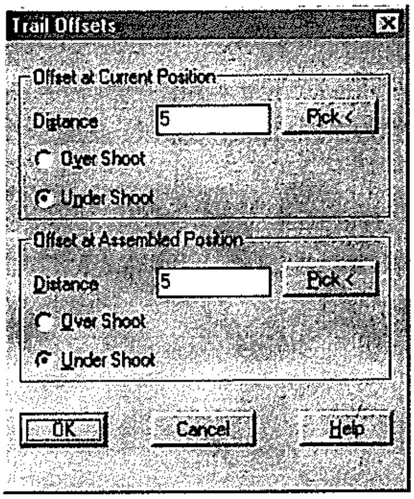


Figure 4.100 Trail Offsets dialog box

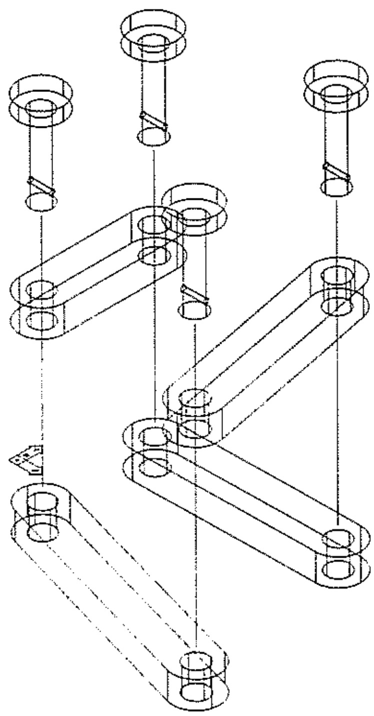


Figure 4.101 Trail lines added

Note that in Figure 4.101 the default linetypes for the trail lines are continuous lines. Because the trail lines reside on layer AM_TR, you can change their linetype by using the LAYER command to change the linetype of layer AM_TR to Center. To obtain an appropriate linetype proportion, use the LTSCALE command. (See Figure 4.102.)

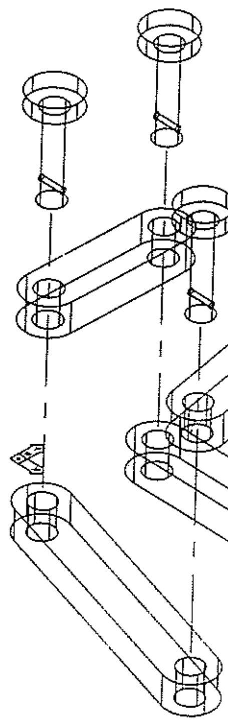


Figure 4.102 Trail lines linetype changed

The trail lines can be edited and deleted by using the AMEDITTRAIL and AMDELTRAIL commands.

<Assembly> <Exploded Views> <Edit Trail...>

Command: **AMEDITTRAIL**
Select trail to edit: **[Select the trail line that you want to edit.]**

<Assembly> <Exploded Views> <Delete Trail>

Command: **AMDELTRAIL**
Select trail to delete: **[Select the trail line that you want to delete.]**

Now construct another scene with no explosion. (See Figure 4.103.)

<Assembly> <Scene> <New Scene>

Create a new Scene of Active Assembly (4BAR) named: **4BAR02**

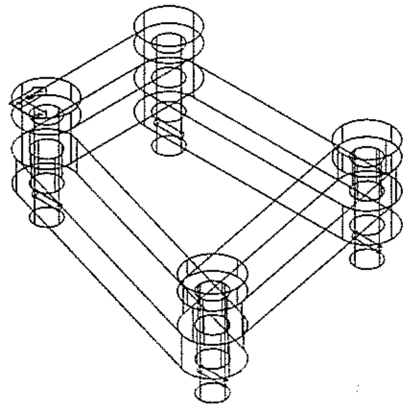


Figure 4.103 Scene with no explosion

Save your drawing.

<File> **<Save>**

Die Set for a Gear (Continued)

Open the drawing Dieset.dwg that you saved earlier in this chapter.

<File> **<Open... >**

File name: **Dieset.dwg**

Set up two scenes, DIESET01 and DIESET02. Activate DIESET01 as the active scene. (See Figure 4.104.)

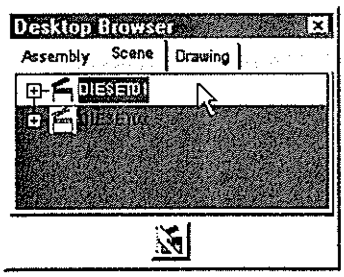


Figure 4.104 Two scenes constructed

Use the AMTWEAK command to rotate the part that is split from the main solid (the right side of the assembly). (See Figure 4.105.)

<Assembly> **<Exploded Views>** **<Add tweaks...>**

Command: **AMTWEAK**
Select part/subassembly to tweak: **[Select the right side of the assembly.]**

[Transform OK]

eXit/Rotate/<Move>: **R**
Center of rotation: **END** of **[Select A (Figure 4.105).]**
Direction Viewdir/Wire/X/Y/Z/<Start point>: **Y**
Flip/<Accept>: **[Accept if the direction of the arrow is upward.]**
Angle of rotation: **90**
eXit/Move/<Rotate>: **X**

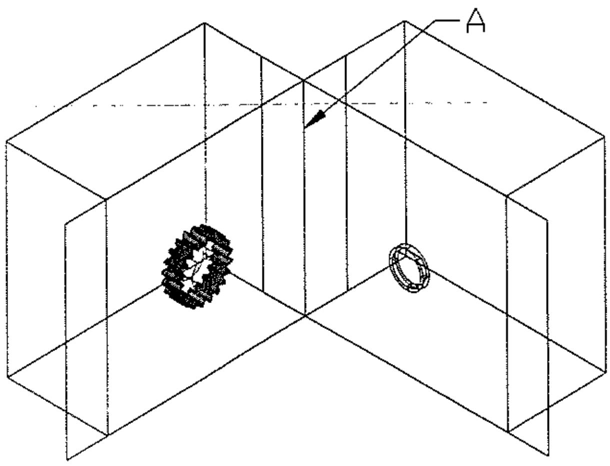


Figure 4.105 Top part rotated in a scene

You should recall that the gear used in this project is a table-driven solid part (linked to an external Excel spreadsheet). To further understand how the Excel spreadsheet works, select the Assembly tab of the browser to activate assembly mode. Then expand the graphics tree and select the Table (gear.xls) item in the browser (Figure 4.106). After that, double-click on the 101teeth5 item to change the gear. After that, select the Scene tab of the browser. (See Figure 4.107.)

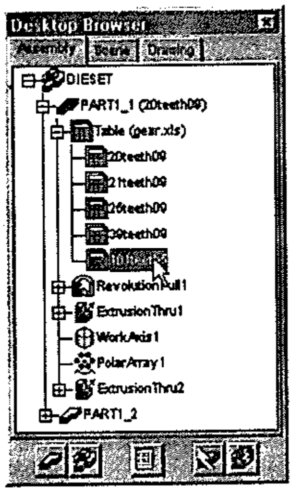


Figure 4.106 Selecting the Excel table in the browser

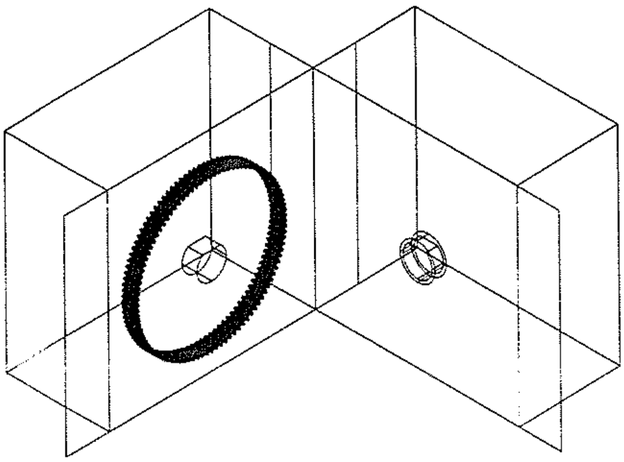


Figure 4.107 Upper part tweaked further

Save your drawing.

<File> **<Save>**

To sum up, you can set up a number of scenes after assembling the solid parts properly. Assembly scenes are places where you can explode the assembled solid parts in a variety of ways to illustrate how they are related to each other. There are two ways to explode the solid parts of an assembly apart: specifying an explosion factor and tweaking individual solid parts apart manually.

Scene Preferences

To set the system variables that are related to scenes of an assembly, you can select the Scene Preferences... item of the Assembly pull-down menu. Figure 4.108 shows the Scene tab of the Desktop Preferences dialog box. There are three entries. The Update Scene as Modified box determines whether the exploded solid parts are moved apart by applying the explosion factor or tweaking. The second box determines whether the parts are locked at their current positions if a suppressed feature is encountered. The third box sets the prefix for the names of the scenes.

<Assembly> **<Scene Preferences...>**

Command: **AMPREFS**

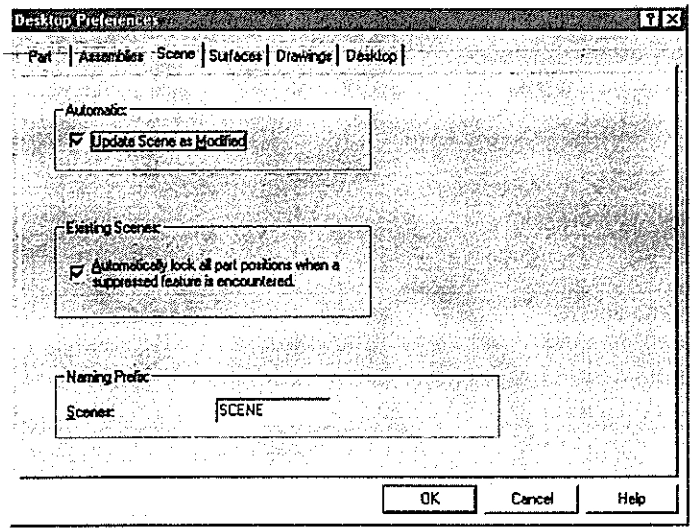


Figure 4.108 Scene tab of the Desktop Preferences dialog box

4.7 Assembly Modeling Utilities

To conclude this chapter, let us have a quick tour of some other assembly modeling utilities that we have not covered earlier.

Hatch Pattern and Attributes

Open the assembly drawing Remote.dwg that you saved earlier in this chapter.

<File> **<Open...>**

File name: **Remote.dwg**

While constructing a section view for an assembly of more than two component parts, you need to set a different hatch pattern for each of the solid parts. To set the hatch pattern, you can use the AMPATTERNDEF command. This command is applicable to instances of external solid parts in the assembly drawing. The hatch pattern of the original solid part is not affected. (See Figure 4.109, the Hatch Pattern dialog box.) Select the LCASE item from the Parts/Subassemblies list box. Then set the Pattern Properties appropriately. After that, select the [OK] button. This way, the hatch pattern for the solid part Lcase is set.

<Assembly> **<Assembly>** **<Hatch Patterns...>**

Command: **AMPATTERNDEF**

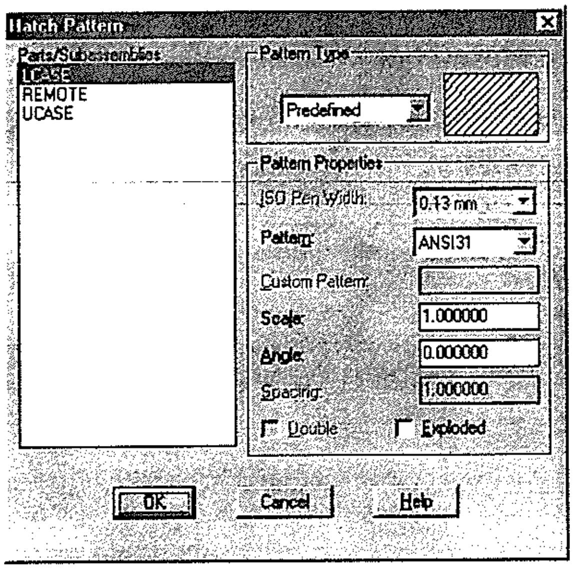


Figure 4.109 Hatch Pattern dialog box

Repeat the AMPATTERNDEF command. Select the UCASE item. Then set the pattern properties as follows:

[Pattern Properties
Pattern: **ANSI31**
Scale: **1**
Angle: **90**]

Now assign attributes to the solid parts. Attributes are textual information that is associated with each solid part. To define such information, you can use the AMASSIGN command. (See Figure 4.110, the Assign Attributes dialog box.) Select the LCASE item from the Parts/Subassembly definitions list box. Then select the [Add...] button. (See Figure 4.111, the Add new Attribute dialog box.) There are three kinds of attributes: String, Integer, and Real. Set the attributes appropriately. Then select the [OK] button.

<Assembly> **<Assembly>** **<Assign Attributes...>**

Command: **AMASSIGN**

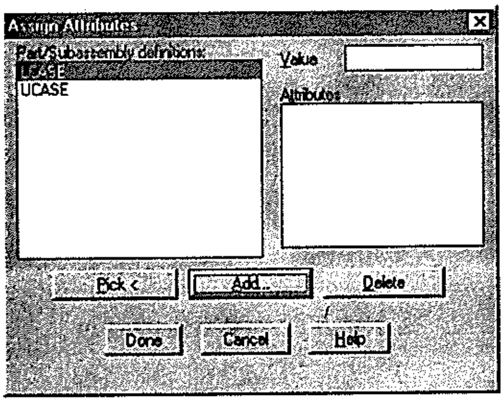


Figure 4.110 Assign Attributes dialog box

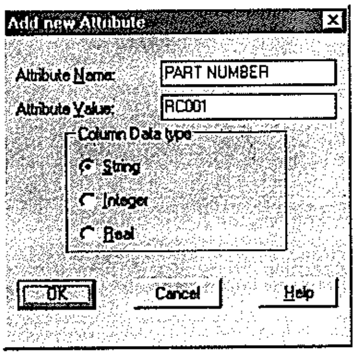


Figure 4.111 Add new Attribute dialog box

On returning to the Assign Attributes dialog box, select the UCASE item. Then set the attributes as follows:

[Attribute Name: **PART NUMBER**
Attribute Value: **RC002**
String
OK]

After setting the attributes, select the [OK] button and the [Done] button to exit. Now assignment of attributes is complete. When you construct a section view in the next chapter, these solid parts will have different hatch patterns.

Material Properties

To assign material types to the solid parts of an assembly and to evaluate the mass properties, use the AMASSMPROP command and select the solid parts. (See Figure 4.112, the Assembly Mass Properties dialog box.) Select the [Material...] button to assign a material type to the solid part. (See Figure 4.113, the Select Material dialog box.) Select Aluminum in the list box. Then select the UCASE item in the Part/Subassembly definition list box. After that, select the [Assign] button to assign the material to the solid part. This way, the material Aluminum is assigned to the upper casing. Now select the LCASE item in the Part/Subassembly definition list box. Then select Copper in the list box. After that, select the [Assign] button. The material Copper is assigned to the lower casing. Material assignment is complete. Select the [OK] button. Figure 4.114 shows the mass property results.

<Assembly> **<Analysis>** **<Mass Properties>**

Command: **AMASSMPROP**
Select parts/subassemblies Name/<Select>: **[Enter]**
Select part: **[Select the upper casing and the lower casing.]**
Select part: **[Enter]**

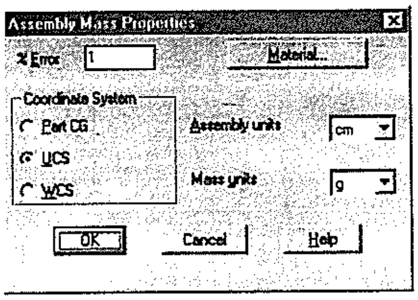


Figure 4.112 Assembly Mass Properties dialog box

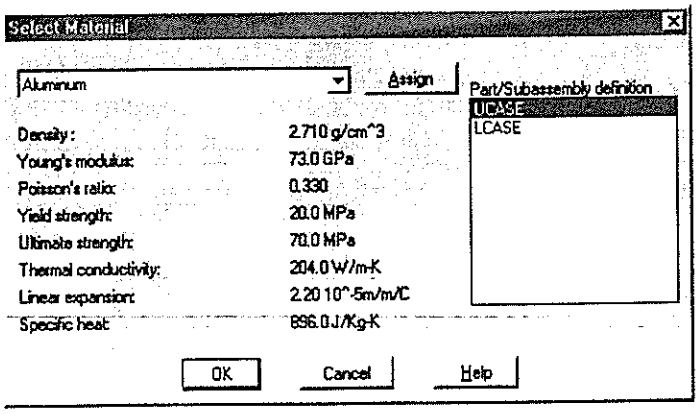


Figure 4.113 Select Material dialog box

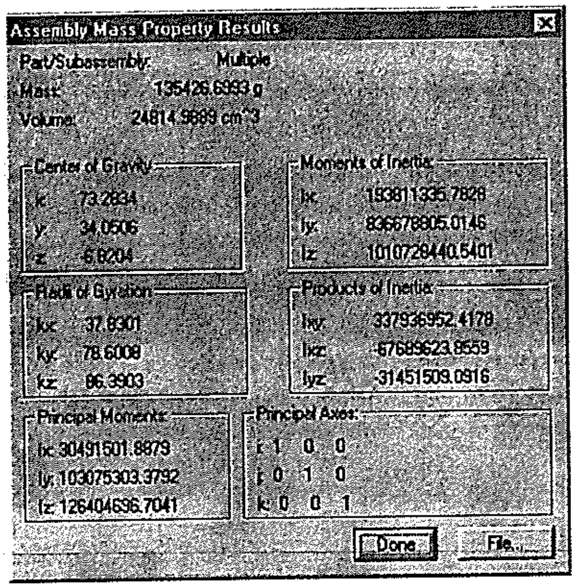


Figure 4.114 Assembly Mass Property Results dialog box

Save your drawing.

<File> **<Save>**

Refreshing External Solid Parts

In an assembly drawing, there are two kinds of solid definitions, local and external. While you are working on an assembly with external solid parts, the external parts may be modified by some other person. To refresh the external solid parts so that the latest updated versions of the solids are loaded to the drawing, use the AMREFRESH command.

<Assembly> **<Assembly>** **<Update...>**

Command: **AMREFRESH**

Minimum Distance

To measure the minimum distance between two objects, you can use the AMDIST command.

<Assembly> **<Analysis>** **<Minimum Distance>**

Command: **AMDIST**
Select first set: Objects/<Instance>: **[Select the upper casing.]**
Next instance: **[Enter]**
Select second set: Objects/<Instance>: **[Select the lower casing.]**
Next instance: **[Enter]**
Output: Line/<Display>: **[Enter]**

```
Minimum distance:<0.000000>
```

Output in Virtual Reality Modeling Language (VRML) Format

Virtual Reality Modeling Language (VRML) is a kind of file format that enables 3D objects to be viewed and manipulated on a Web page. To save selected parts in VRML format, you can use the AMVRMLOUT command. This command outputs a file with an extension of .wrl.

Command: **AMVRMLOUT**
Select objects: [**Select the objects to export.**]
Select objects: [**Enter**]

You can insert a WRL file into a Web page written in HTML format. To view the WRL file, you need VRML plug-in applications.

4.8 Key Points and Exercises

Mechanical Desktop has two kinds of solid part drawings: single-part drawings and multi-part drawings. In a single-part drawing, you can construct only one solid part. In a multi-part drawing, you can construct more than one solid part. Naturally, an assembly drawing is a multi-part drawing.

There are three design approaches to handling a set of parts for an assembly. In one approach, you can construct all the parts in individual drawing files (they can be single-part files or multi-part files) and attach them to an assembly drawing file. Because the parts are external to the assembly drawing, they are called external solid parts. In the second approach, you can construct all the parts in one drawing file. Because all the parts reside in the assembly drawing, they are called local solid parts.

The third approach is a hybrid of the first and second. The parts are divided into two sets. You construct one set in the assembly drawing file and construct the other in separate drawing files. Then you attach the separate files to the assembly.

External solid parts can be copied to the assembly drawing file to become local solid parts, and local solid parts can be exported to become external solid parts.

The common engineering practice for handling an assembly and its parts is to have an assembly drawing and a set of part drawings that are attached to the assembly drawing.

In the assembly drawing, the solid parts (either local or external) are related to each other by a set of assembly constraints. There are four kinds of assembly constraints: mate, flush, angle, and insert. To apply these commands, you can use the AMMATE, AMFLUSH, AMANGLE, and AMINSERT commands, respectively. To use these commands collectively, you can use the AMCONSTRAIN command. You can edit or delete the assembly constraints by using the AMEDITCONST command. To find out the number of degrees of freedom. you can display the CG (center of geometry) and DOF (degrees of freedom) symbols.

After assembling, you can check interference with the AMINTERFERE command. When you apply assembly constraints to the instances, they translate. If you do not want them to translate, you may use the AMPREFS command. After that, you can apply the AMASSEMBLE command to assemble the instances of parts together.

To see how parts are assembled, you can set up assembly scenes in which you can explode or tweak the solid parts apart. Explosion or tweaking does not affect the integrity of the solid parts in the assembly.

In the next chapter, you will construct associative engineering drawings from 3D NURBS surface models, 3D parametric solid parts, and 3D assemblies. Now work on the following exercises to enhance your knowledge.

Exercise 4.1

Outline the three approaches for constructing a set of solid parts of an assembly. Explain how these approaches can be implemented by using Mechanical Desktop.

Exercise 4.2

What are the key concepts of assembly modeling? How many degrees of freedom are there in a solid part? What are they? What kinds of constraints can you apply to paired solid parts?

Exercise 4.3

How can you establish an exploded assembly view from a set of properly assembled solid parts?

Exercise 4.4

Open the drawing Boxf.dwg that you constructed in Chapter 3 (Exercise 3.9). Use the AMCATALOG command to export the two solid parts to two single-part drawing files: Boxf1.dwg and Boxf2.dwg.

Exercise 4.5

Open the drawing Boxr.dwg that you saved in Chapter 3 (Exercise 3.10). Export the two solid parts to two drawing files: Boxr1.dwg and Boxr2.dwg.

Exercise 4.6

Figure 4.115 shows a turning arm assembly. It has four solid parts, which you constructed in the last chapter.

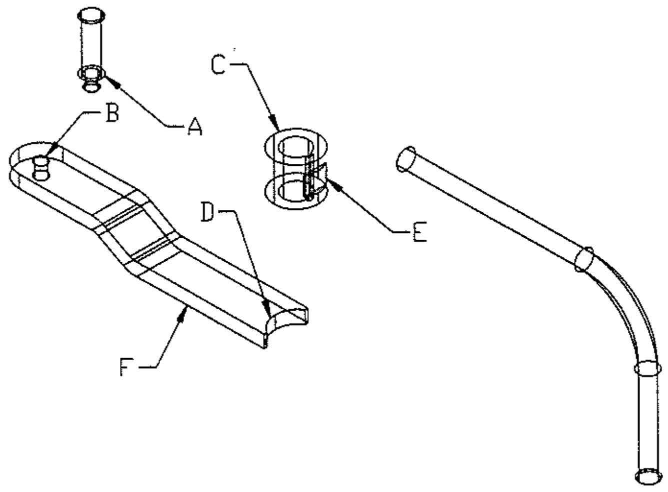


Figure 4.115 Turning arm assembly

Open the drawing Tarm.dwg that you saved in the last chapter.

<File> **<Open...>**

File name: **Tarm.dwg**

Use the AMINSERT command to constrain circular edge A (Figure 4.115) to circular edge B (Figure 4.115). Then repeat the AMINSERT command to constrain circular edge C (Figure 4.115) to circular edge D (Figure 4.115) with an offset value of 20 units. Note that the direction arrows should point in opposing directions.

After that, use the AMANGLE command to constrain vertical face E (Figure 4.115) to vertical face F (Figure 4.115) with an angle of 270°. (See Figure 4.116.)

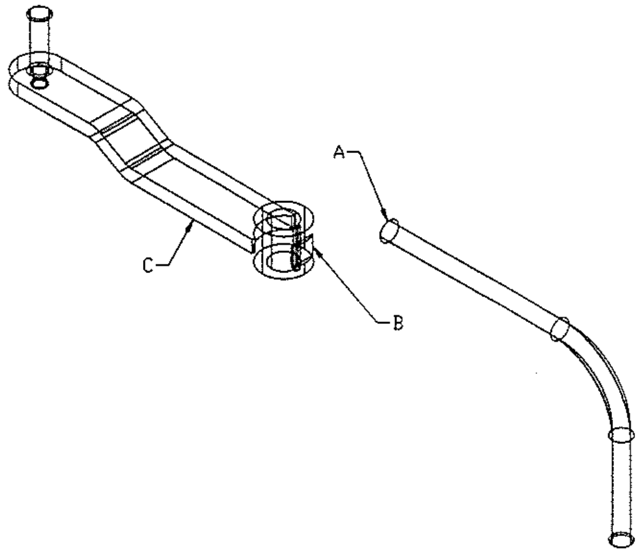


Figure 4.116 Three solid parts assembled

Now use the AMMATE command to apply constraints to vertical faces A and B (Figure 4.116), axis A and vertical face C (Figure 4.116) with an offset of –35 units, and axis A and horizontal face C (Figure 4.116) with an offset of –10 units. (See Figure 4.117.)

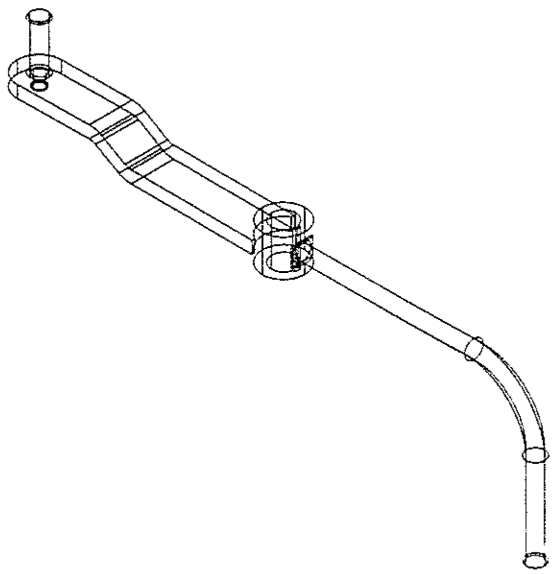


Figure 4.117 Solid parts assembled

The assembly is complete. Construct two scenes. Then tweak one of the scenes for a distance of 40 units as shown in Figure 4.118.

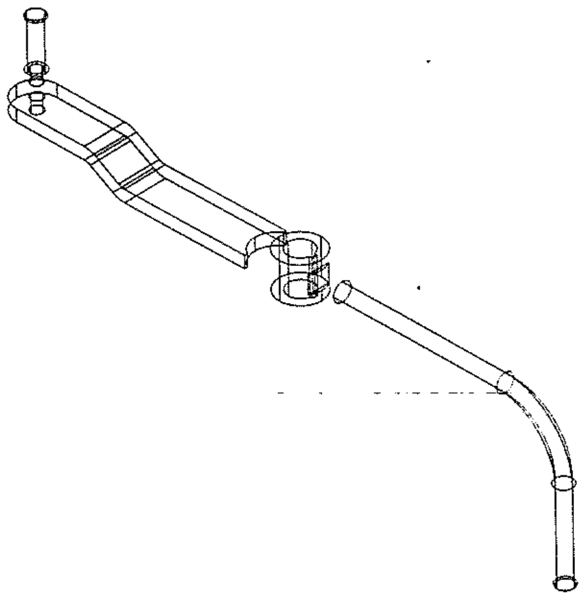


Figure 4.118 Assembled solid parts tweaked apart

Now use the AMCATALOG command to export the local solid parts to external drawing files. Then save your drawing.

Exercise 4.7

Figure 4.119 shows the assembly of a tire and a wheel of a scale model car.

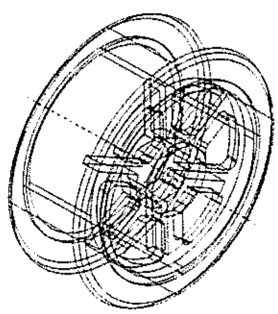


Figure 4.119 Tire and wheel assembly

Open the drawing Whl_tr.dwg that you saved in the last chapter. (See Figure 4.120.)

<File> **<Open...>**

File name: **Whl_tr.dwg**

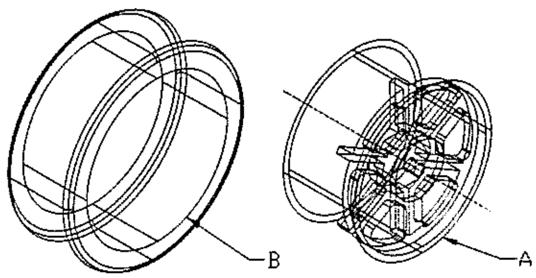


Figure 4.120 Wheel and tire

Use the AMINSERT command to constrain circular edge A and circular edge B (Figure 4.120) with opposite directions and an offset of 1 unit.

The assembly is complete. Use the AMCATALOG command to export the local solid parts. Then save your drawing.

Exercise 4.8

Figure 4.121 shows the rendered image of a shock absorber of a scale model car.

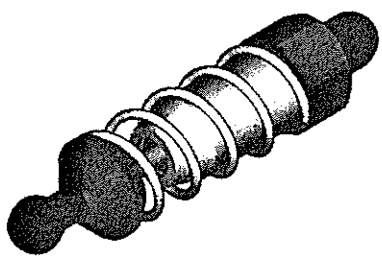


Figure 4.121 Shock absorber

Open the drawing Shocks.dwg that you constructed in Chapter 3 (Exercise 3.25).

<File> **<Open...>**

File name: **Shocks.dwg**

As we have mentioned, this is a multi-part drawing consisting of a number of solid parts for the shock absorber. In addition, you need two more solid parts: the shock absorber cylinder and the spring. To put them into this drawing, you can use the AMCATALOG command. Use the drawings Spring.dwg and Shockt.dwg. (See Figure 4.122.)

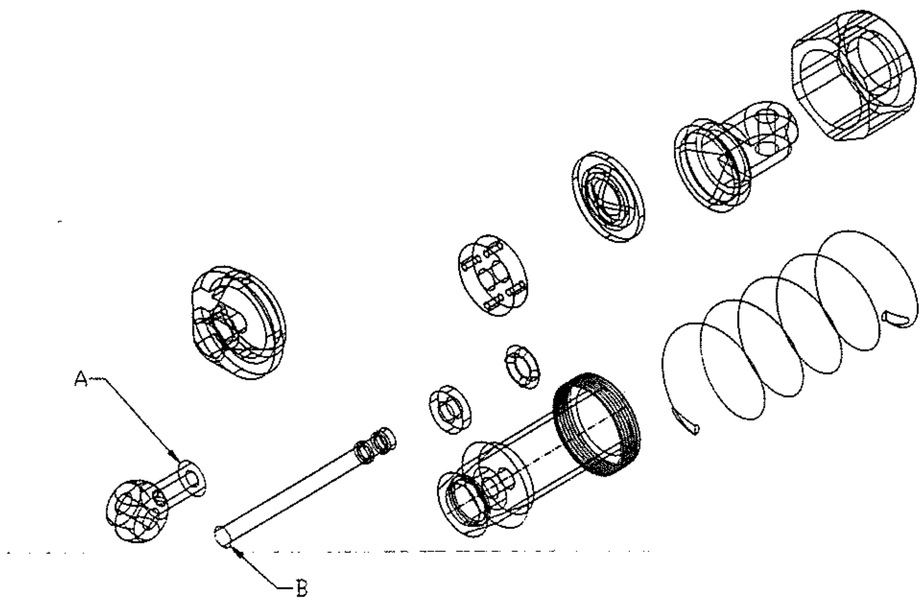
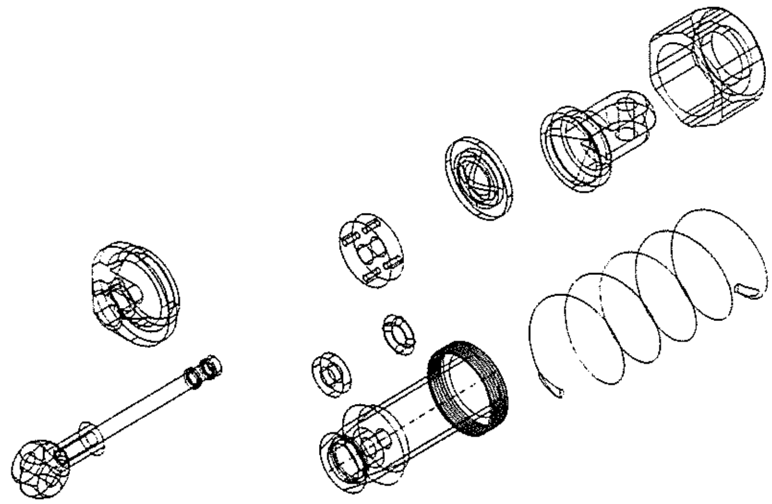


Figure 4.122 Solid parts of the shock absorber

Use the AMINSERT command to constrain the paired circular edges A and B (Figure 4.122) with an offset of –5 units. (See Figure 4.123.)



4.123 Circular edges constrained

Complete the assembly as shown in Figure 4.124. Then use the AMCATALOG command to export all the local solid parts. Then save your drawing.

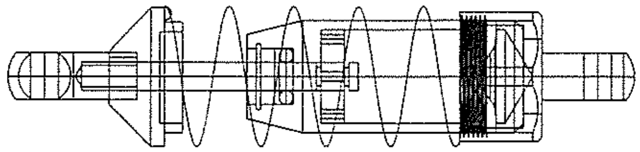


Figure 4.124 Solid parts properly assembled

Exercise 4.9

Figure 4.125 shows the top view and front view of a 1/10 scale model car.

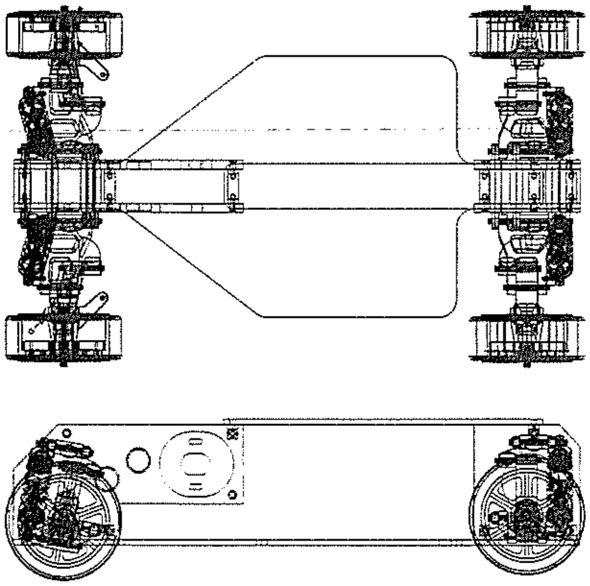


Figure 4.125 1/10 scale model car

Start a new multi-part drawing.

<File> **<New...>**

Use the AMCATALOG command to attach the drawing file Chassis_l.dwg (Exercise 3.7). (See Figure 4.126.)

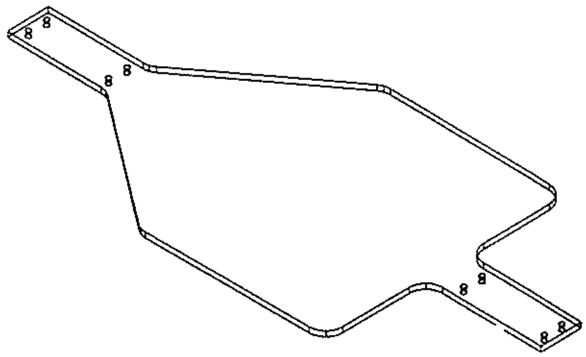


Figure 4.126 Lower chassis attached

Attach the drawing file Chassis_u.dwg (Exercise 3.6). (See Figure 4.127.)

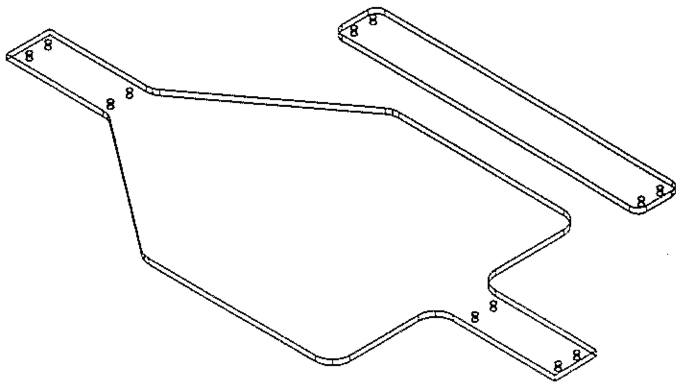


Figure 4.127 Upper chassis attached

Now attach six instances of the drawing file Mountg.dwg (Exercise 3.11). (See Figure 4.128.)

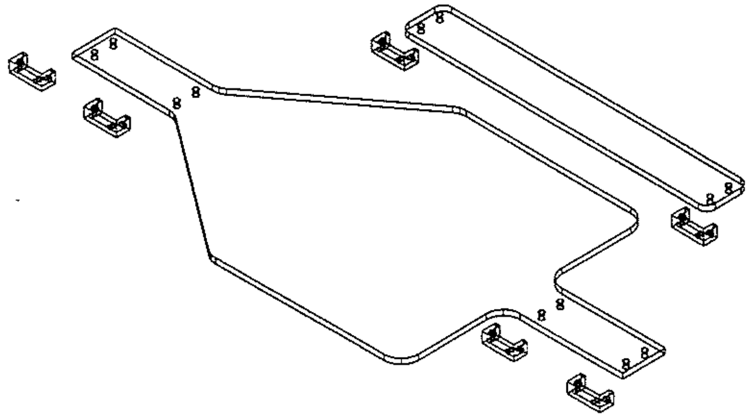


Figure 4.128 Mounting blocks attached

As shown in Figure 4.129, use the AMINSERT command to assemble four mounting blocks to the lower chassis and two mounting blocks to the upper chassis.

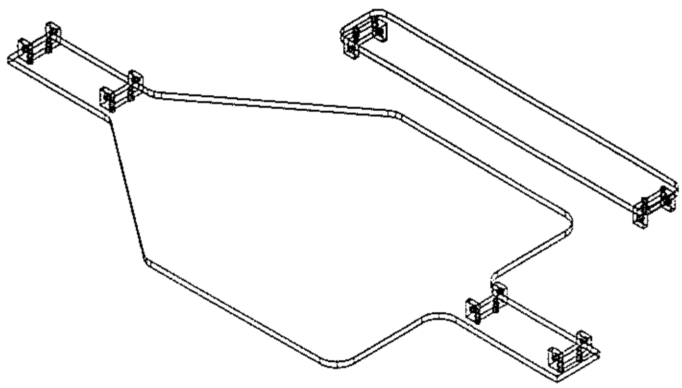


Figure 4.129 Mounting blocks assembled to the chassis plates

To avoid loss of data, save your drawing now and later from time to time.

<File> **<Save>**

File name: **Car.dwg**

Now construct a subassembly REAR_W and activate it. After activation, the screen becomes blank because it is switched to the new subassembly and there is not yet any solid part in it.

<Assembly> **<Assembly>** **<New Subassembly>**

Create a new Subassembly named: **REAR_W**

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **REAR_W**

Figure 4.130 shows the exploded view of this subassembly for your reference. It consists of a rear hub, an axle, a hexagon, two bearings, and the tire and wheel of a scale model car.

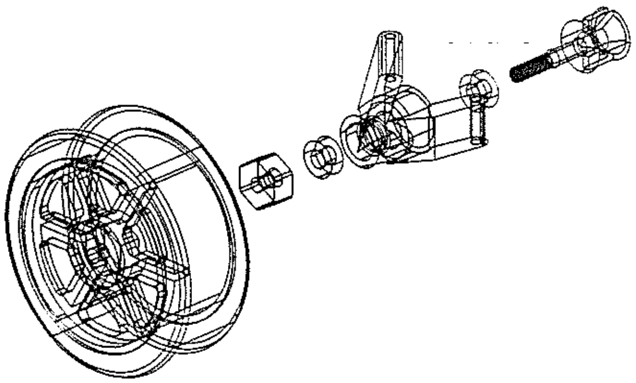


Figure 4.130 Exploded view of the rear hub and wheel subassembly

Use the AMCATALOG command to attach the following solid parts as shown in Figure 4.131:

Hub_r.dwg (Exercise 3.20)
Axle.dwg (Exercise 2.6)
Whl_tr.dwg (Exercise 4.7)
Hexblk.dwg (Exercise 3.14)
Bearing1.dwg (Exercise 3.4) (Two instances are required in the subassembly.)

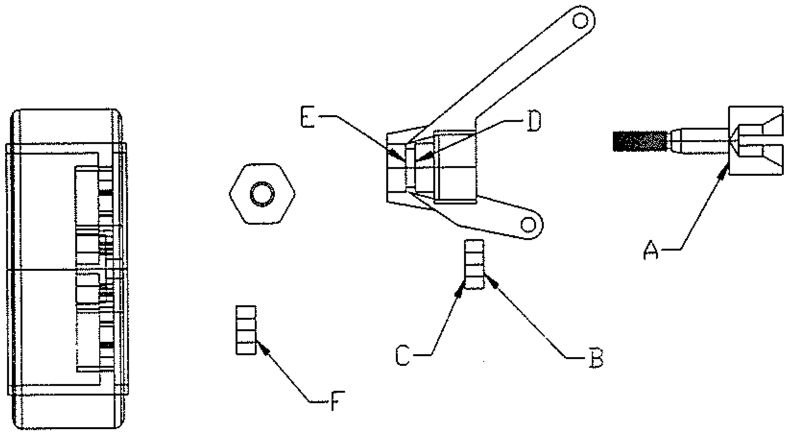


Figure 4.131 Solid parts attached to the subassembly

Use the AMINSERT command to constrain paired circular edges of A and B (Figure 4.131), C and D (Figure 4.131), and E and F (Figure 4.131). After that, use the DDVPOINT command to set the display to rotate 200° from the X axis and 25° from the XY plane. (See Figure 4.132.)

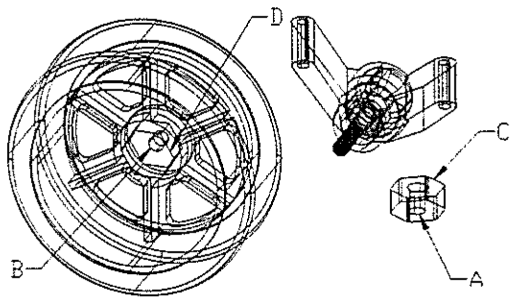


Figure 4.132 Bearings and axle constrained to the rear hub

Use the AMINSERT command to constrain circular edge A (Figure 4.132) to circular edge B (Figure 4.132). Then use the AMMATE command to constrain line C (Figure 4.132) to line D (Figure 4.132). Line D is the edge of the hexagonal slot of the wheel. After that, use the DDVPOINT command to set the display to rotate 200° from the X axis and 60° from the XY plane. (See Figure 4.133.)

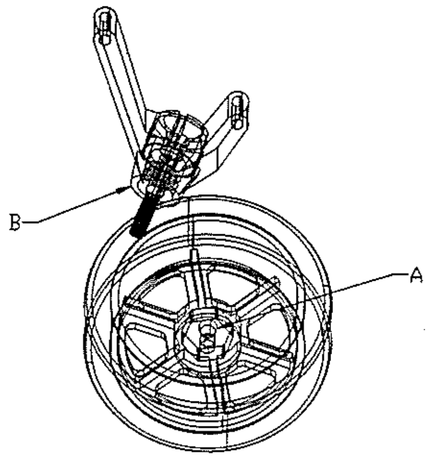


Figure 4.133 Wheel and hexagon assembled

Use the AMINSERT command to constrain circular edge A (Figure 4.133) to circular edge B (Figure 4.133). (See Figure 4.134.)

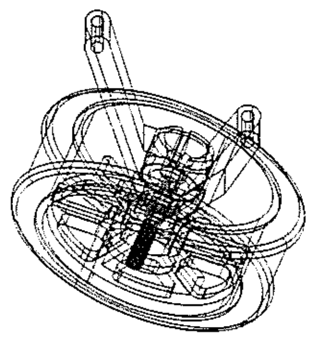


Figure 4.134 Subassembly completed

The subassembly is complete; you will now construct another subassembly. Figure 4.135 shows the exploded view of this subassembly. It has the front hub, axle, bearing, hexagon, and tire and wheel. Because the next subassembly is not a subassembly of this subassembly but a subassembly of the main assembly, you must make the main assembly active before you construct another subassembly, FRONT_W. Otherwise, you will get a subassembly of a subassembly.

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **CAR**

<Assembly> **<Assembly>** **<New Subassembly>**

Create a new Subassembly named: **FRONT_W**

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **FRONT_W**

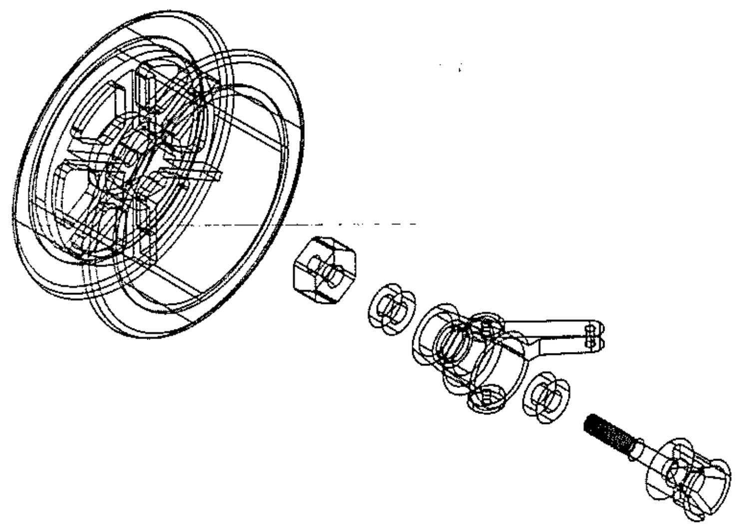


Figure 4.135 Exploded view of the front hub and wheel subassembly

As shown in Figure 4.136, use the AMCATALOG command to attach the following drawing files:

Hub_f.dwg (Exercise 3.19)
Axle.dwg (Exercise 2.6)
Whl_tr.dwg (Exercise 4.7)
Hexblk.dwg (Exercise 3.14)
Bearing1.dwg (Exercise 3.4) (Two instances are required in the subassembly.)

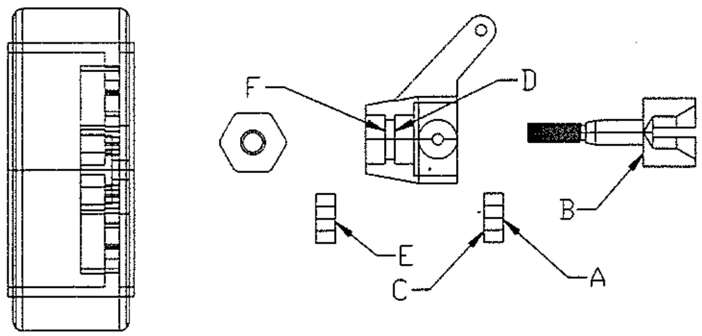


Figure 4.136 Solid parts attached to the subassembly

Use the AMINSERT command to constrain paired circular edges of A and B (Figure 4.136), C and D (Figure 4.136), and E and F (Figure 4.136). After that, use the DDVPOINT command to set the display to rotate 200° from the X axis and 25° from the XY plane. (See Figure 4.137.)

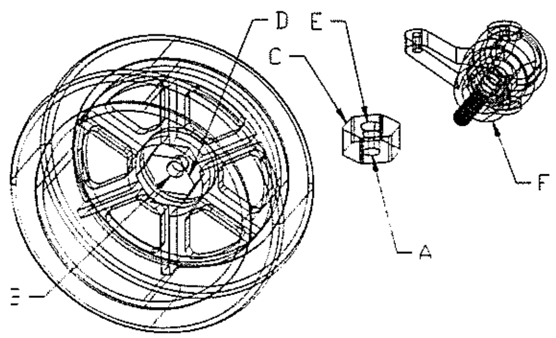


Figure 4.137 Bearings and axle constrained to the front hub

Use the AMPREFS command. Then uncheck the Update Assembly as Constrained box. Then use the AMMATE command on lines C and D (Figure 4.137). After that, use the AMINSERT command on circular edges A and B (Figure 4.137) and edges E and F (Figure 4.137). After constraining, use the AMASSEMBLE command and use the shortcut key [8] to display an isometric view. (See Figure 4.138.)

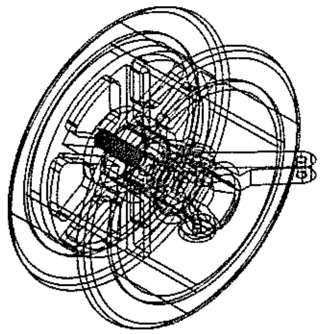


Figure 4.138 Front wheel hub subassembly completed

The subassembly is complete. Activate the main assembly and then construct another subassembly, FRONT_M, and activate it. Figure 4.139 shows the rendered image of this subassembly.

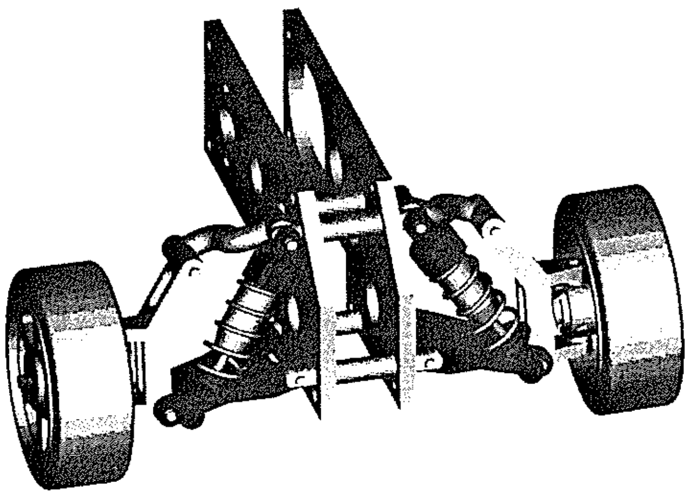


Figure 4.139 Rendered image of the front end subassembly

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **CAR**

<Assembly> **<Assembly>** **<New Subassembly>**

Create a new Subassembly named: **FRONT_M**

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **FRONT_M**

Use the AMCATALOG command to attach the following solid parts to the drawing. (See Figure 4.140.)

Solid Part File	Number of Instances
Boxf1.dwg (Exercise 4.4)	1
Boxf2.dwg (Exercise 4.4)	1
Hinge.dwg (Exercise 3.18)	4
Arm_u.dwg (Chapter 3)	2
Arm_l.dwg (Chapter 3)	2
Ubracket.dwg (Exercise 3.13)	2
Hpin1.dwg (Exercise 3.5)	4
Hpin2.dwg (Exercise 3.5)	2
Hpin3.dwg (Exercise 3.5)	2

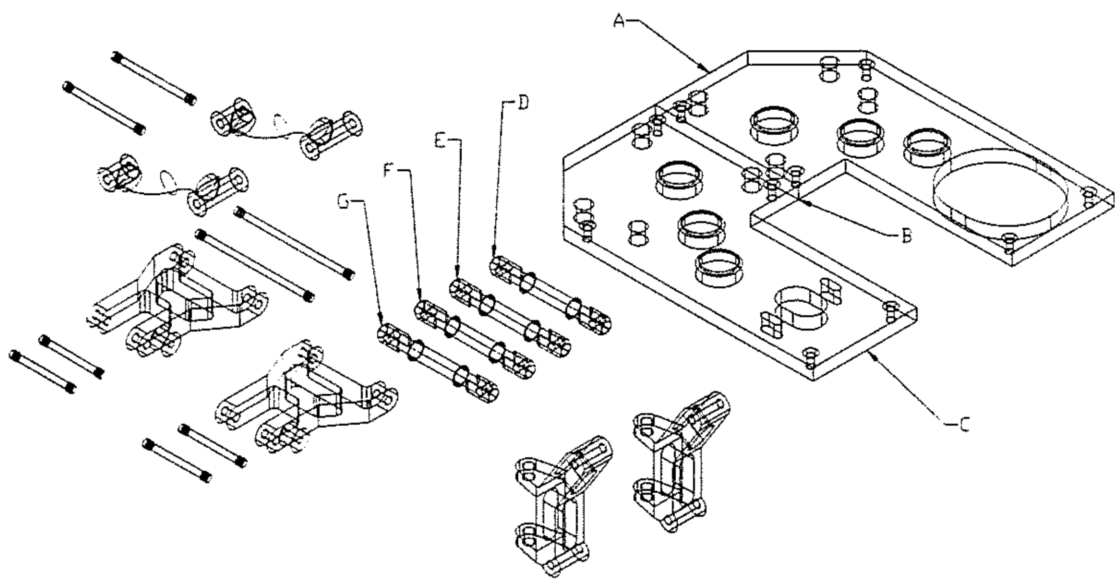


Figure 4.140 Solid parts attached to the subassembly

Use the ROTATE3D command to rotate solid A (Figure 4.140) about the X axis at B (Figure 4.140) for 90°. Then use the AMINSERT command to constrain the solids C, D, E, F, and G to A (Figure 4.140). (See Figure 4.141.)

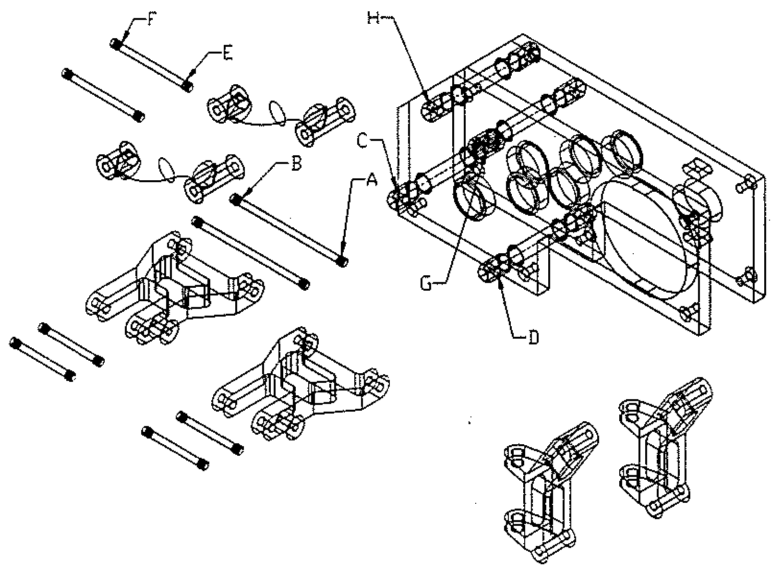


Figure 4.141 Suspension hinge pins and front mounting plates assembled

Use the AMINSERT command to constrain paired circular edges A and D (Figure 4.141) and E and G (Figure 4.141). Then use the AMMATE command to constrain paired axes B and C (Figure 4.141) and F and G (Figure 4.141). (See Figure 4.142.)

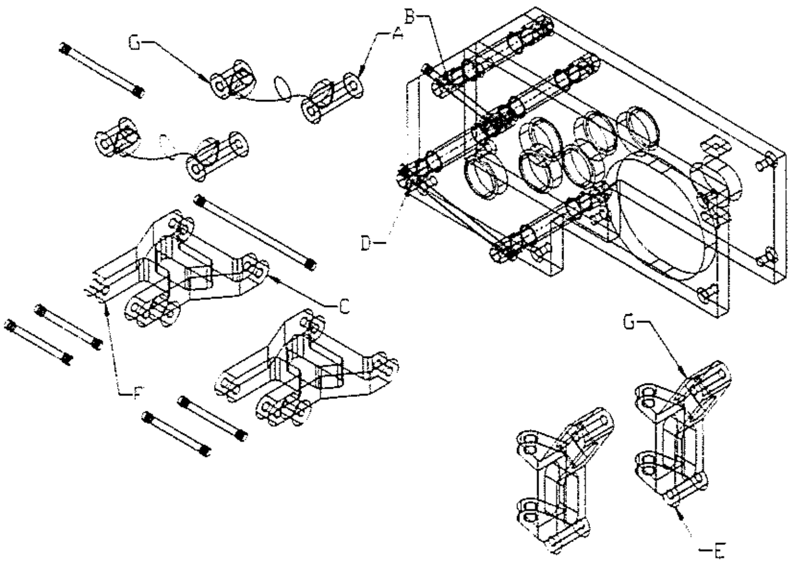


Figure 4.142 Hinge pins assembled

Use the AMINSERT command to constrain the paired circular edges A and B (Figure 4.142), C and D (Figure 4.142), E and F (Figure 4.142), and G and H (Figure 4.142). (See Figure 4.143.)

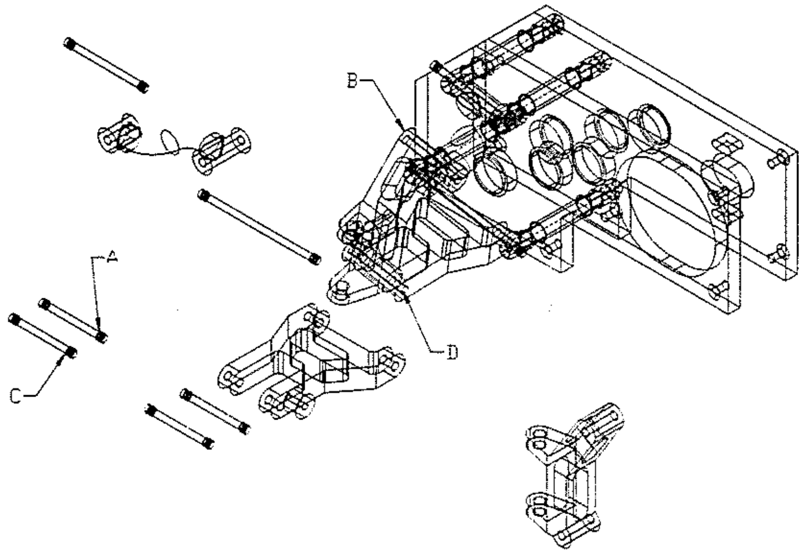


Figure 4.143 Suspension arms and U bracket assembled

Use the AMINSERT command to constrain the paired circular edges A and B (Figure 4.143) and C and D (Figure 4.143). (See Figure 4.144.)

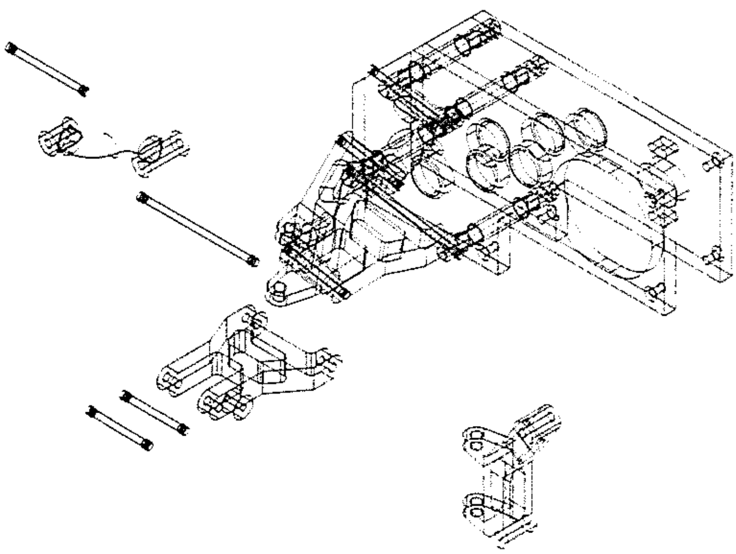


Figure 4.144 Hinge pins constrained

Set the display to a back left isometric view. Then complete the other side of the front mounting. (See Figure 4.145.)

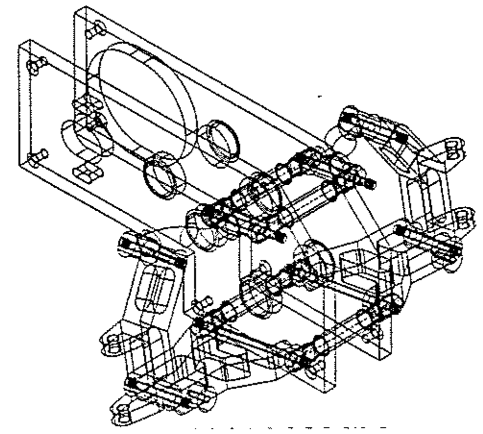


Figure 4.145 Parts assembled

Now use the AMCATALOG command to attach two instances of the drawing file Shocks.dwg (Exercise 4.6). (See Figure 4.146.)

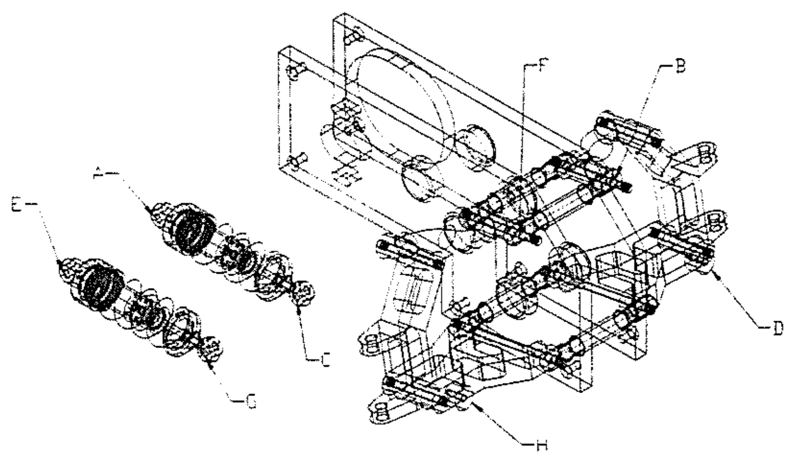


Figure 4.146 Shock absorbers instanced

Use the AMINSERT command to constrain the paired circular edges A and B (Figure 4.146), C and D (Figure 4.146), E and F (Figure 4.146), and G and H (Figure 4.146). Then use the AMCATALOG command to attach two instances of the local assembly definition FRONT_W that you constructed earlier in this exercise. (See Figure 4.147.)

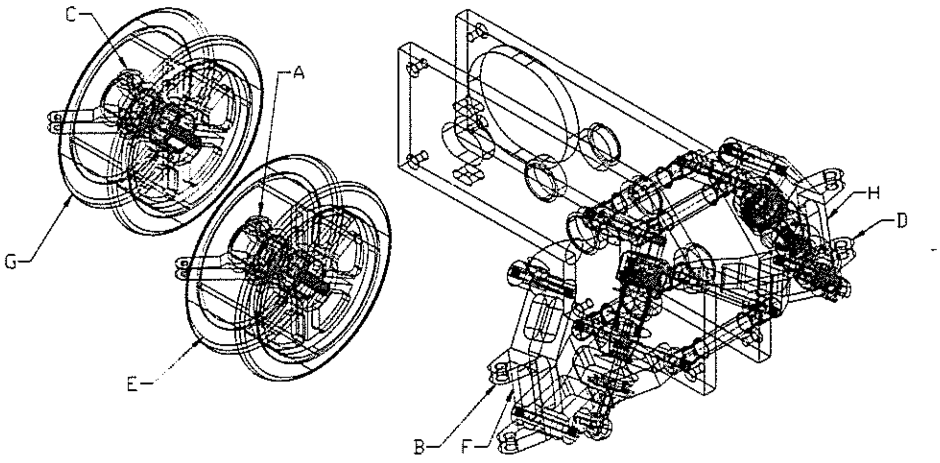


Figure 4.147 Shock absorbers constrained, front-wheel subassemblies instanced

Use the AMINSERT command to constrain paired circular edges A and B (Figure 4.147) and C and D (Figure 4.147). Then use the AMANGLE command to constrain paired vertical planes E and F (Figure 4.147) and G and H (Figure 4.147) to an angle of 180°. (See Figure 4.148.)

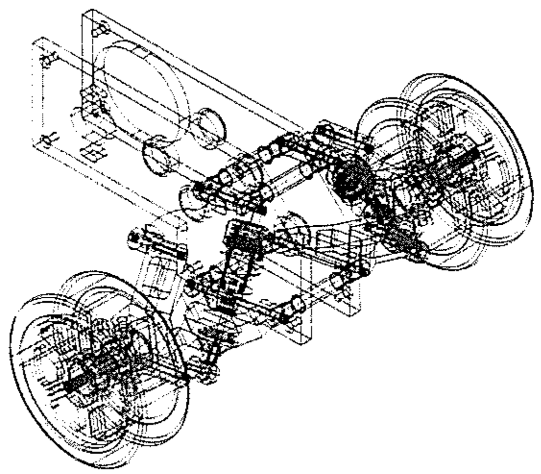


Figure 4.148 Subassembly completed

The subassembly is complete. Activate the main assembly and then construct a new subassembly, REAR_M, and activate it. Figure 4.149 shows the rendered image of this subassembly.

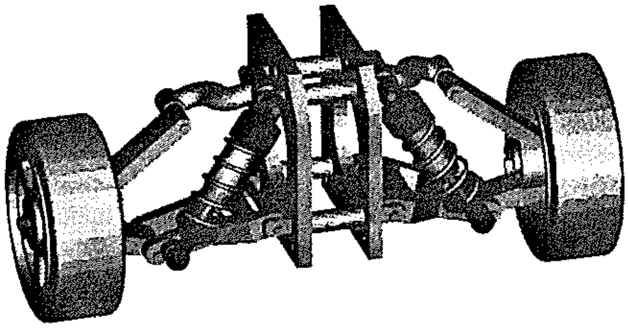


Figure 4.149 Rendered image of the rear mounting of the scale model car

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **CAR**

<Assembly> **<Assembly>** **<New Subassembly>**

Create a new Subassembly named: **REAR_M**

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **REAR_M**

Use the AMCATALOG command to attach the following solid parts to the drawing. (See Figure 4.150.)

Solid Part File	Number of Instances
Boxr1.dwg (Exercise 4.5)	1
Boxr2.dwg (Exercise 4.5)	1
Hinge.dwg (Exercise 3.18)	4
Arm_u.dwg (Chapter 3)	2
Arm_l.dwg (Chapter 3)	2
Hpin1.dwg (Exercise 3.5)	4
Hpin2.dwg (Exercise 3.5)	2
Hpin3.dwg (Exercise 3.5)	2

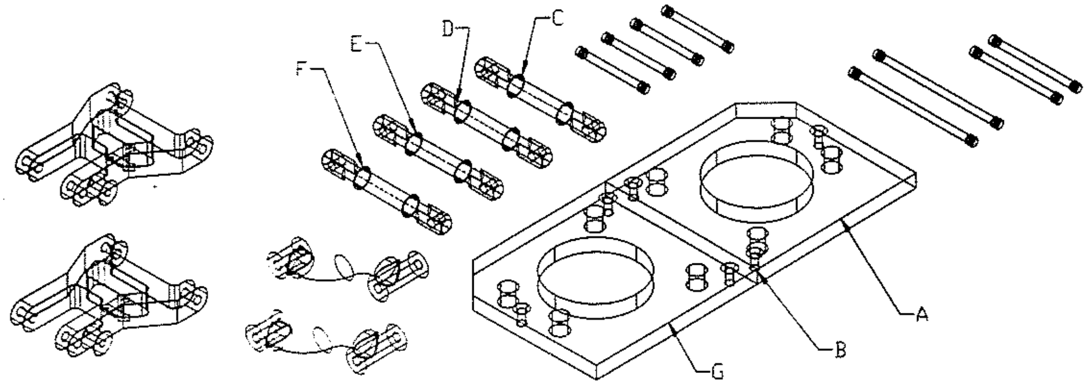


Figure 4.150 Solid parts attached

Rotate solid A (Figure 4.150) 90° about the X axis at B (Figure 4.150). Then use the AMINSERT command to constrain solid parts C, D, E, F, and G to A (Figure 4.150). (See Figure 4.151.)

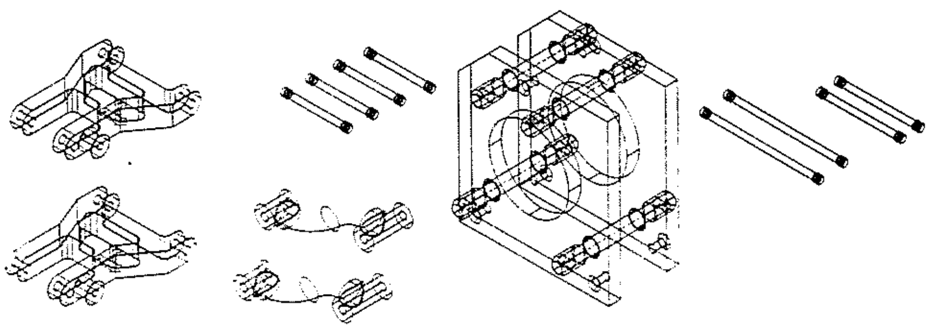


Figure 4.151 Suspension hinge pins assembled

Use the AMINSERT command to assemble all the solid parts together as shown in Figure 4.152.

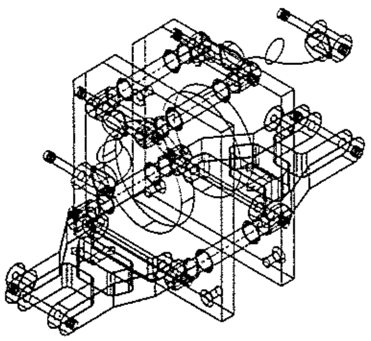


Figure 4.152 Solid parts assembled

Set the display to a back left isometric view. Then use the AMCATALOG command to attach two instances of the subassembly REAR_W and two instances of the drawing Shocks.dwg (Exercise 4.8). (See Figure 4.153.)

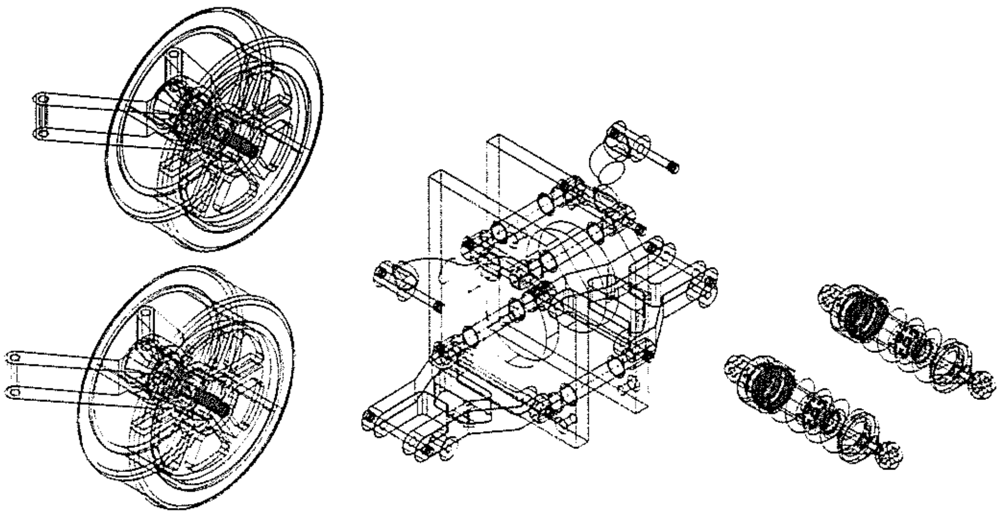


Figure 4.153 Display set, solid attached

Complete the subassembly as shown in Figure 4.154.

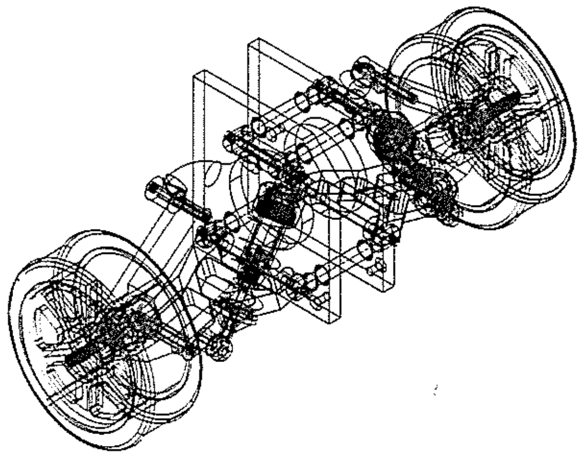


Figure 4.154 Subassembly completed

The subassembly is complete. Activate the main assembly. (See Figure 4.155.)

<Assembly> **<Assembly>** **<Make Active>**

Command: **AMACTIVATE**
Activate: Assembly/Scene/<Part>: **ASSEMBLY**
Activate assembly (or ?): **CAR**

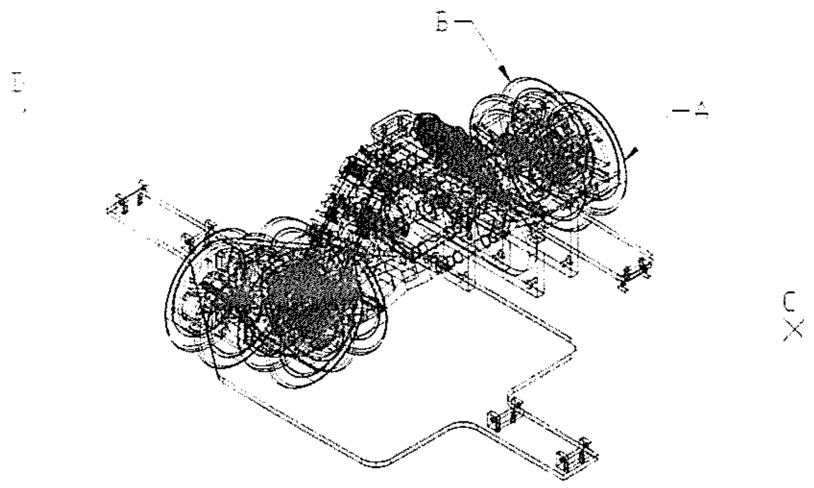


Figure 4.155 Main assembly activated

Now you have all the subassemblies displayed. On your screen, they may overlap. Use the MOVE command to translate A and B (Figure 4.155) to locations C and D (Figure 4.155). (See Figure 4.156.)

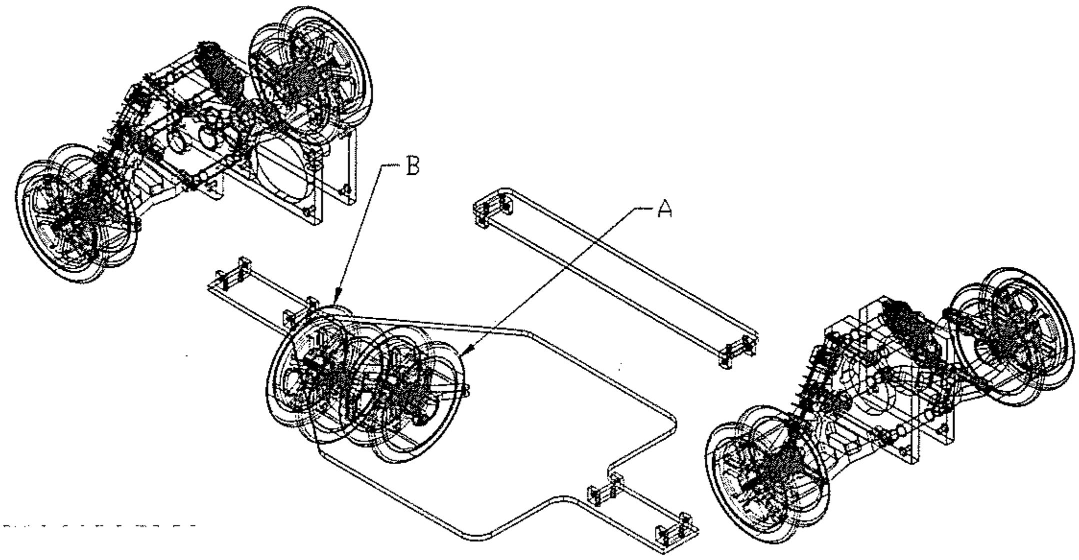


Figure 4.156 Subassemblies translated

The two instances A and B (Figure 4.156) are not required in the main assembly because they are already instanced in the subassemblies. Delete them. (See Figure 4.157.)

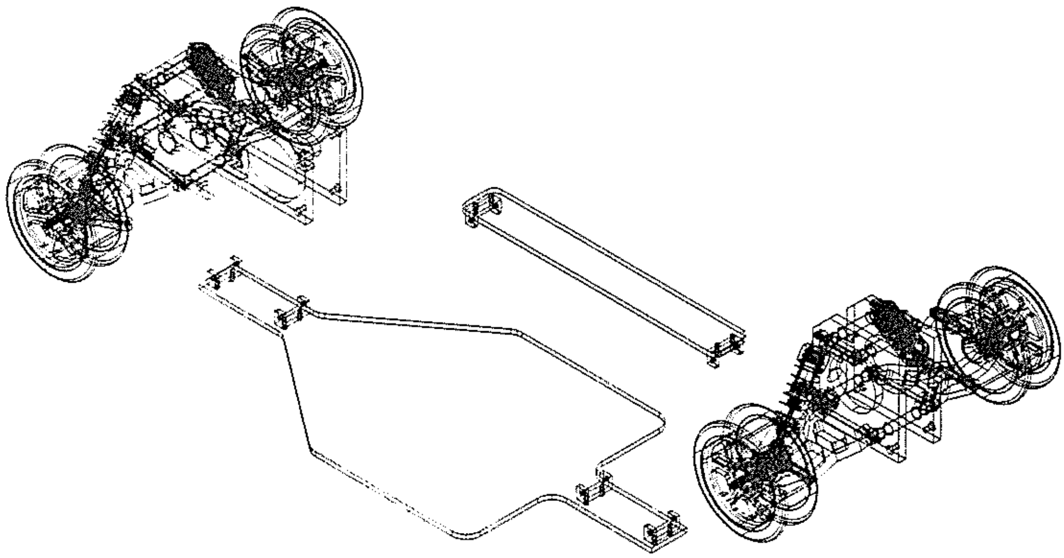


Figure 4.157 Unwanted instances deleted

Complete the assembly as shown in Figure 4.158.

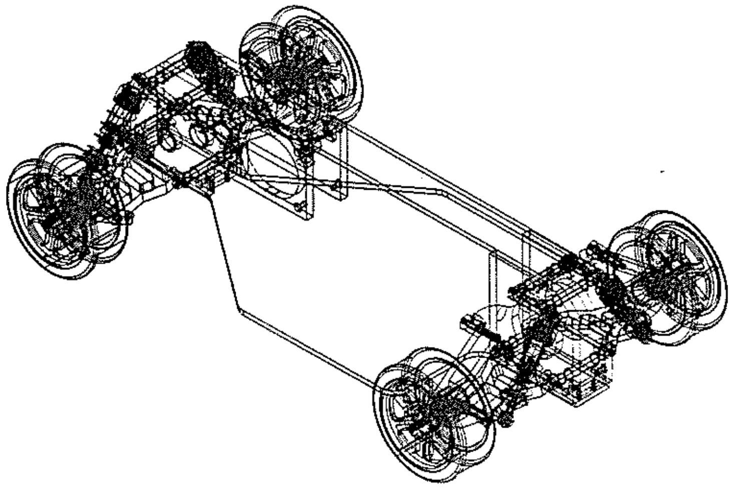


Figure 4.158 Complete assembly

The assembly is complete. Save your drawing. In Chapter 2, you constructed a number of car bodies. Figures 4.159 and 4.160 show the solid parts together with these car bodies.

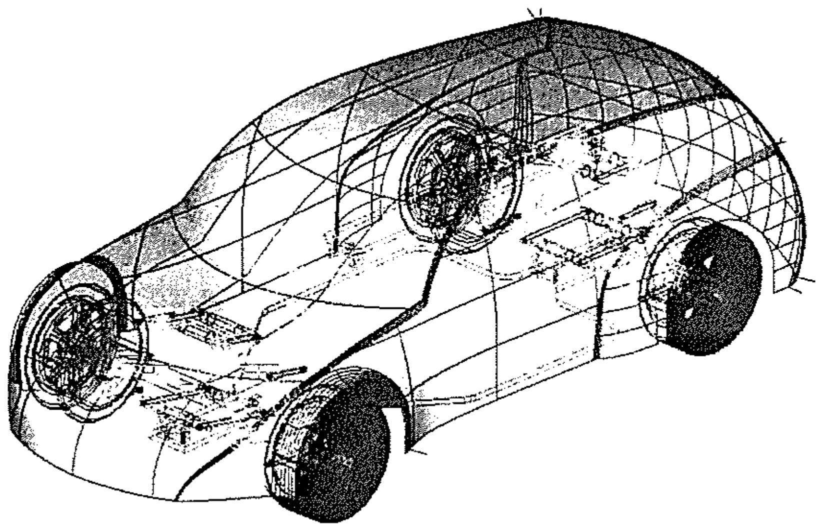


Figure 4.159 Assembly and car body

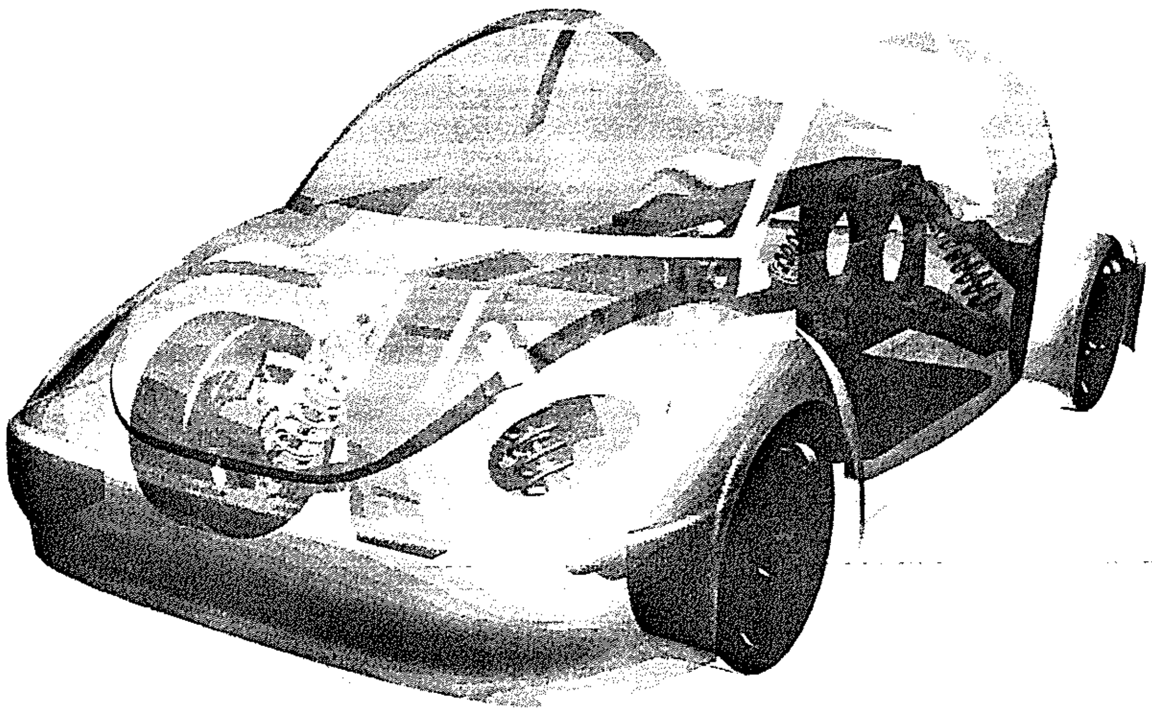


Figure 4.160 Assembly and car body

Chapter 5

Associative Drafting

Aims and Objectives

The aims of this chapter are to introduce the key concepts of associative drafting; to explain the standard procedures in engineering drafting; to illustrate how dimensions and annotations are incorporated in a drawing; to familiarize you with the use of surface texture symbols, welding symbols, and tolerance symbols; to explain the ways to set up a bill of materials; and to output a 2D drawing from a 3D model. After studying this chapter, you should be able to

- Describe the key concepts of associative drafting
- Construct associative engineering drawings from 3D models
- Use the dimensioning and annotation tools in engineering documentation
- Apply surface texture symbols, welding symbols, and tolerance symbols
- Set up and construct a bill of materials
- Output 2D engineering drawings from 3D objects

Overview

Mechanical Desktop has four engineering design tools. In Chapters 2, 3, and 4, you learned ways to construct 3D NURBS surface models, parametric solid models, and virtual assemblies. Here you will learn how to use the fourth tool, the associative drafting tool.

A NURBS surface model can serve many purposes. You can use it to manufacture the component described by the model, or to use a surface to cut a native solid or a parametric solid. Sometimes you may simply want to construct a 2D engineering drawing from it.

A solid model consists of edge data, surface data, and volume data. You can use a solid model for many purposes in manufacturing, such as computer-aided manufacturing, rapid prototyping, and so on. However, there are occasions when you have to communicate a 3D design in a conventional way by outputting a 2D engineering drawing.

By using virtual assembly, you can verify the validity of individual solid parts and the integrity of all the solid parts. As with 3D surfaces and 3D solids, you can output a 2D engineering drawing. In addition, you can generate a bill of materials and exploded views.

By using the associative drafting tool, you can output 2D engineering drawings from 3D NURBS surfaces, 3D parametric solids, and 3D virtual assemblies. The drawings generated from the 3D objects and the 3D objects associate to each other bidirectionally. If you update a 3D object, the drawing changes. If you change a parametric dimension (of a solid) in a drawing, the solid part is also modified.

This chapter will introduce you to the standard procedures for preparing engineering drawings and guide you in producing orthographic engineering drawings from 3D objects that you constructed in Chapters 2, 3, and 4.

5.1 Associative Drafting Concepts

Mechanical Desktop, like AutoCAD, has two working environments in which you can place entities: model mode and drawing mode. Model mode is the environment in which you construct the main constituents of the drawing — 3D NURBS surfaces, 3D parametric solids, or 3D virtual assemblies. Drawing mode is the environment in which you prepare an engineering drawing document from the 3D Mechanical Desktop objects.

To toggle between the two working environments, you can use the AMMODE command or select the appropriate tab of the Desktop Browser. By selecting the Drawing tab, you switch to drawing mode. By selecting the Assembly tab (multi-part drawing) or the Part tab (single-part drawing), you switch to model mode. (See Figure 5.1.)

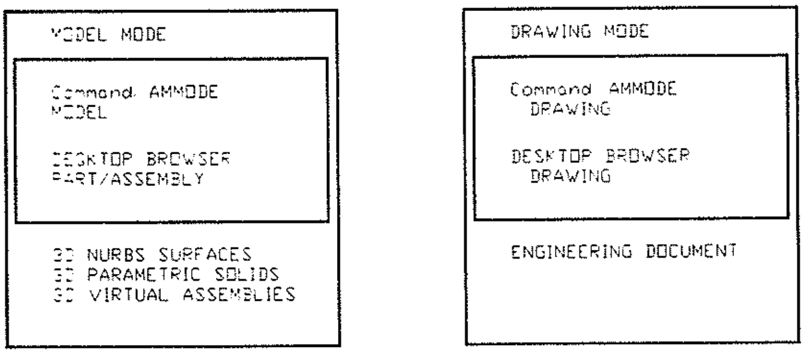


Figure 5.1 Model mode and drawing mode

Figure 5.2 shows the drawing tabs of the multi-part and single-part drawing browsers. There are two buttons in each of these tabs. The left button causes the drawing views to be updated automatically if the objects constructed in model mode are changed. The right button causes the drawing views constructed in drawing mode to display hidden lines.

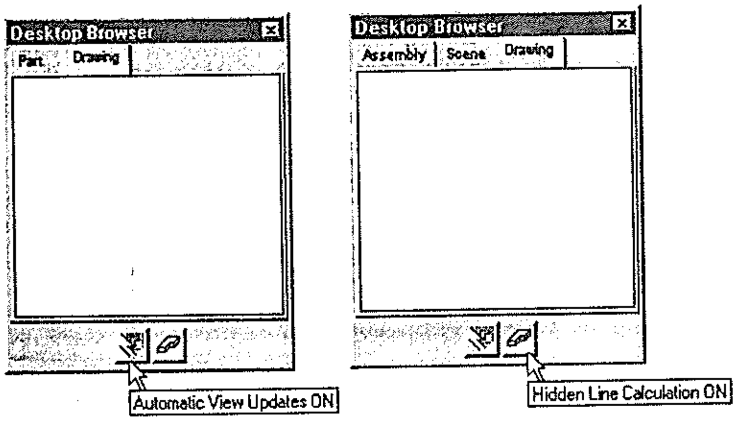


Figure 5.2 Drawing tabs of the Desktop Browser

After you enter drawing mode, the objects you constructed in model mode disappear, because the two working modes are individual working environments. To remind you that you are working in drawing mode, the UCS icon changes to a set square. (See Figure 5.3.)

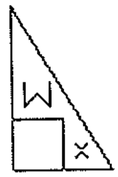


Figure 5.3 UCS icon to depict working in the paper space environment

The process of preparing an engineering drawing document involves two major tasks: adding a title block and constructing floating viewports in drawing mode. (See Figure 5.4.)

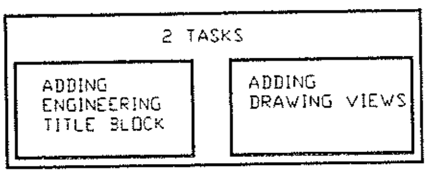


Figure 5.4 Two major tasks

In Mechanical Desktop, there are six kinds of drawing views: base view, orthographic view, auxiliary view, isometric view, detailed view, and broken view. Figure 5.5 shows the six options of the Create Drawing View dialog box. The images shown in the dialog boxes illustrate the kinds of drawing views that you can construct in drawing mode.

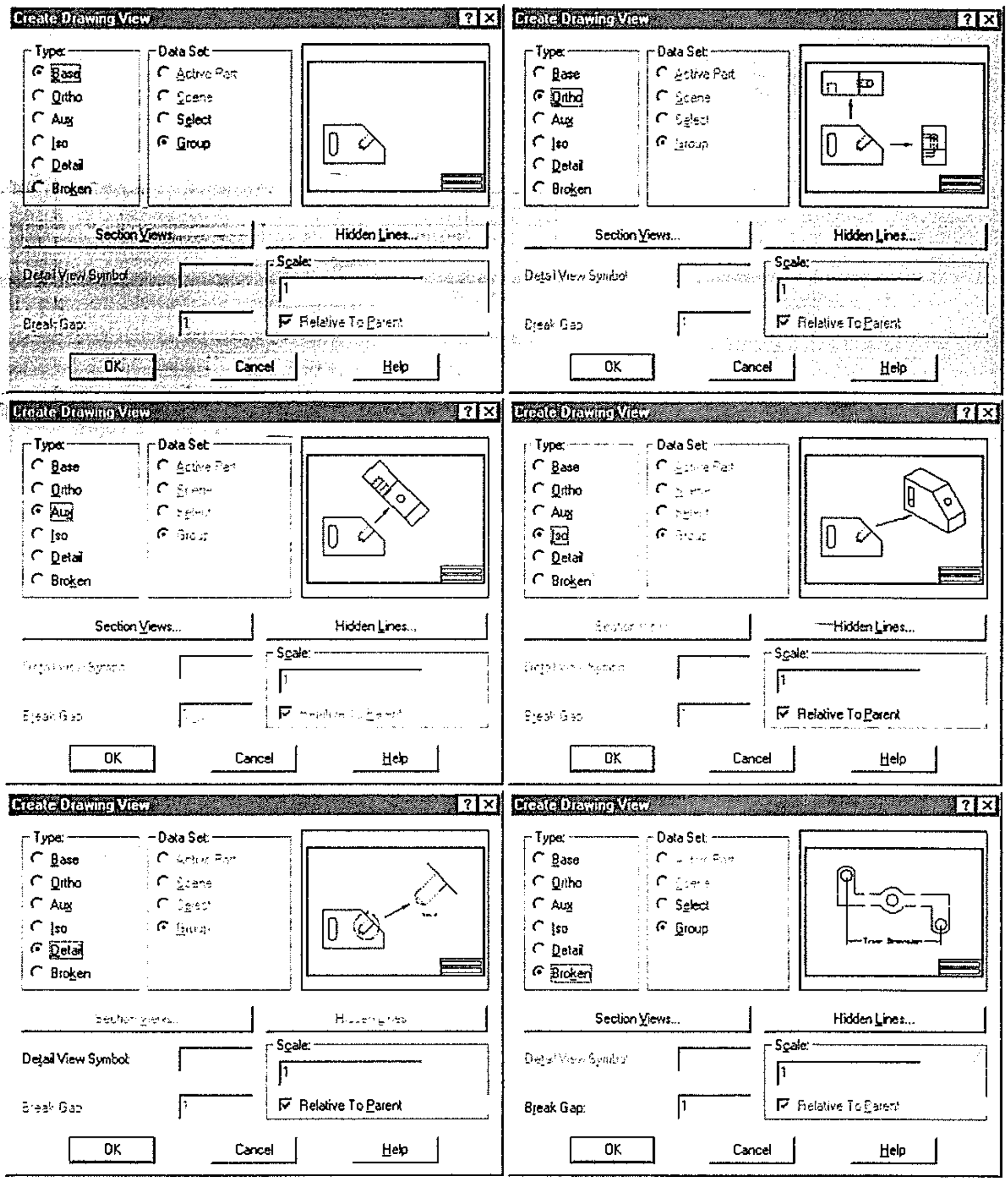


Figure 5.5 Six kinds of drawing views

Dimensioning

Dimensioning in an engineering drawing serves as a supplement to the graphics representations in the drawing views. By reason of the natures of the three kinds of objects (3D NURBS surfaces, 3D parametric solids, and 3D virtual assemblies), their dimensioning requirements are not the same. The solid parts are among the three kinds of 3D objects that we need to dimension fully. The prime function of assemblies is to illustrate and explain how a number of parts are assembled together. Dimensions, if required, are limited to key dimensions having to do with the relations of parts. For 3D NURBS surfaces, dimensions are not usually added to a drawing, because it is impractical to depict free-form surfaces by specifying their dimensions.

By default, the parametric dimensions that are used to construct 3D parametric solids will appear automatically in the drawing views in drawing mode. The display of these dimensions is in accordance with the way you put them in your sketch while constraining it. Because the format in which you place the parametric dimensions may not conform with appropriate drawing standards and practical needs, you have to adjust the display of dimensions by hiding inappropriate dimensions, moving dimensions from one drawing view to another, and adding relevant reference dimensions. To meet these requirements, there are two kinds of dimensions in drawing mode: parametric dimensions and reference dimensions.

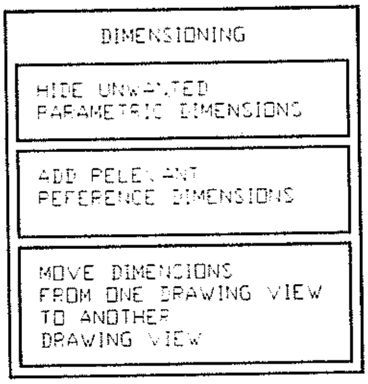


Figure 5.6 Dimensioning a solid part

Associativity

Parametric dimensions and reference dimensions placed in drawing mode are associative to the 3D objects constructed in model mode. If you change the sizes of the 3D objects, the dimensions update automatically because they are associated with the 3D objects.

Working in the other direction, you can change the parametric dimensions in drawing mode and the 3D solids in model mode will change as well. This is called bidirectional associativity. As for reference dimensions, they simply report the dimension size. They cannot drive the changes in 3D solids.

Annotations and Symbols

In addition to placing dimensions in the drawing views, you can add annotation text, welding symbols, surface texture symbols, and geometric tolerance symbols. (See Figures 5.7 through 5.9.)

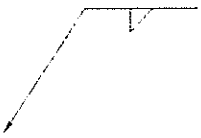


Figure 5.7 Welding symbol

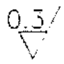


Figure 5.8 Surface texture symbol

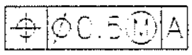


Figure 5.9 Geometric tolerance frame

Summary

There are two working modes, model mode and drawing mode. In model mode, you construct 3D NURBS surfaces, 3D parametric solids, and 3D virtual assemblies. In drawing mode, you construct 2D engineering drawings from the 3D objects that are placed in model mode. To construct an engineering drawing, you perform two major tasks: creating an engineering title block to comply with appropriate engineering drawing standards, and adding orthographic drawing views to depict the 3D objects.

With 3D solids, the parametric dimensions that are used to construct them appear automatically. You can hide unwanted parametric dimensions, add reference dimensions, and move dimensions from one drawing view to another. You can also add annotations, welding symbols, surface texture symbols, and geometric tolerance frames to the drawing views. The 2D engineering drawing in drawing mode is associated to the 3D objects in model mode. Changes in the 3D objects cause corresponding changes in the 2D drawings. In particular, the 3D parametric solids and their drawings are bidirectionally associative. Changing the parametric dimensions in the 2D drawing causes changes in the 3D solid.

5.2 Standard Practice and Drafting Preferences

The first time you switch to drawing mode, your screen becomes blank. By default, it displays an area equal to the size limit of the drawing file. Before you add drawing views, you have to define a working area by using the LIMITS command and insert an engineering title block by using the DDINSERT command.

The engineering title block serves two purposes: to comply with appropriate drawing standards and to define a working space within which you place floating viewports.

To produce a 2D engineering document, you need to follow a standard practice, such as ANSI, DIN, ISO, and so on. Whatever the standard is, you need to place the engineering content of the drawing within a proper title block.

Title Block

A title block is a rectangle, or four border lines, with margins around the edge of the drawing paper. Within the title block, in addition to the engineering drawing itself, you need to include general engineering information. Typically, it should have the name of the company, possibly with the company logo, and textual data. You need to state who constructs the drawing, who checks the drawing, who approves the drawing, the date, the plotting scale, the material, the surface finish requirement, the tolerances, and so on Figure 5.10 is a sample drawing of a Letter size (or A4) title block.

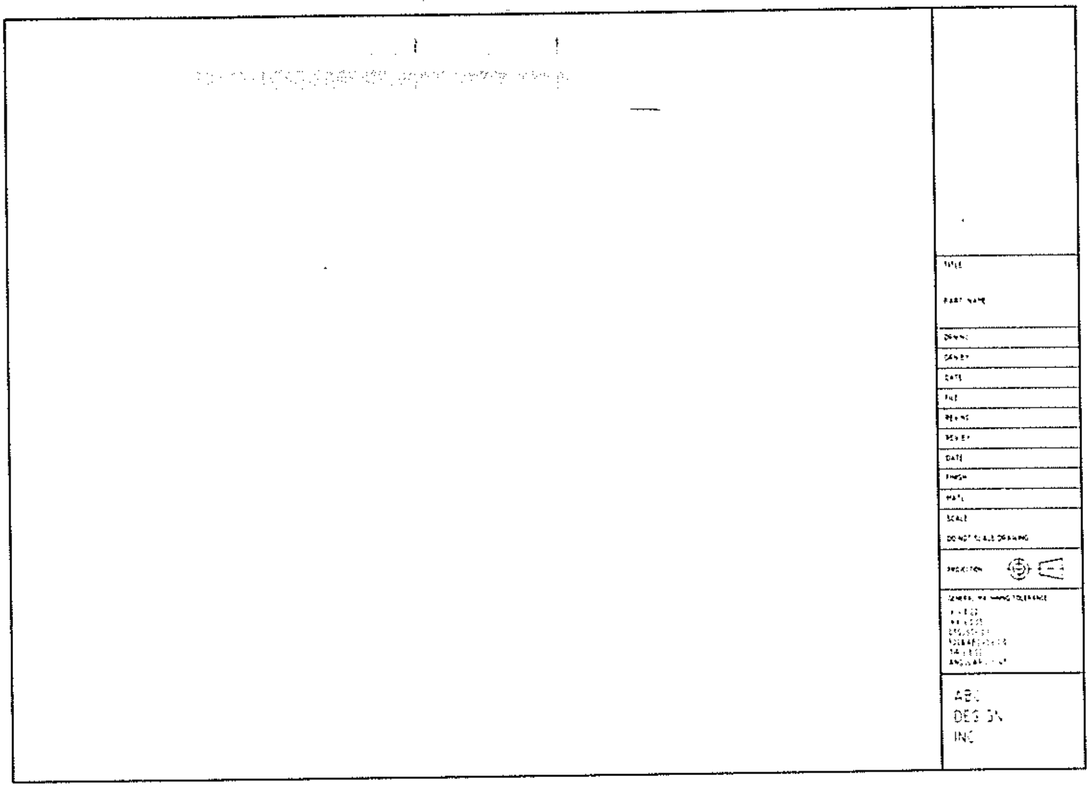


Figure 5.10 Letter size (or A4) engineering title block

You should have a similar title block ready before you start constructing engineering drawings in this chapter. If you do not have one, make one now. Then save it in the computer for later insertion into the engineering drawing.

In making the title block, you may ask what the widths of the margins between the four borders and the edges of the paper should be. The answer is that it depends on how much space your plotting device needs to clamp the paper while it prints. You can check it out by drawing a rectangle. Then plot the rectangle with your plotting device, using the "Fit to size" option. After plotting, measure the lengths of the printed lines. These lengths are the width and height of the rectangle that you should use to create a formal title block.

Drawing Preferences

To conform to standard engineering drawing practice, you have to set the preferences before starting to construct a drawing. Select the Preferences... item from the Drawing pull-down menu. (See Figure 5.11, the Desktop Preferences dialog box.) In the dialog box, there are eleven check boxes, two pull-down boxes, and two buttons:

<Drawing> **<Preferences...>**

Command: **AMPREFS**

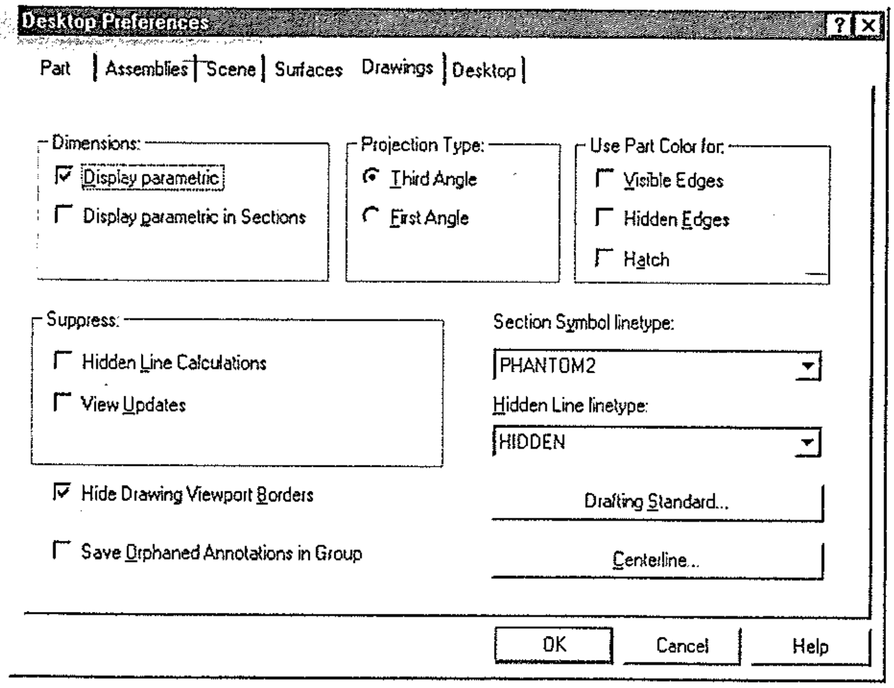


Figure 5.11 Drawings tab of Desktop Preferences dialog box

Check Boxes:

Dimensions: Display parametric	Checking this box displays the parametric dimensions of the solids in all the drawing views except the sectional views.
Display parametric in Sections	Checking this box displays the parametric dimension in the sectional views.
Projection Type: Third Angle First Angle	Checking one of these two boxes sets the drawing views to be created in third angle or first angle projection.
Use Part Color for: Visible Edges Hidden Edges Hatch	These three boxes concern the colors of the lines of the drawing views. Checking them causes them to use the color of the solid parts. By default, the visible edges, hidden edges, and hatches reside on the layers Am_vis, Am_hid, and Amv_2_hatch.

Suppress:

Hidden Line Calculations — Checking this box suppresses the display of hidden lines in the drawing views.

View Updates — Checking this box causes the drawing views to update automatically if objects in model mode are modified. If updating is suppressed, you can update the drawing views by using the AMUPDATEDWGVIEW command (the Update Drawing View item of the Drawing pull-down menu).

Hide Drawing Viewport Borders — Viewports have borders that reside on layer Am_views. Checking this box turns off this layer, thus hiding the border of the drawing views.

Save Orphaned Annotations in Group — Checking this box causes unupdated annotations to be saved in a group. Otherwise, they are deleted.

Pull-Down Boxes

Section Symbol linetype — This pull-down box enables you to set the linetype for the sectional view cutting plane.

Hidden Line linetype — This pull-down box enables you to set the linetype for the hidden lines in the drawing views.

Buttons

The [Drafting Standard...] button (Figure 5.11) brings up the Drafting Standards dialog box. (See Figure 5.12.) In the Drafting Standards dialog box, you can select one of the four standards: ANSI, DIN, ISO, or JIS.

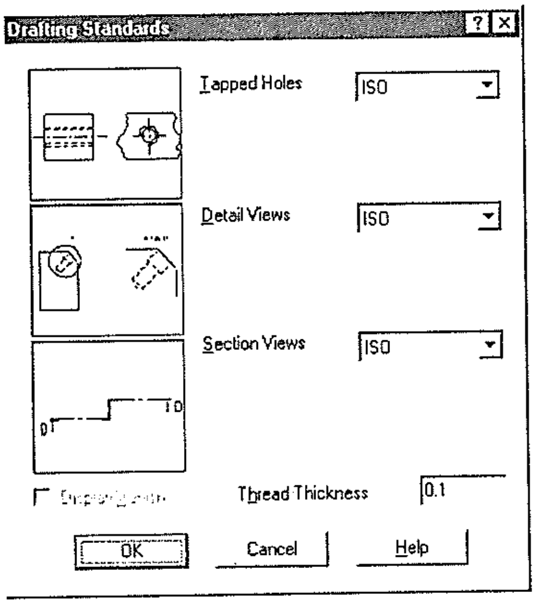


Figure 5.12 Drawing Standards dialog box

The [Centerline...] button (Figure 5.11) brings up the Centerlines dialog box. (See Figure 5.13.) Using the Centerlines dialog box, you can set the centerline linetype proportion and select a linetype for the center lines of the drawing views.

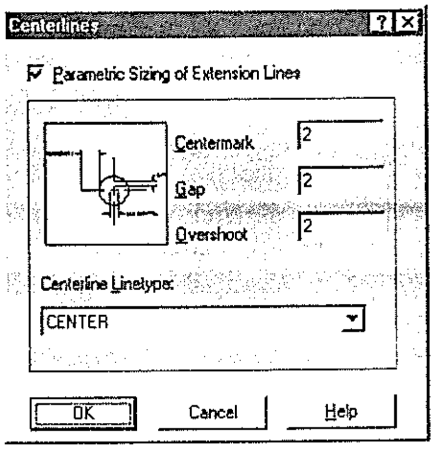


Figure 5.13 Centerlines dialog box

5.3 Associative Engineering Drawings

Now you will construct associative engineering drawings from the 3D objects you constructed in Chapters 2, 3, and 4.

Base View and Orthographic Views

Open the drawing Mblock.dwg that you constructed in Chapter 3. (See Figure 5.14.)

<File> **<Open...>**

File name: **Mblock.dwg**

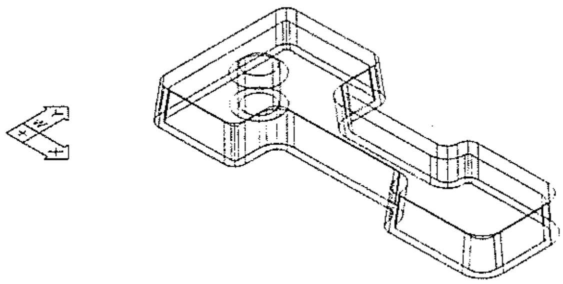


Figure 5.14 Mounting block constructed in model mode

To prepare an engineering drawing, you have to switch to drawing mode by selecting the Drawing Mode item of the Drawing pull-down menu or selecting the Drawing tab of the Desktop Browser. (See Figure 5.15.)

<Drawing> **<Drawing Mode>**

Command: **AMMODE**
Model/<Drawing>: **DRAWING**

After you switch to drawing mode, the solid part that you constructed in model mode disappears. The screen is blank because you have not done anything in this mode.

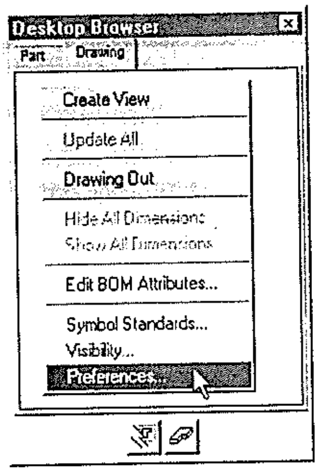


Figure 5.15 Drawing tab of the Desktop Browser

Note in Figure 5.15 that the Drawing tab is blank because there is no drawing view in drawing mode. Select the background of the browser (Figure 5.15). Then press the right mouse button to bring up a menu. Select the Drawing Preferences item. Set the preferences as shown in Figures 5.11 through 5.13.

<Drawing> **<Preferences...>**

Command: **AMPREFS**

As we have said, you should have constructed a drawing title block and saved it in your computer. Now use the LAYER command to construct a new layer called Title and set the current layer to Title. Then use the DDINSERT command to insert the title block into your drawing. (See Figure 5.16.)

<Assist> **<Format>** **<Layer...>**

Command: **LAYER**
New current layer: **Title**

<Construct> **<Insert Block...>**

Command: **DDINSERT**

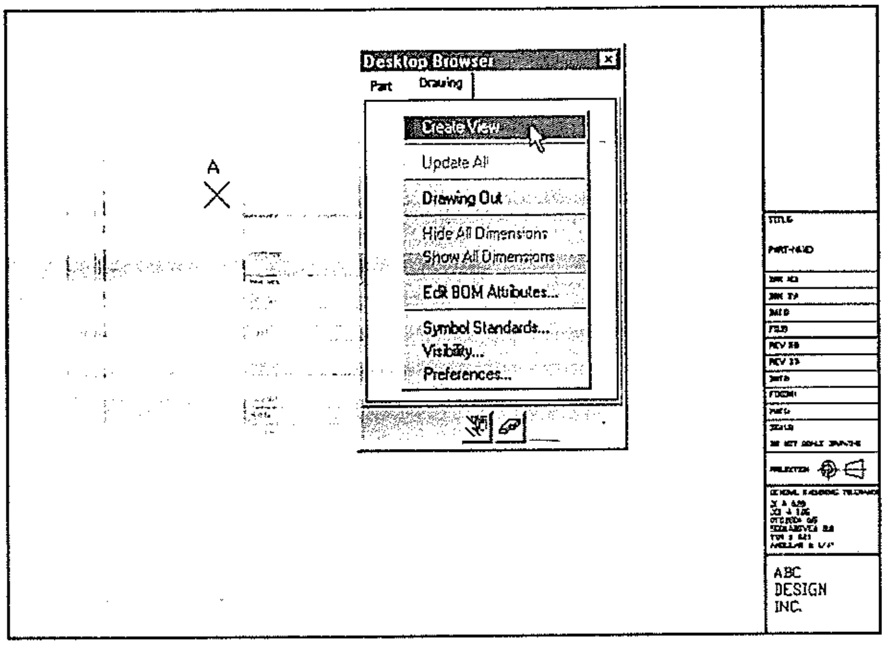


Figure 5.16 Title block inserted

After inserting the title block in layer Title, select the background of the browser (Figure 5.16), press the right mouse button, and then select the Create View item to use the AMDWGVIEW command (or select the New View... item of the Drawing pull-down menu) to construct a drawing view.

Because this is the first drawing view in drawing mode, there are only two options available, Base and Broken views. Choose the Base view. You will work on broken views later in this chapter.

Data Set refers to the kind of objects to be included in the drawing view. Choose Active Part to display the active solid part in the drawing view.

To construct a base view, you have to specify an orientation. Select X to use the worldxy plane as the viewing plane. Then select X to specify the direction of the view. After that, select a point on your screen to specify the location of the drawing view. This way, a drawing view is constructed. (See Figure 5.17.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Active Part**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **X**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**
Location for base view: **[Select A (Figure 5.16).]**
Location for base view: **[Enter]**

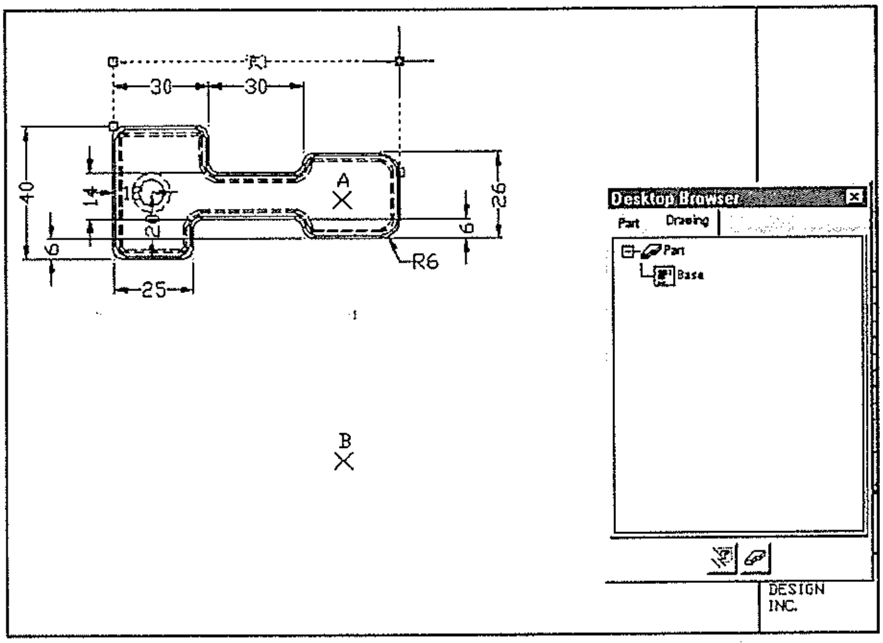


Figure 5.17 Base view constructed

There are two kinds of viewports: tiled viewports in model mode and floating viewports in drawing mode. The drawing view that you constructed by using the AMDWGVIEW command is a floating viewport. Unlike tiled viewports in model mode, drawing viewports are entities that you can manipulate. By default, the floating viewport that you construct in drawing mode is placed on the Am_views layer. It is turned off. Therefore, the viewport borders are hidden.

In the browser, you will find the item Base. It stands for the base drawing view that is constructed. In the graphics window, you will find that the parametric dimensions are displayed automatically in the drawing view. The way the dimensions are placed depends on the way you added them to the sketch while constraining the sketch. Dimensions in Figure 5.17 may not be the same as what is shown in your screen. To adjust the locations of the dimensions, you can select them to make the grip points visible (see Figure 5.17), select the grip points, and drag them to a new position.

Repeat the AMDWGVIEW command to construct an orthographic view. An orthographic view is generated from an existing view, the parent view. Select the base view as the parent view. Then select a point below it. (See Figure 5.18.)

<Drawing> <New View...>

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
OK]

Select parent view: [**Select A (Figure 5.17).**]
Location for orthographic view: [**Select B (Figure 5.17).**]
Location for orthographic view: [**Enter**]

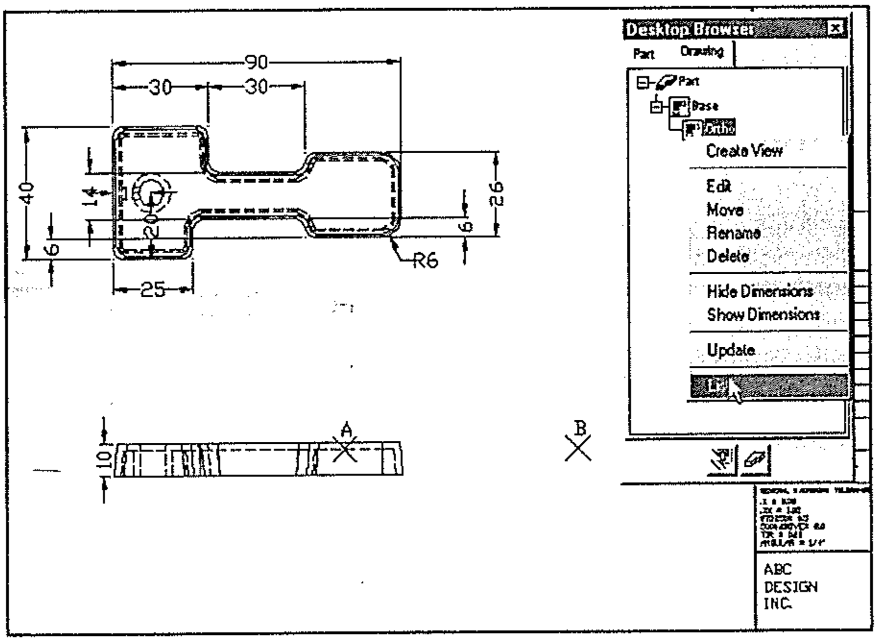


Figure 5.18 Orthographic view constructed

In the browser, select the Ortho item and then press the right mouse button to bring up a pop-up menu. After that, select the List item (Figure 5.18) to use the AMLISTDWG command to display information about the selected view. You can also select the List item of the Utilities cascading menu of the Drawing pull-down menu to use the AMLISTDWG command.

<Drawing> <Utilities> <List>

```
Command: AMLISTDWG
Orthogonal Drawing View
id = 4      view is ACTIVE and up to date
view scale: 1.0000
view direction: 0.0000,-1.0000,0.0000
center point: 78.7947,53.3917    target point: 251.0153,159.0850,5.0000
visible layer: AM_VIS
hidden layer: AM_HID     hidden layer linetype: HIDDEN
Hidden lines are displayed.
Tangent edges are displayed.
View has 0 descendants, 1 dimensions, 0 notes.
View is not aligned to any other view.
View has 0 views aligned to it.
One part represented
Third-angle projection
unfolded to the BOTTOM a distance of 81.4550 from view 2
```

Now select the Create View item (Figure 5.19) to use the AMDWGVIEW command to construct another orthographic view. (See Figure 5.19.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
OK]

Select parent view: **[Select A (Figure 5.18).]**
Location for orthographic view: **[Select B (Figure 5.18).]**
Location for orthographic view: **[Enter]**

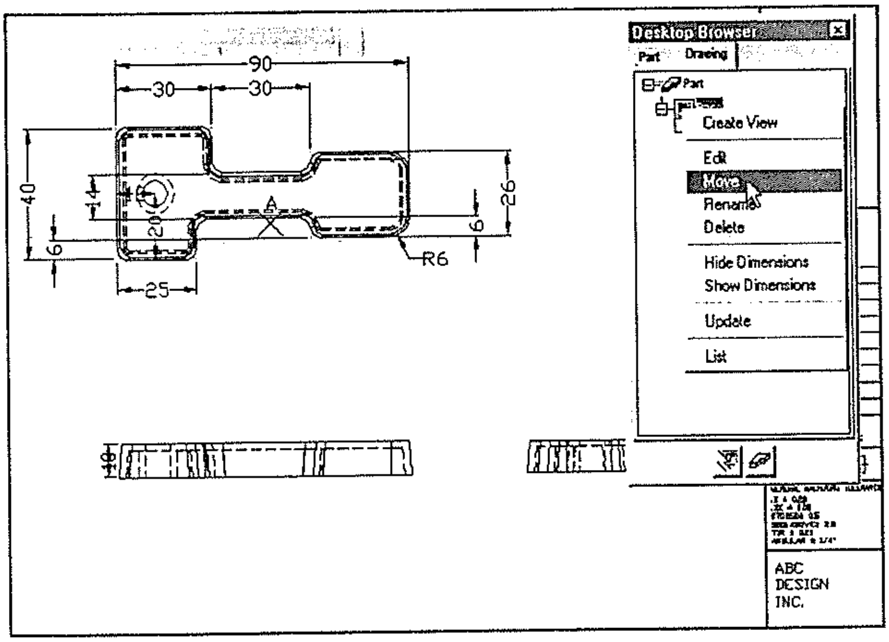


Figure 5.19 Third drawing view constructed

Dimensions can be hidden and unhidden. Select the Base view in the browser and press the right mouse button. Then select Hide Dimensions (Figure 5.19) to hide the dimensions. After that, select Show Dimensions (Figure 5.19) to display the dimensions again.

Now select Move (Figure 5.19) to use the AMMOVEVIEW to move the drawing views to a new position (or select the Move item of the Edit View cascading menu of the Drawing pull-down menu). The two orthographic views that are derived from this base view are called descendant views. Note that they move as well. (See Figure 5.20.)

<Drawing> **<Edit View>** **<Move>**

Command: **AMMOVEVIEW**
2 descendant views will also be moved.
View location: **[Select A (Figure 5.19).]**
View location: **[Enter]**

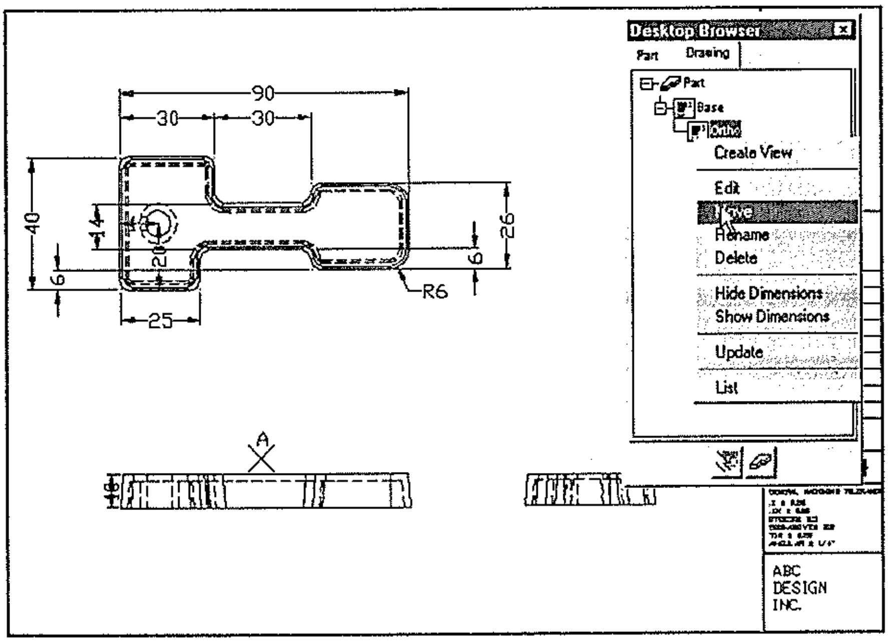


Figure 5.20 Base view and its descendant views moved

Select the first Ortho view in the browser and press the right mouse button. Then select Move (Figure 5.20) to move this view. Note that the base view, which is the parent view of the selected view, does not move. Only the descendant view (the side view) moves with it. (See Figure 5.21.)

Command: **AMMOVEVIEW**
1 descendant view will also be moved.
View location: [**Select A (Figure 5.20).**]
View location: [**Enter**]

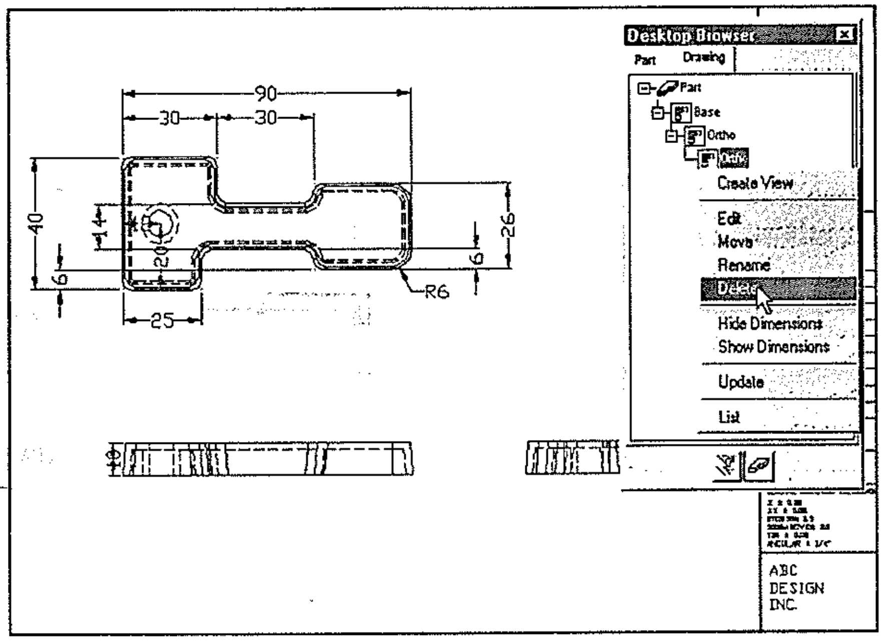


Figure 5.21 Front view and its descendant view moved

Now select the second Ortho view in the browser and press the right mouse button. Then select the Delete item (Figure 5.21) to use the AMDELVIEW command to delete the view (or select the Delete item of the Edit View cascading menu of the Drawing pull-down menu). This way, the selected view is deleted. (See Figure 5.22.)

<Drawing> **<Edit View>** **<Delete>**

Command: **AMDELVIEW**

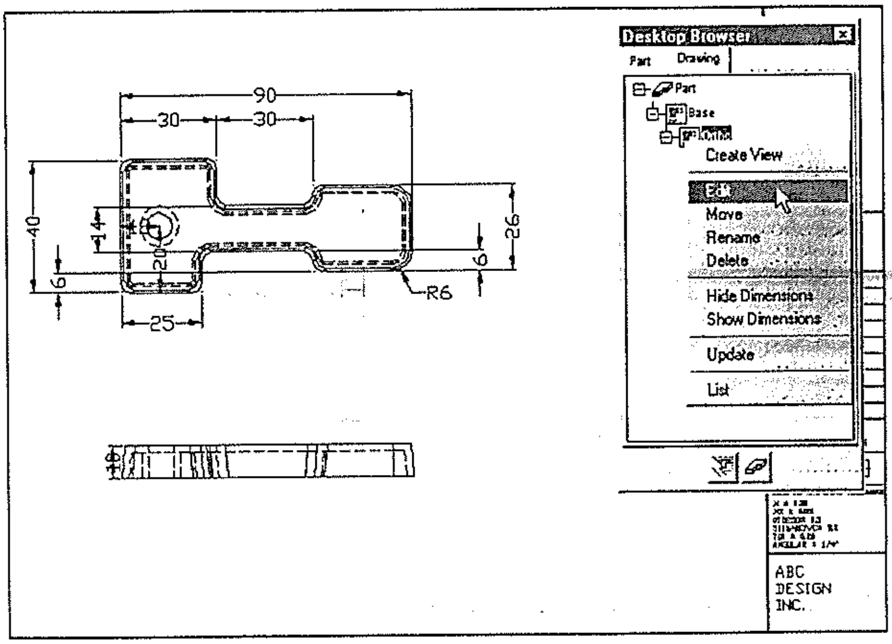


Figure 5.22 Side view deleted

Now select the Edit item (Figure 5.22) to use the AMEDITVIEW command to edit the base view (or select the Attributes... item of the Edit View cascading menu of the Drawing pull-down menu). (See Figure 5.23, the Edit Drawing View dialog box.)

<Drawing> <Edit View> <Attribute...>

Command: **AMEDITVIEW**

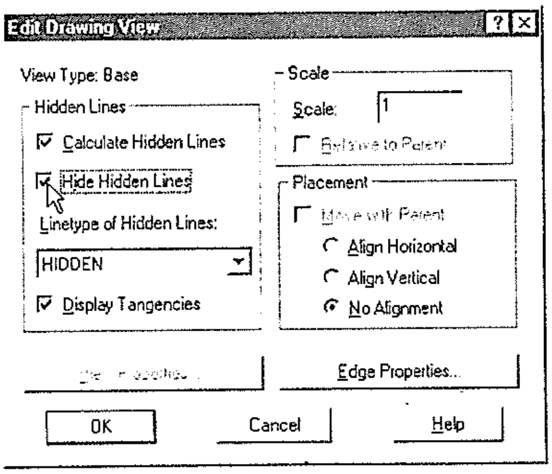


Figure 5.23 Edit Drawing View dialog box

The Edit Drawing View dialog box enables you to edit various attributes of a drawing view. Unchecking the Calculate Hidden Lines box displays the 3D model in wireframe mode. Checking the Calculate Hidden Lines box and unchecking the Hide Hidden Lines box displays only the outlines of the 3D model. Tangencies are lines at the adjoining edges of changing curvatures such as a fillet surface meeting a flat surface. Unchecking the Display Tangencies box suppresses the display of such tangency lines. The Scale box determines the display zoom scale of the drawing view in relation to the actual size of the model. The Placement area determines how the drawing view is aligned. Because this is the base view, the [View Properties...] button is disabled.

Selecting the [Edge Properties...] button and then selecting edges of the drawing view brings up the Edge Properties dialog box. (See Figure 5.24.)

Unhide all/<Select>: **[Enter]**
Select Edges: **[Select edges that you want to manipulate.]**

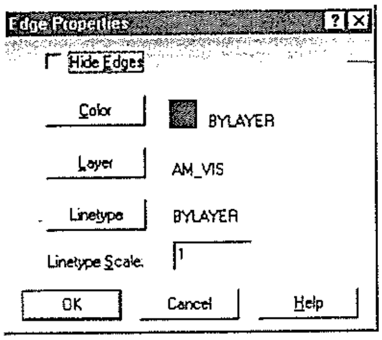


Figure 5.24 Edge Properties dialog box

In the Edge Properties dialog box, you can change the color, layer, linetype, and linetype scale of a selected line. You can hide the selected edge by checking the Hide Edges box. Now select the [OK] button. On returning to the Edit Drawing View dialog box, check the Hide Hidden Lines box and then select the [OK] button. (See Figure 5.25.) The hidden lines of the base view are hidden.

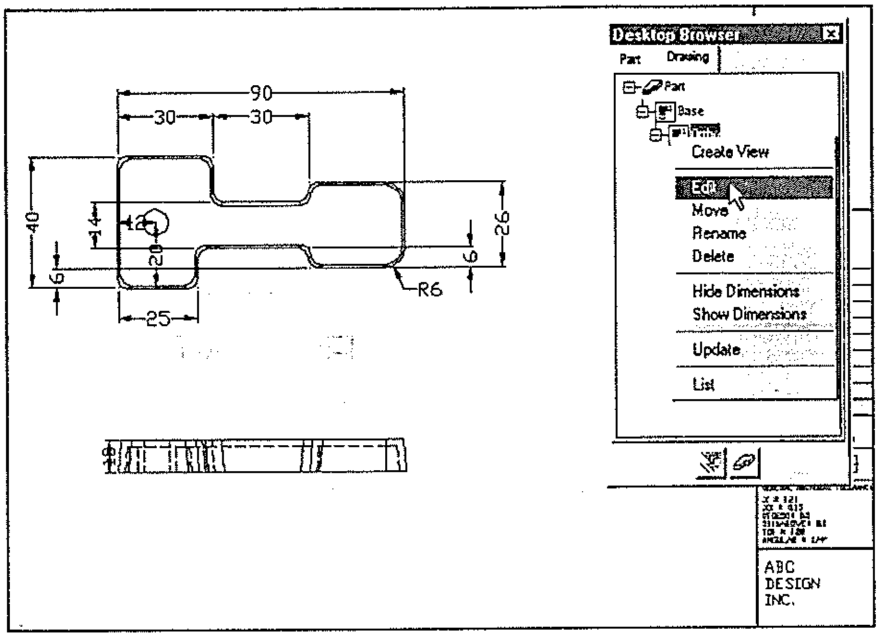


Figure 5.25 Hidden lines hidden in the base view

Apply the AMEDITVIEW command to the top view and the front view. Then uncheck the Display Tangencies box to suppress the tangency lines in the two drawing views. (See Figure 5.26.)

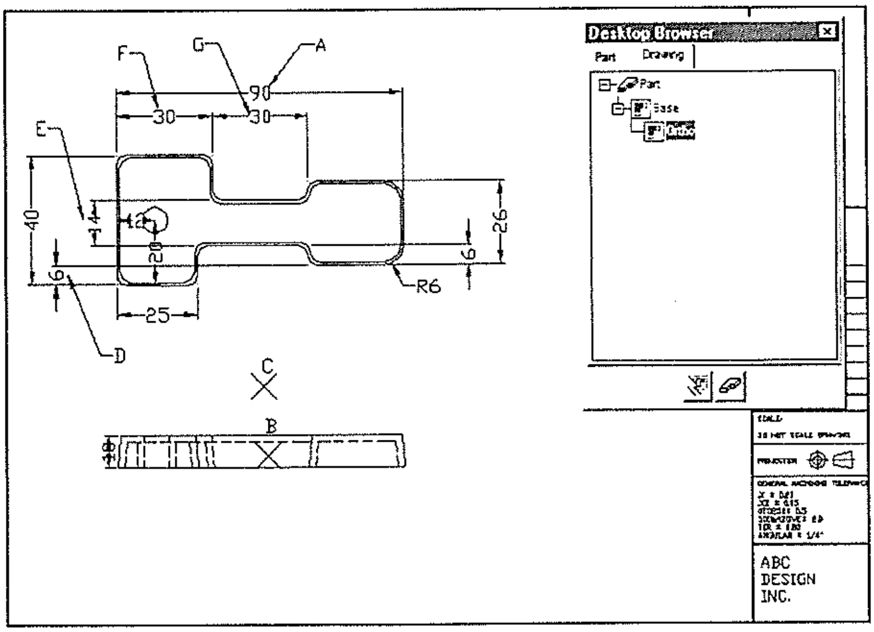


Figure 5.26 Tangency lines suppressed in the drawing views

As we have said, the display of parametric dimensions in drawing mode depends on the way you place them when constraining the sketch. Therefore, a dimension that is used to constrain the top view profile is placed on the top view. To move it to some other drawing view, you can use the AMMOVEDIM command. To align two dimensions, you can use the AMDIMALIGN command. To combine two dimensions to form a single dimension, you can use the AMDIMJOIN command. (See Figure 5.27.)

<Drawing> <Move Dimension>

Command: **AMMOVEDIM**
Flip/Reattach/<Move>: **M**
Select dimension: **[Select A (Figure 5.26).]**
Select view to place dimension: **[Select B (Figure 5.26).]**
Location for dimension: **[Select C (Figure 5.26).]**
Location for dimension: **[Enter]**
Select dimension: **[Enter]**

<Drawing> <Edit Dimensions> <Align>

Command: **AMDIMALIGN**
Select base dimension: **[Select D (Figure 5.26).]**
Select linear dimensions to align: **[Select E (Figure 5.26).]**
Select linear dimensions to align: **[Enter]**

<Drawing> <Edit Dimensions> <Join>

Command: **AMDIMJOIN**
Select base dimension: **[Select F (Figure 5.26).]**
Select linear dimensions to join: **[Select G (Figure 5.26).]**
Select linear dimensions to join: **[Enter]**

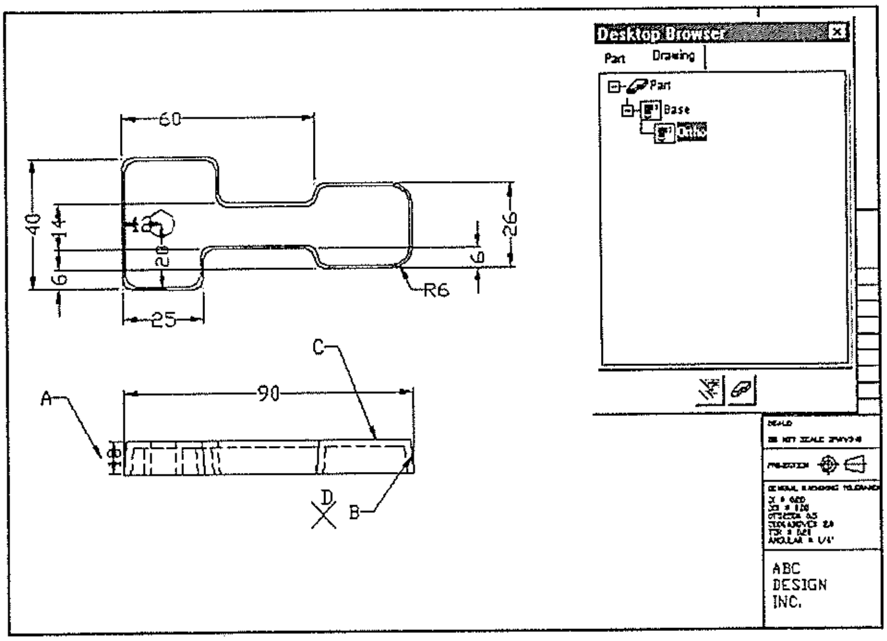


Figure 5.27 Dimension moved, dimensions aligned, and dimensions joined

–Now edit the format of a dimension by using the AMDIMFORMAT command. (See Figure 5.28, the Dimension Formatter – Linear dialog box.) Select the Format tab. Then select the [Text Above] button, check the Default Text Position box, and select the

[Apply] button. After that, select the [OK] button. The format of the selected dimension is changed.

<Drawing> **<Edit Format...>**

Command: **AMDIMFORMAT**
Select a dimension entity: **[Select A (Figure 5.27).]**

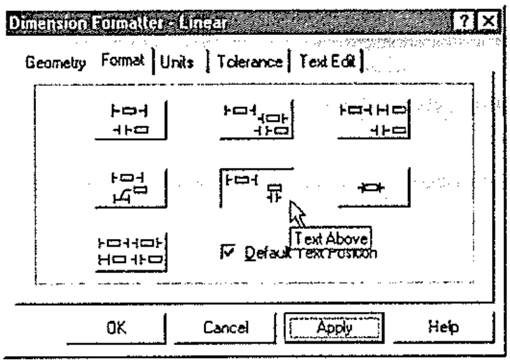


Figure 5.28 Dimension Formatter – Linear dialog box

Because not all the dimensions required in an engineering drawing are displayed automatically, you may need to add some dimensions. Now use the AMREFDIM command to add a reference dimension. (See Figure 5.29.)

<Drawing> **<Add Ref Dimension>**

Command: **AMREFDIM**
Select first object: [**Select B (Figure 5.27).**]
Select second object or place dimension: [**Select C (Figure 5.27).**]
Specify dimension placement: [**Select D (Figure 5.27).**]

Depending on where you select the objects and the placement position, the dimension given can be a horizontal dimension or a vertical dimension. To set the dimension to an angular dimension, type N.

Undo/Hor/Ver/Align/Par/aNgle/Ord/reF/Basic/pLacement point: **N**
Undo/reF/Basic/pLacement point: [**Enter**]
Select first object: [**Enter**]

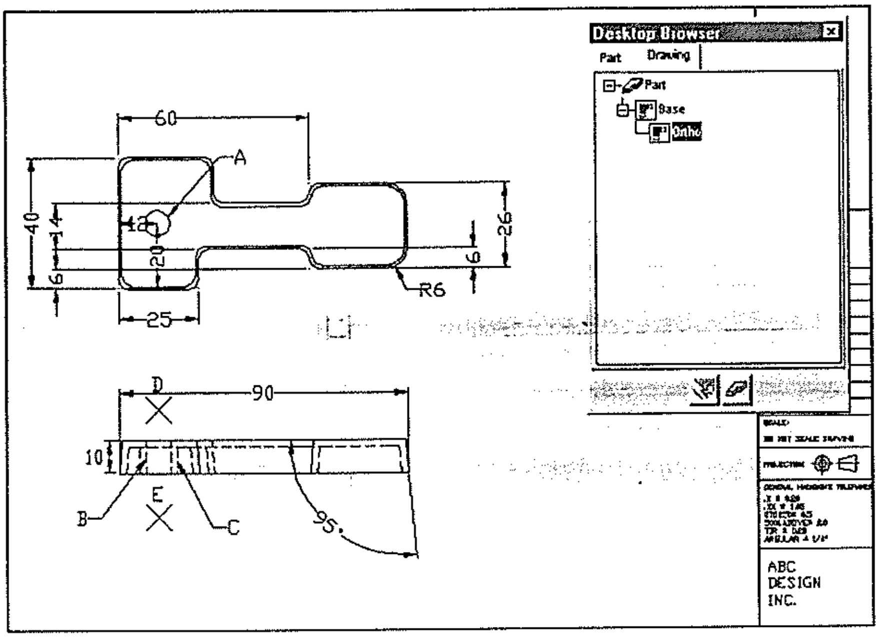


Figure 5.29 Dimension format changed, reference dimension constructed

Now set the center line linetype scale by using the AMPREFS command. Then use the AMCENLINE command to construct a pair of center lines in the top view and a center line in the front view. (See Figure 5.30.)

<Drawing> **<Preferences...>**

Command: **AMPREFS**

[Drawings
Centerline...]
[Centerlines
Centermark **2**
Gap **2**
Overshoot **2**
Centerline Linetype: **CENTER**
OK]
[**OK**]

<Drawing> **<Annotate>** **<Centerline>**

Command: **AMCENLINE**
Select Edge: [**Select A (Figure 5.29).**]
Select mirrored edge or <RETURN>: [**Enter**]

<Drawing> **<Annotate>** **<Centerline>**

Command: **AMCENLINE**
Select Edge: [**Select B (Figure 5.29).**]
Select mirrored edge or <RETURN>: [**Select C (Figure 5.29).**]
Select first trim point: [**Select D (Figure 5.29).**]
Select second trim point: [**Select E (Figure 5.29).**]

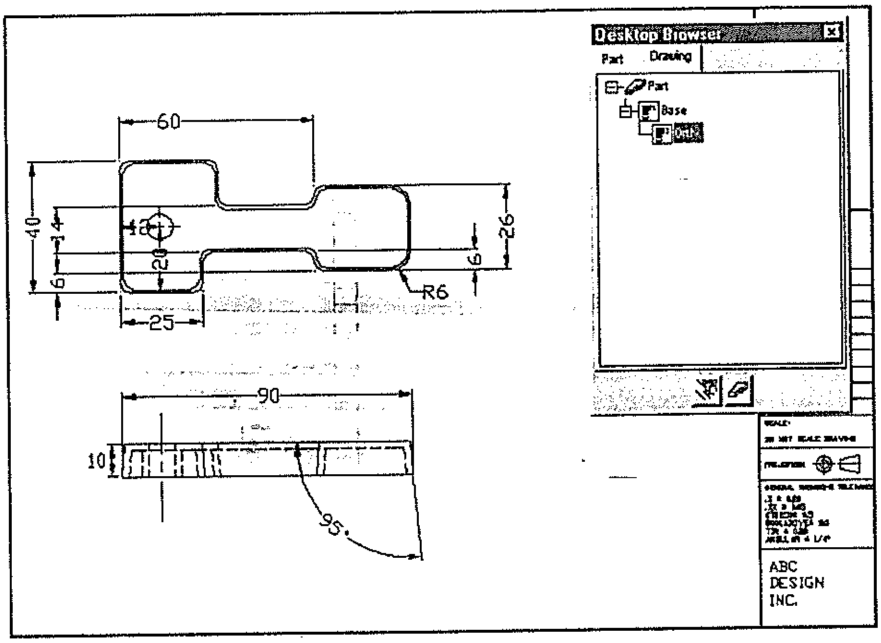


Figure 5.30 Center lines constructed

Now construct the dimensions as shown in Figure 5.31.

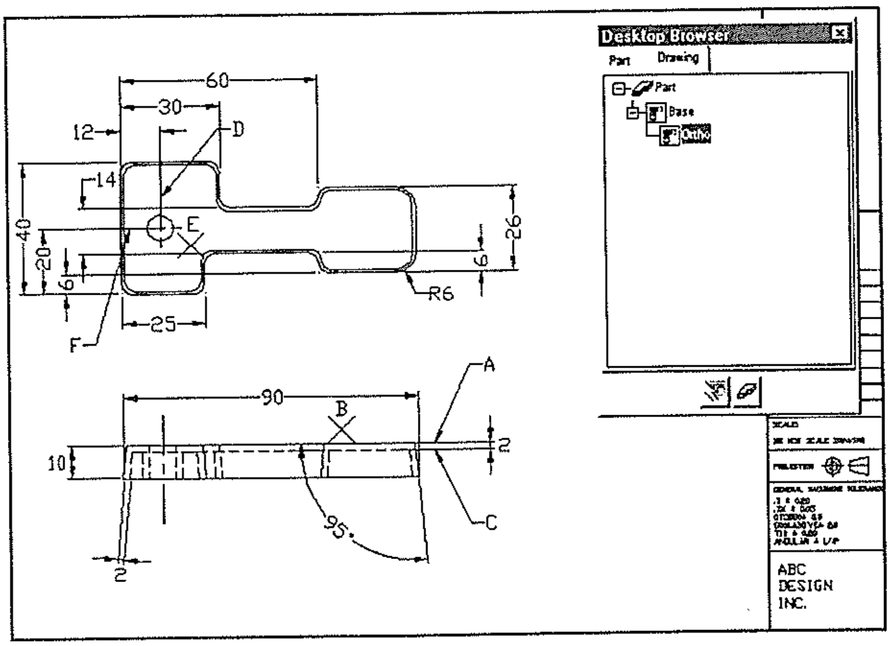


Figure 5.31 Dimensions added and edited

It is normal engineering practice to leave a small gap between the dimension extension line and the drawing view. Note that in Figure 5.31 the gap does not exist at A, B, C, and D. Use the AMDIMBREAK command to open up the gaps. (See Figure 5.32.)

<Drawing> <Edit Dimensions> <Break>

Command: **AMDIMBREAK**
Select dimension or extension line to break: **[Select A (Figure 5.31).]**
First point/Restore/<Second point>: **[Select B (Figure 5.31).]**

Command: [**Enter**]
AMDIMBREAK
Select dimension or extension line to break: **[Select C (Figure 5.31).]**
First point/Restore/<Second point>: **[Select B (Figure 5.31).]**

Command: [**Enter**]
AMDIMBREAK
Select dimension or extension line to break: **[Select D (Figure 5.31).]**
First point/Restore/<Second point>: **[Select E (Figure 5.31).]**

Command: [**Enter**]
AMDIMBREAK
Select dimension or extension line to break: **[Select F (Figure 5.31).]**
First point/Restore/<Second point>: **[Select E (Figure 5.31).]**

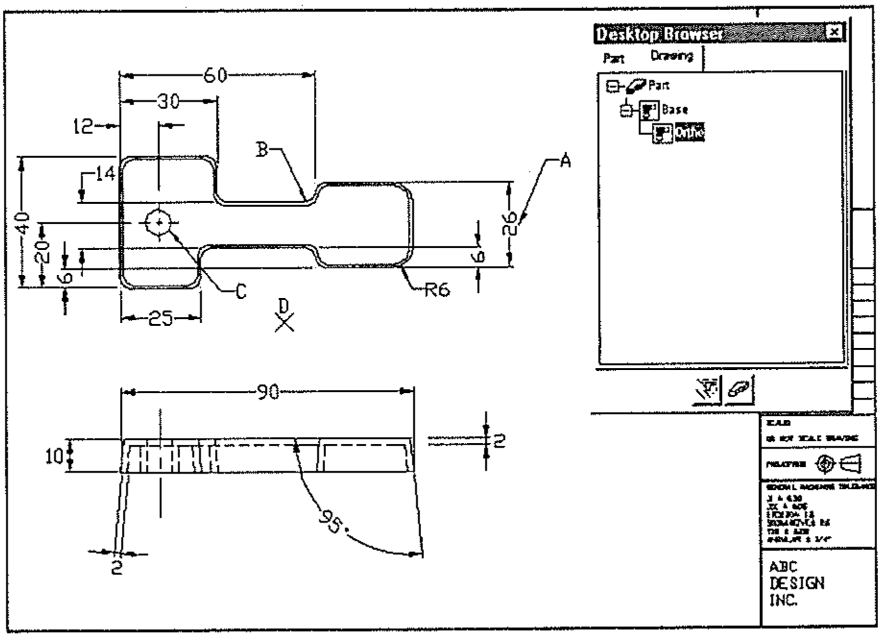


Figure 5.32 Dimension extension lines broken

Now you will insert a dimension and add a hole note. (See Figure 5.33.)

<Drawing> <Edit Dimensions> <Insert>

Command: **AMDIMINSERT**
Select base dimension: **[Select A (Figure 5.32).]**
Select point on object for extension line origin: **END** of **[Select B (Figure 5.32).]**

<Drawing> <Annotate> <Hole Note...>

Command: **AMHOLENOTE**
Edit/<New>: **[Enter]**
Select hole feature: **[Select C (Figure 5.32).]**

After you select a hole feature, the Create Holenote dialog box is displayed. (See Figure 5.33.)

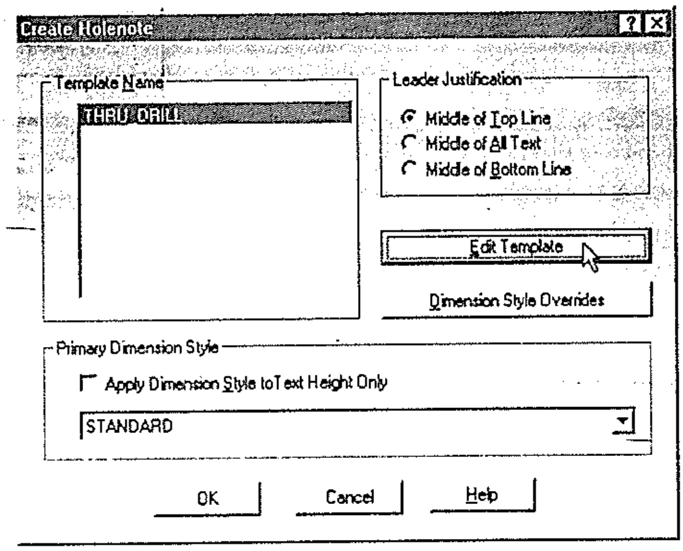


Figure 5.33 Create Holenote dialog box

To edit the hole notes, select the [Edit Template] button. The Multiline Text Editor dialog box appears. (See Figure 5.34.)

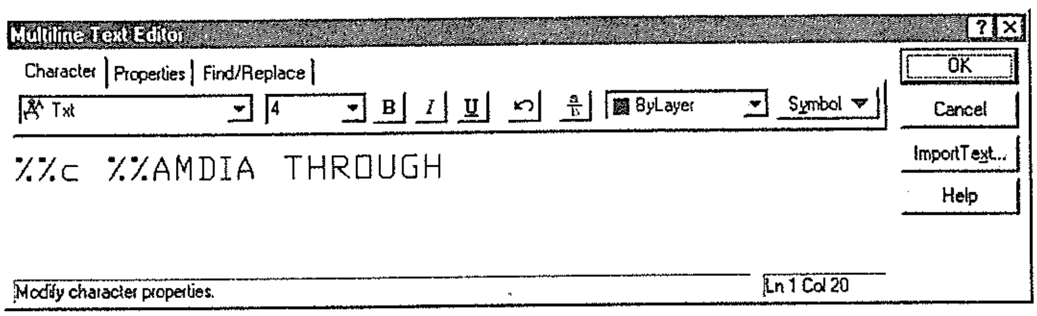


Figure 5.34 Multiline Text Editor dialog box

Edit the text appropriately. Then select the [OK] button.

Location for hole note: **[Select D (Figure 5.32).]**
Location for hole note: **[Enter]**
Select hole feature: **[Enter]**

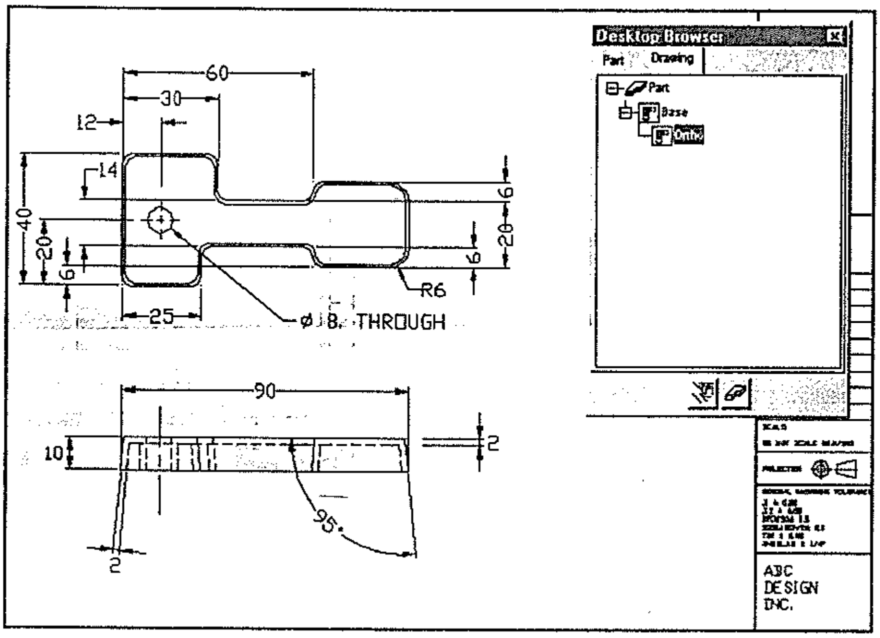


Figure 5.35 Dimension inserted, hole note added

The drawing document is now complete. Save your drawing.

<File> **<Save>**

Section View and Offset Section View

Open the drawing Arm_u.dwg (see Figure 5.36) that you saved in Chapter 3. You will construct an associative engineering drawing that consists of a front view, an offset section top view, and a section side view. Constructing an offset section requires a parametric cutting plane. In Chapter 3, you learned that a rough sketch can be resolved to form a profile, a path, a split line, or a cut line, and you learned how to use profiles, paths, and split lines. Here you will construct a rough sketch, resolve it to form a cut line, and use this line as a cut plane to construct an offset section.

<File> **<Open...>**

File name: **Arm_u.dwg**

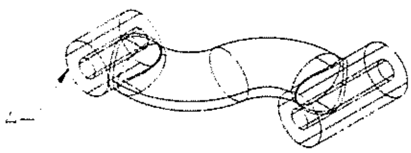


Figure 5.36 Upper swing arm

Use the AMSKPLN command to set the sketch plane to face A (Figure 5.36) as shown in Figure 5.37.

<Part> **<Sketch>** **<Sketch Plane>**

Command: **AMSKPLN**

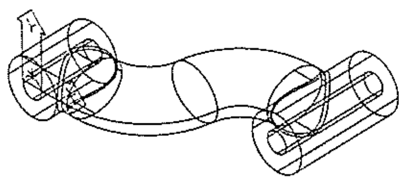


Figure 5.37 Sketch plane set

Use the shortcut key [9] to set the display to the plan view of the current UCS. Then construct a polyline as shown in Figure 5.38.

Command: **9**

<Design> **<Polyline>**

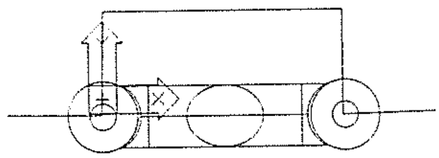


Figure 5.38 Display set, polyline constructed

Use the AMCUTLINE command to resolve the polyline to a cut line. Then add parametric dimensions to the cut line as shown in Figure 5.39.

<Part> **<Sketch>** **<Cutting Line>**

Command: **AMCUTLINE**
Select objects for section cutting line:
Select objects: **LAST**
Select objects: **[Enter]**

<Part> **<Add Dimension>**

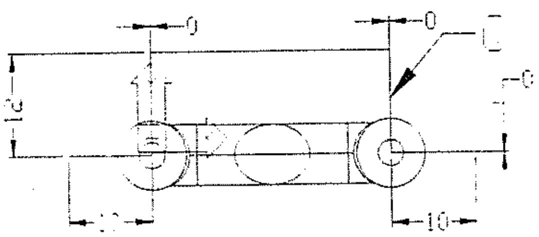


Figure 5.39 Cut line resolved and constrained

The cut line is complete. Use the AMMODE command to change to drawing mode. Then make a new layer called Title and insert a drawing title block. (See Figure 5.40.)

<Drawing> **<Drawing Mode>**

<Assist> **<Format>** **<Layer...>**

New current layer: **Title**

<Construct> **<Insert Block...>**

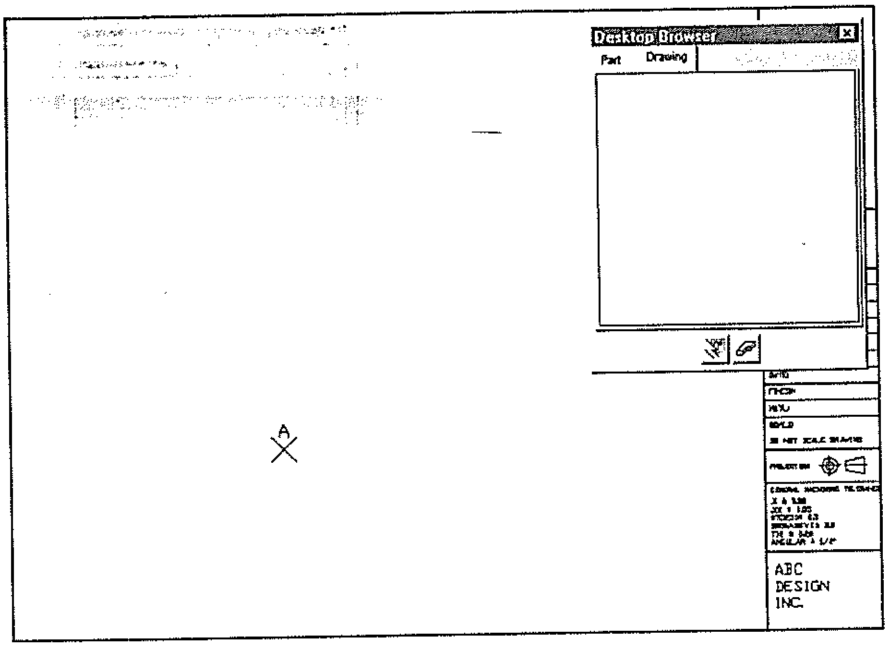


Figure 5.40 Drawing title block inserted in drawing mode

Use the AMPREFS command to set drawing preferences as shown in Figures 5.11 through 5.13. Then construct a front view. (See Figure 5.41.)

<Drawing> **<Preferences...>**

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Active Part**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **Z**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**
Location for base view: **[Select A (Figure 5.40).]**
Location for base view: **[Enter]**

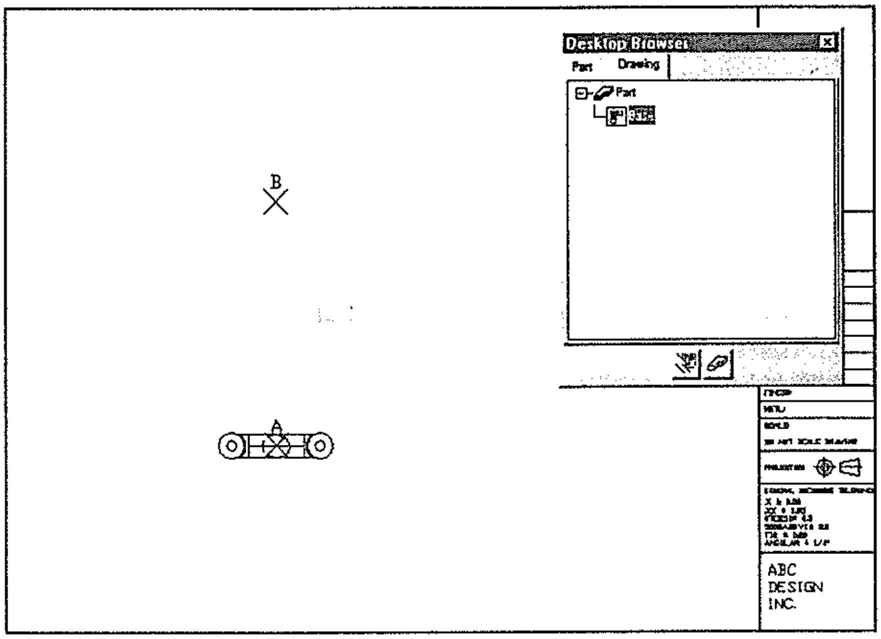


Figure 5.41 Front view constructed

Repeat the AMDWGVIEW command to construct an offset section top view. Select Ortho, because a section view is a kind of orthographic projection. To set a section view, select the [Section Views...] button. In the Section View dialog box, select Offset to construct an offset section. Then select the [Pattern...] button.

Select ANSI31 as the hatch pattern. Then set the hatch pattern scale to 1 and the hatch pattern rotation angle to 0°. After that, select the [OK] button.

On returning to the Section View dialog box, set the section symbol to A. Then select the [OK] button to exit.

After a while, the model appears in model mode. Select the cut line. (See Figure 5.42.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
Section Views...]

[Section View
Section Type **Offset**
Hatching **Hatch**
Pattern...]

[Hatch Pattern
Pattern: **ANSI31**
Scale **1**
Angle **0**
OK]

[Section View
Section Symbol: **A**
OK]

[**OK**]

Select parent view: [**Select A (Figure 5.41).**]
Location for orthographic view: [**Select B (Figure 5.41).**]
Location for orthographic view: [**Enter**]
Select cutting line sketch: [**Select C (Figure 5.39).**]

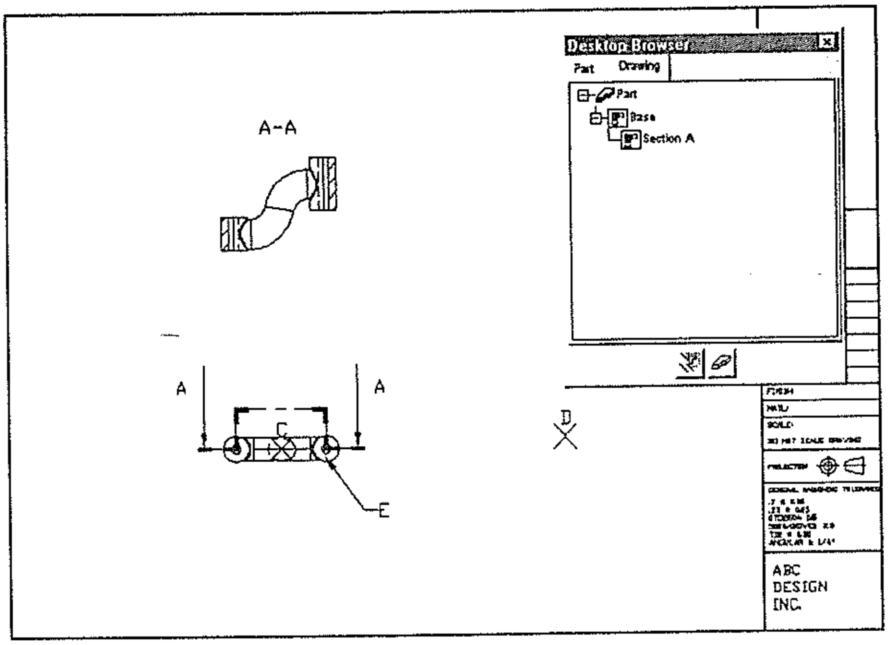


Figure 5.42 Offset section view constructed

Repeat the AMDWGVIEW command to construct a sectional side view. As in constructing an offset section, you have to specify a cutting plane. Select the work plane as the cutting plane. (See Figure 5.43.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
Section Views...]

[Section View
Section Type **Full**
Hatching **Hatch**
Section Symbol: **B**
OK]

[**OK**]

Select parent view: **[Select C (Figure 5.42).]**
Location for orthographic view: **[Select D (Figure 5.42).]**
Location for orthographic view: **[Enter]**
Section through Point/Ucs/<Work plane>: **[Enter]**
Select work plane in parent view for the section: **[Select E (Figure 5.42).]**

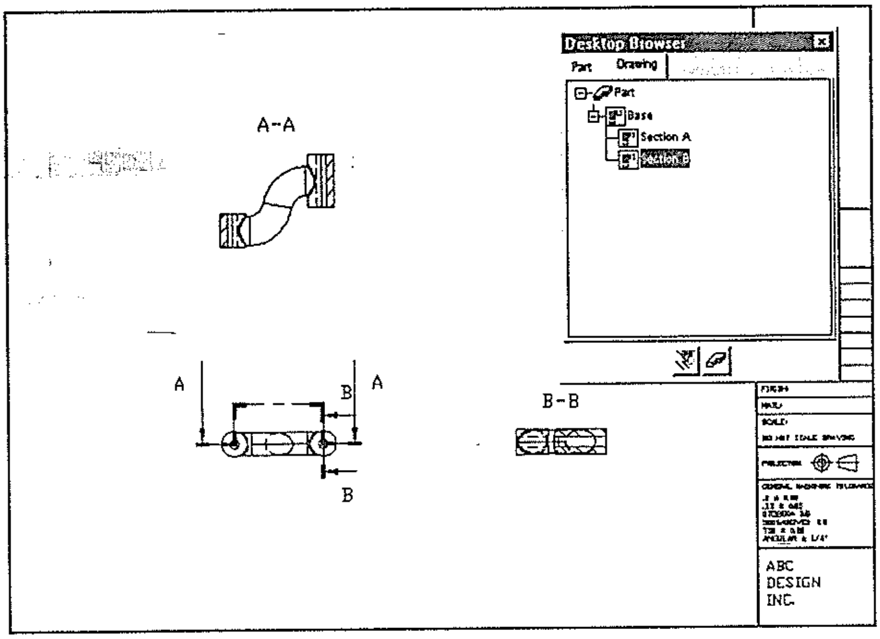


Figure 5.43 Sectional side view constructed

The drawing views are complete. Add appropriate center lines and dimensions. Then save your drawing.

<File> **<Save>**

Half Section View

Open the drawing Shockt.dwg (see Figure 5.44) that you saved in Chapter 3. You will construct an associative engineering drawing that consists of a top view and a half section front view. To construct a half section view, you need two work planes to define the location of the half section.

<File> **<Open...>**

File name: **Shock.dwg**

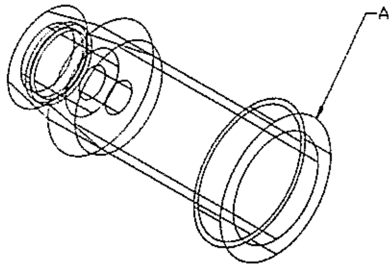


Figure 5.44 Shock absorber tube

Use the AMWORKAXIS command to construct a work axis. Then use the AMWORKPLN command to construct a work plane. (See Figure 5.45.)

<Part> **<Work Features>** **<Work Axis>**

Command: **AMWORKAXIS**
Select cylinder/cone/torus: [**Select A (Figure 5.44.**]

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[Work Plane Feature
1st Modifier **World XY**]

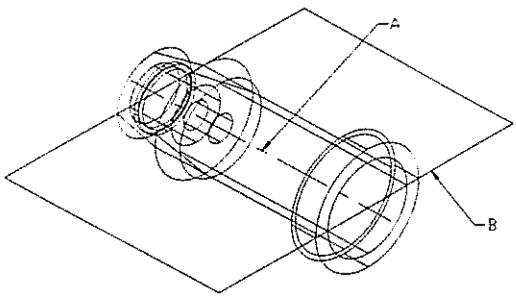


Figure 5.45 Work axis and work plane constructed

Now construct another work plane. (See Figure 5.46.)

<Part> **<Work Features>** **<Work Plane...>**

Command: **AMWORKPLN**

[1st Modifier **On Edge/Axis**
2nd Modifier **Planar Normal**
OK]

worldX/worldY/worldZ/<Select work axis or straight edge>: **[Select A (Figure 5.45).]**
worldXy/worldYz/worldZx/Ucs/<Select work plane or planar face>: **[Select B (Figure 5.45).]**

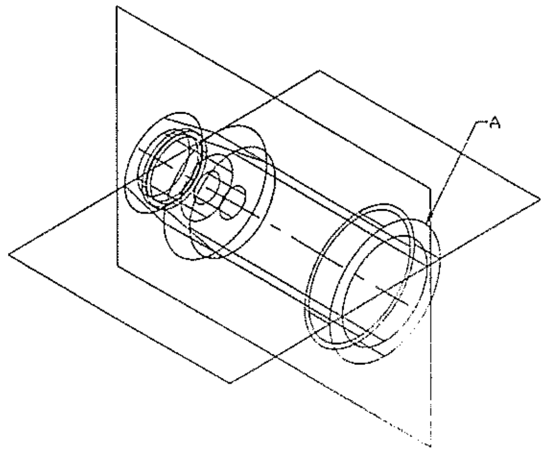


Figure 5.46 Second work plane constructed

Use the AMMODE command to enter drawing mode. Then use the AMPREFS command to set drawing preferences. After that, add a layer called Title and insert the drawing Title.dwg. (See Figure 5.47.)

<Drawing> **<Drawing Mode>**

<Assist> **<Format>** **<Layer...>**

New current layer: **Title**

<Construct> **<Insert Block...>**

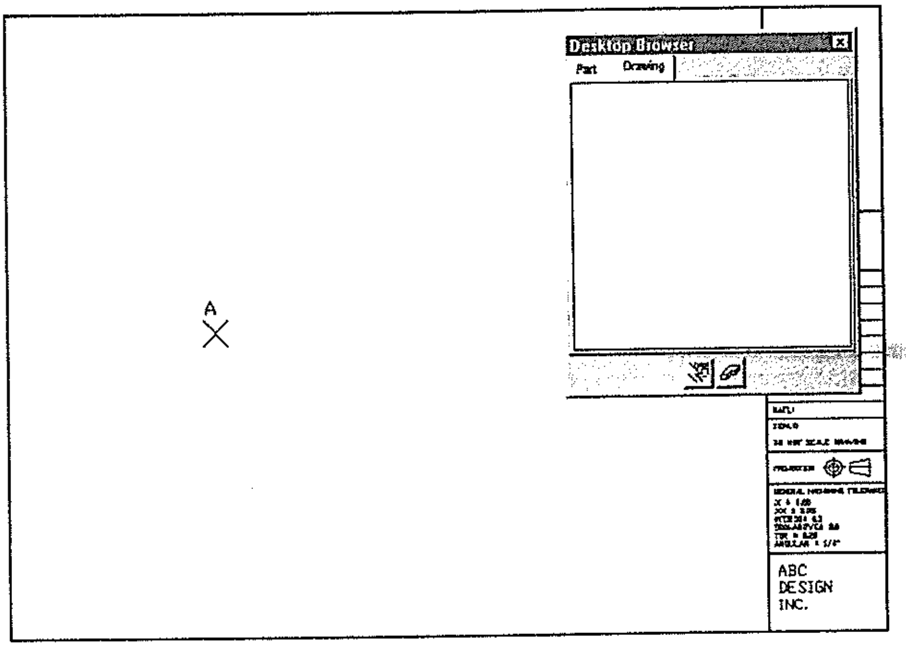


Figure 5.47 Drawing mode, title block inserted

Use the AMDWGVIEW command to construct a base view. (See Figure 5.48.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Active Part**
Scale: **2**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: [**Select A, the vertical face of the solid (Figure 5.46).**]
Next/<Accept>: [**Accept if the vertical face A (Figure 5.46) is highlighted.**]
worldX/worldY/worldZ/<Select work axis or straight edge>: **Y**
Rotate/Z-flip/<Accept>: [**Enter**]
Location for base view: [**Select A (Figure 5.47).**]
Location for base view: [**Enter**]

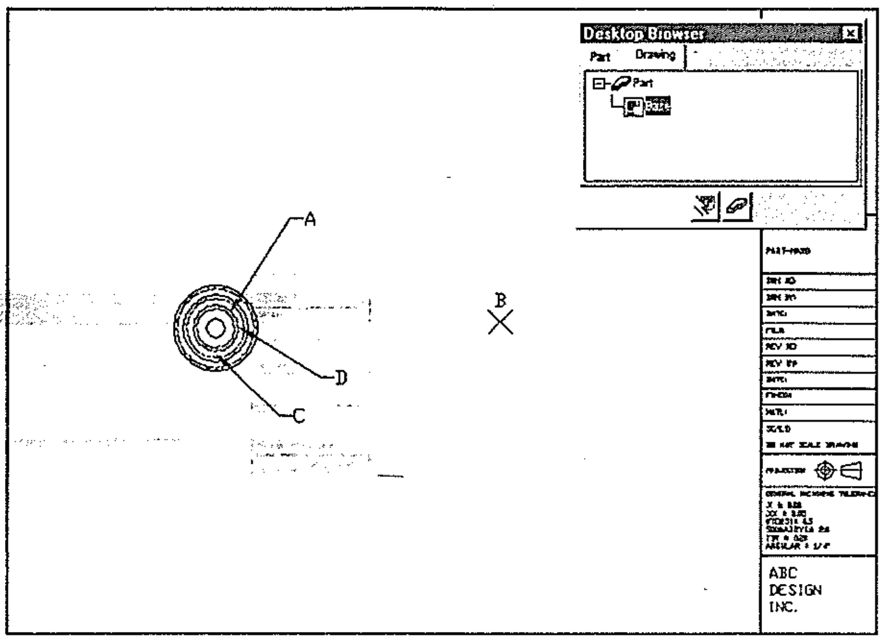


Figure 5.48 Front view constructed

Repeat the AMDWGVIEW command to construct a half section view. (See Figure 5.49.)

<Drawing>   <New View...>

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
Section Views...]

[Section View
Section Type **Half**
Hatching **Hatch**
Pattern...]

[Hatch Pattern
Pattern: **ANSI31**
Scale **1**
Angle **0**
OK]

[Section View
Section Symbol: **A**
OK]

[**OK**]

Select parent view: [**Select A (Figure 5.48).**]
Location for orthographic view: [**Select B (Figure 5.48).**]
Location for orthographic view: [**Enter**]
Section through Point/<Work plane>: [**Enter**]

Select work plane in parent view for the section: **[Select C, the work plane (Figure 5.48).]**
Select second work plane in parent view for the section: **[Select D, the work plane (Figure 5.48).]**
Side of half section Flip/<Accept>: **[Enter]**

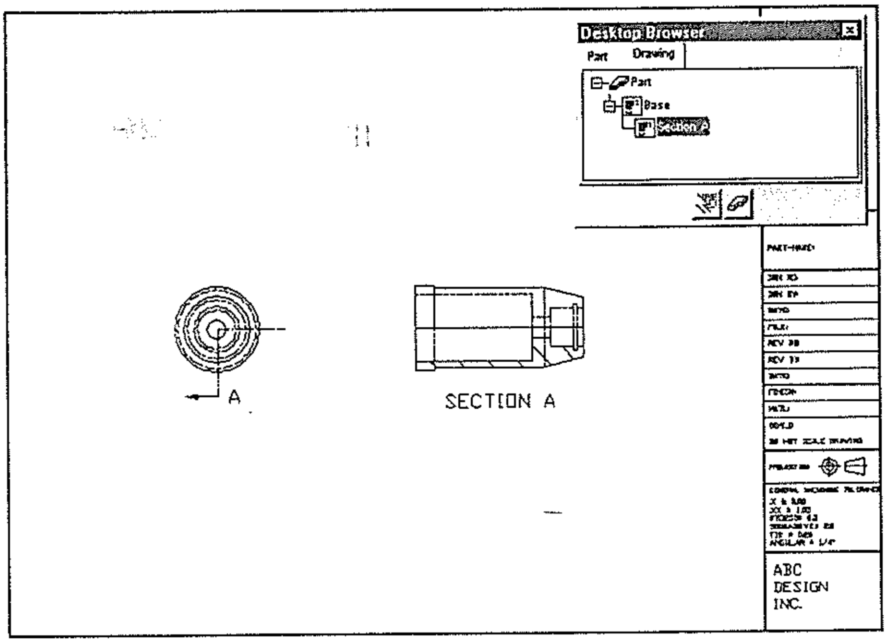


Figure 5.49 Half section side view constructed

The drawing views are complete. Add appropriate dimensions and center lines. Then save your drawing.

<File> **<Save>**

Isometric View

Open the drawing Hwheel.dwg (see Figure 5.50) that you saved in Chapter 3. You will construct a top view, a front view, and an isometric view of this solid part.

<File> **<Open...>**

Command: **Hwheel.dwg**

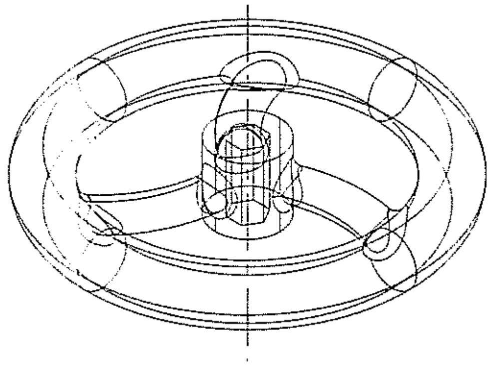


Figure 5.50 Hand wheel

Use the AMPREFS command to set drawing preferences. In the Drawing tab of the Desktop Preferences dialog box, uncheck the Display parametric box to suppress the display of parametric dimensions.

<Drawing> **<Preferences...>**

Use the AMMODE command to enter drawing mode. Then construct a new layer Title and insert a drawing title block.

<Drawing> **<Drawing Mode>**

<Assist> **<Format>** **<Layer...>**

New current layer: **Title**

<Construct> **<Insert Block...>**

Use the AMDWGVIEW command to construct a top view and a front view with a drawing scale of 0.5 as shown in Figure 5.51.

<Drawing> **<New View...>**

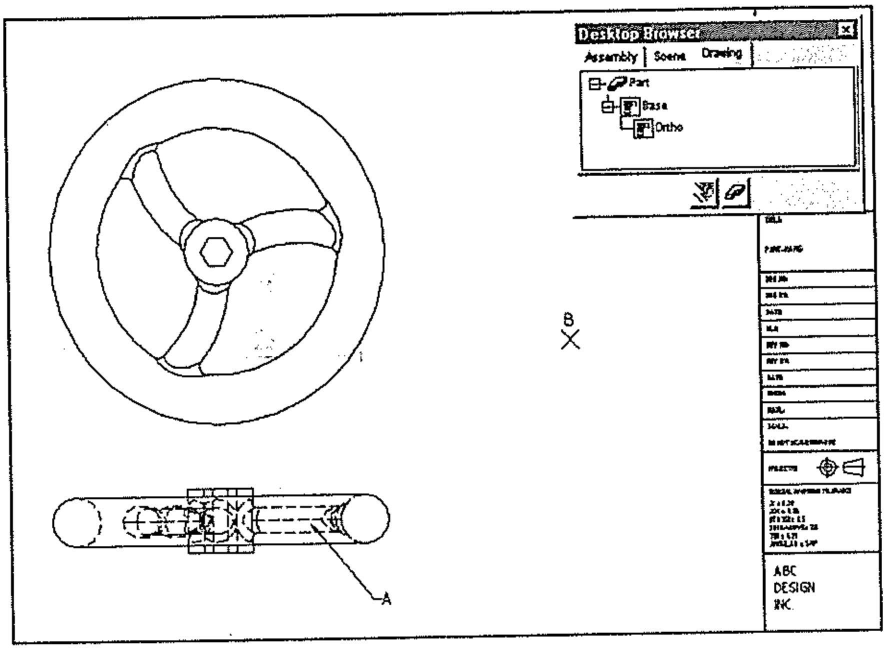


Figure 5.51 Drawing mode, title block inserted and drawing views constructed

Repeat the AMDWGVIEW command to construct an isometric view. (See Figure 5.52.)

<Drawing> <New View...>

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Iso**
Scale: **1** **Relative to Parent**
OK]

Select parent view: [**Select A (Figure 5.51).**]
Location for isometric view: [**Select B (Figure 5.51).**]
Location for isometric view: [**Enter**]

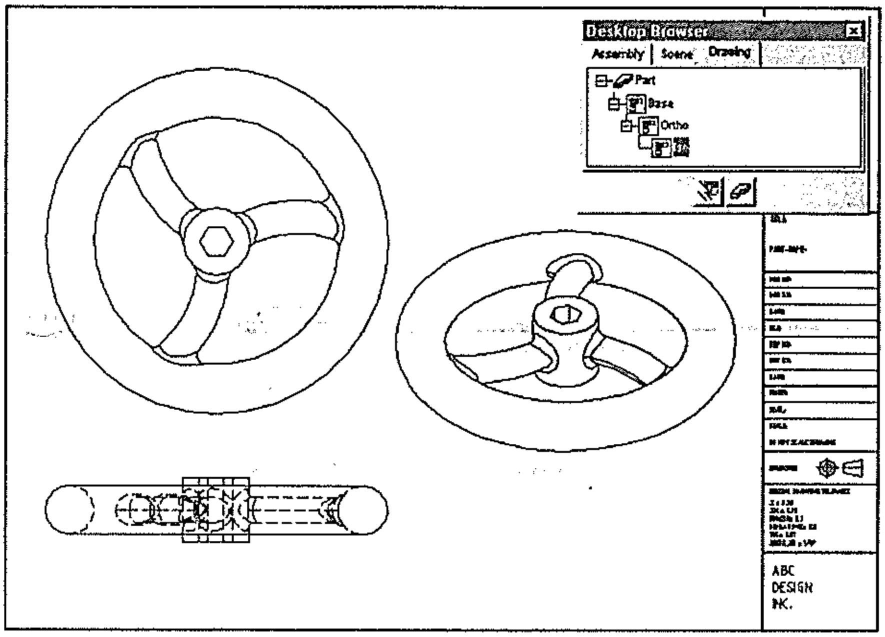


Figure 5.52 Isometric view constructed

The drawing views are complete. Add appropriate center lines and dimensions. Then save your drawing.

<File> <Save>

Auxiliary View

Open the drawing Ubracket.dwg (see Figure 5.53) that you saved in Chapter 3. You will construct a front view, a side view, and an auxiliary view.

<File> <Open...>

Command: **Ubracket.dwg**

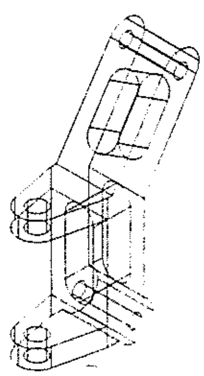


Figure 5.53 U-shaped mounting bracket

Set drawing preferences by using the AMPREFS command. Then use the AMMODE command to enter drawing mode. After that, make a new layer Title and insert the drawing title block.

<Drawing> <Preferences...>

<Drawing> <Drawing Mode>

<Assist> <Format> <Layer...>

New current layer: **Title**

<Construct> <Insert Block...>

As shown in Figure 5.54, construct a top view, a front view, and an isometric view, and add appropriate center lines and dimensions.

<Drawing> <New View...>

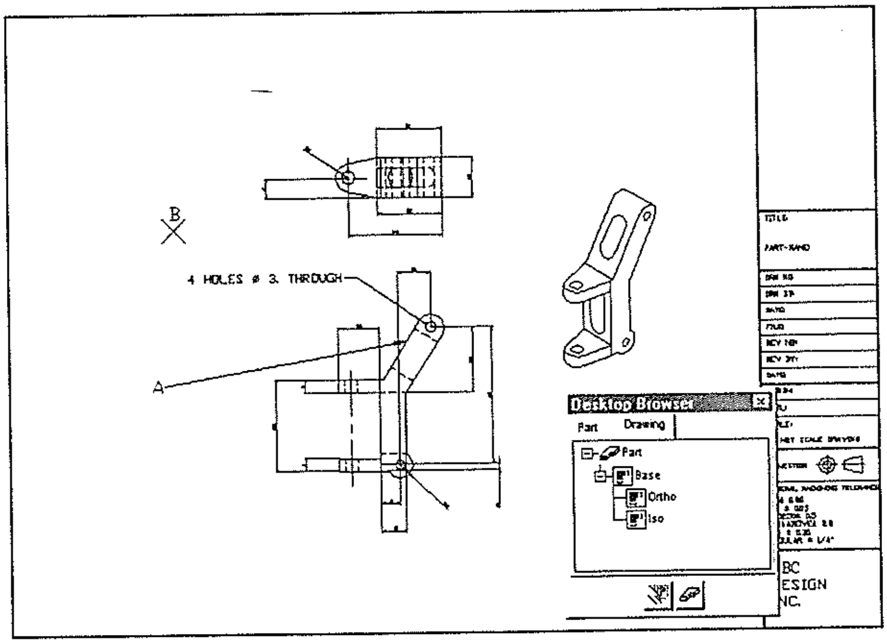


Figure 5.54 Top view, front view, and isometric view constructed

Use the AMDWGVIEW command to add an auxiliary view. Then add appropriate dimensions as shown in Figure 5.55.

<Drawing> <New View...>

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Aux**
OK]

Select a straight edge in the parent view: **[Select A (Figure 5.54).]**
Select second point or <RETURN> to use the selected edge: [**Enter**]
Location for auxiliary view: **[Select B (Figure 5.54).]**
Location for auxiliary view: [**Enter**]

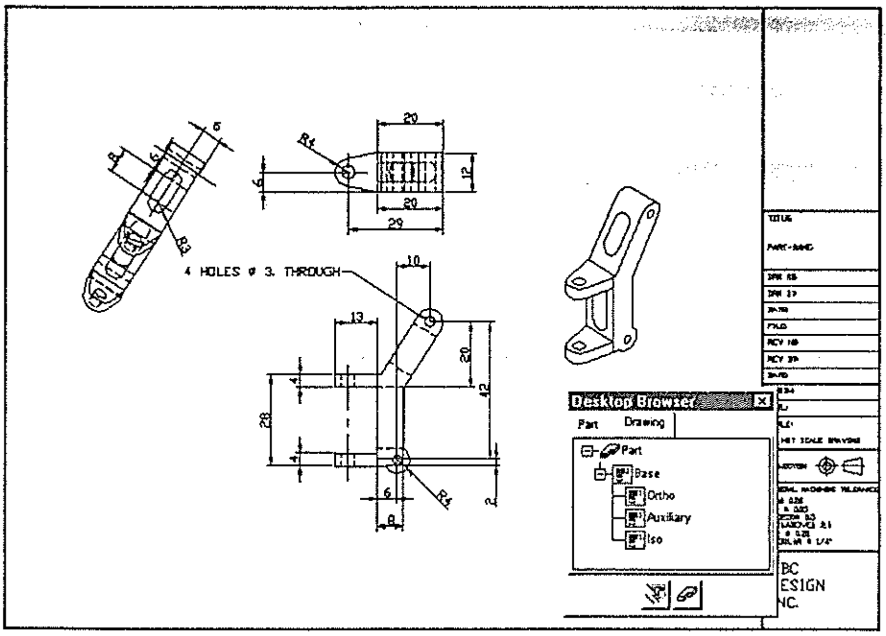


Figure 5.55 Auxiliary view constructed

The drawing is complete. Save your drawing.

<File> **<Save>**

Broken View

It is normal engineering practice to remove the central portion of the drawing view of a very long object and move the remaining portions of the drawing view toward each other to minimize the space requirement of the drawing view. This is called a broken view. Open the drawing file Linkage1.dwg (see Figure 5.56) that you saved in Chapter 4. You will construct a broken view.

<File> **<Open...>**

File name: **Linkage1.dwg**

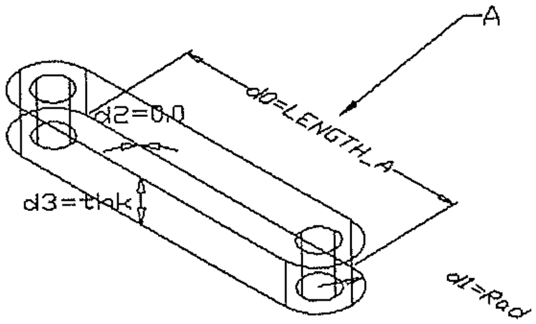


Figure 5.56 A component of the four-bar linkage

Use the AMEDITFEAT command to change dimension A (Figure 5.56) to 150. Then update the solid part and save it to a new file name. (See Figure 5.57.)

<Part> <Edit Feature>

<Part> <Update>

<File> <Save As...>

File Name: **Lbar.dwg**

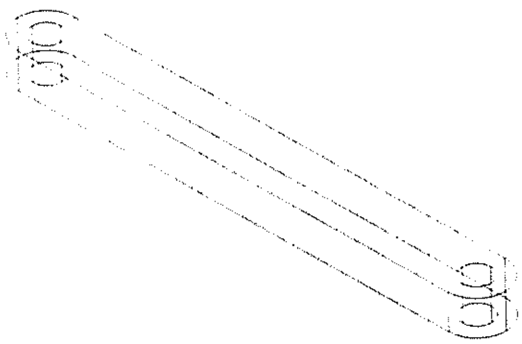


Figure 5.57 Solid part edited and updated

Use the AMPREFS command to set drawing preferences and use the AMMODE command to enter drawing mode. Then add a new layer called Title and insert a drawing title block.

<Drawing> <Preferences...>

<Drawing> <Drawing Mode>

<Assist> <Format> <Layer...>

New current layer: **Title**

<Construct> **<Insert Block...>**

Now construct a broken front view. In your broken view, set the gap between the two broken parts to 10 units. When a temporary view is displayed on your screen, select a vertex of this view. (See Figure 5.62.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Broken** Data Set: **Active Part**
Break Gap: **10**
Scale: **1**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **X**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: [**Enter**]
Location for broken view: [**Select a point near the center of the title block.**]
Location for broken view: [**Enter**]
Select vertex in temporary view for the first or parent sub-view center: [**Select A (Figure 5.58).**]

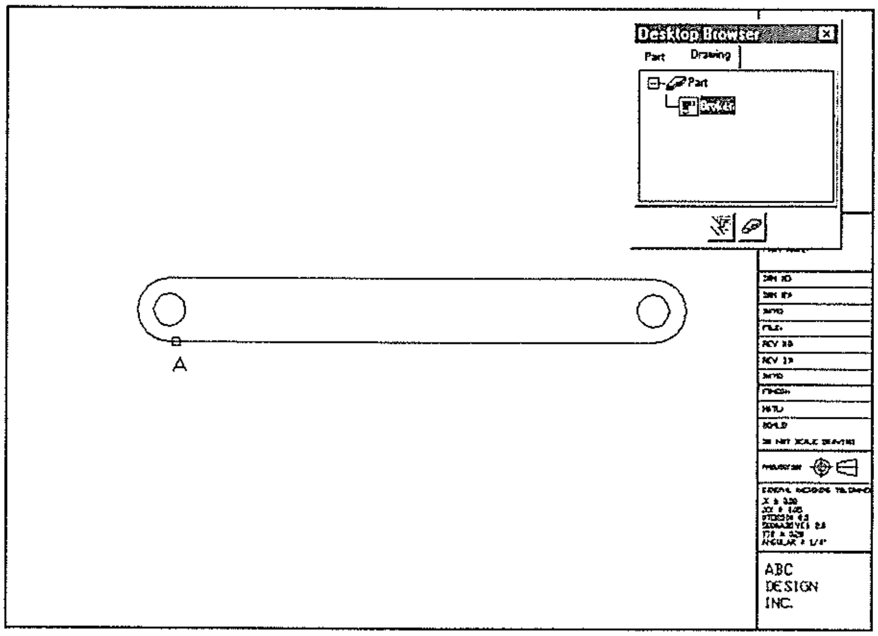


Figure 5.58 Temporary view displayed, a point selected

After selecting a vertex on the temporary view, you are required to drag a rectangular box to define a subview. (See Figure 5.59.)

Drag rectangle to define sub-view: [**Select A (Figure 5.59).**]
Other corner: [**Select B (Figure 5.59).**]

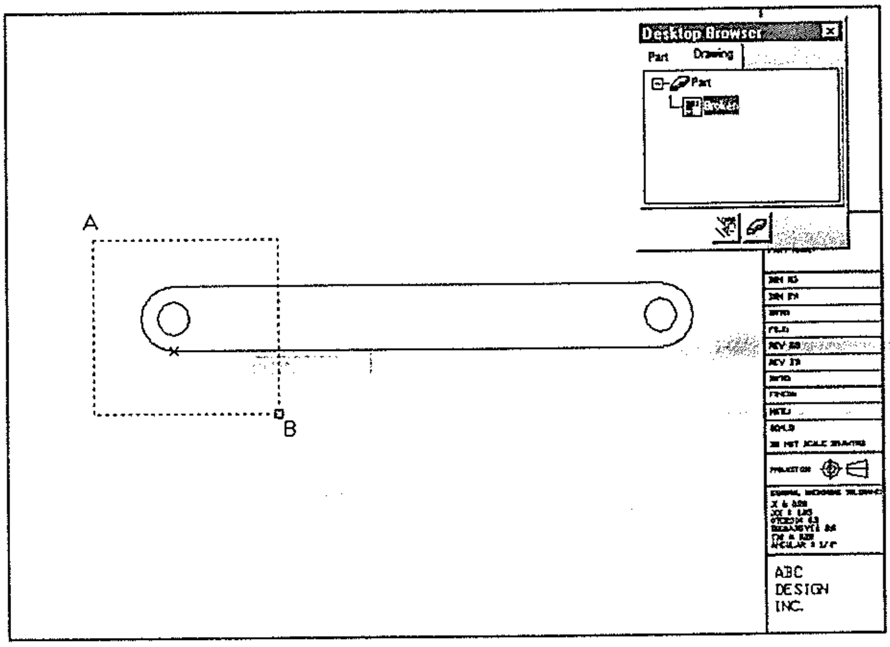


Figure 5.59 A rectangle described

Now select another point on the temporary view. (See Figure 5.60.)

Select vertex in temporary view for second sub-view center: **[Select A (Figure 5.60).]**

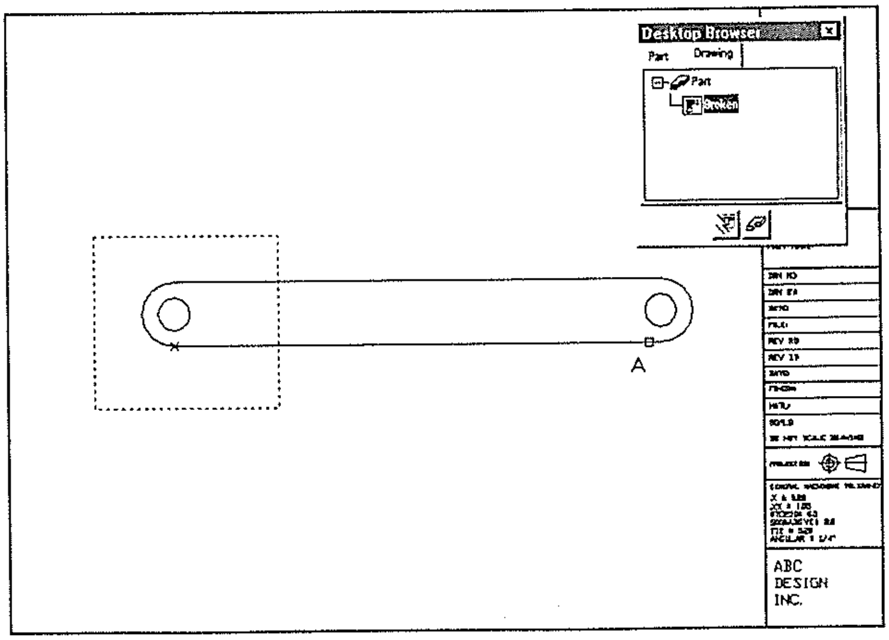


Figure 5.60 Second point selected on the temporary view

Describe another rectangle to define the second subview. (See Figure 5.61.)

Drag rectangle to define sub-view: **[Select A (Figure 5.61).]**
Other corner: **[Select B (Figure 5.61).]**

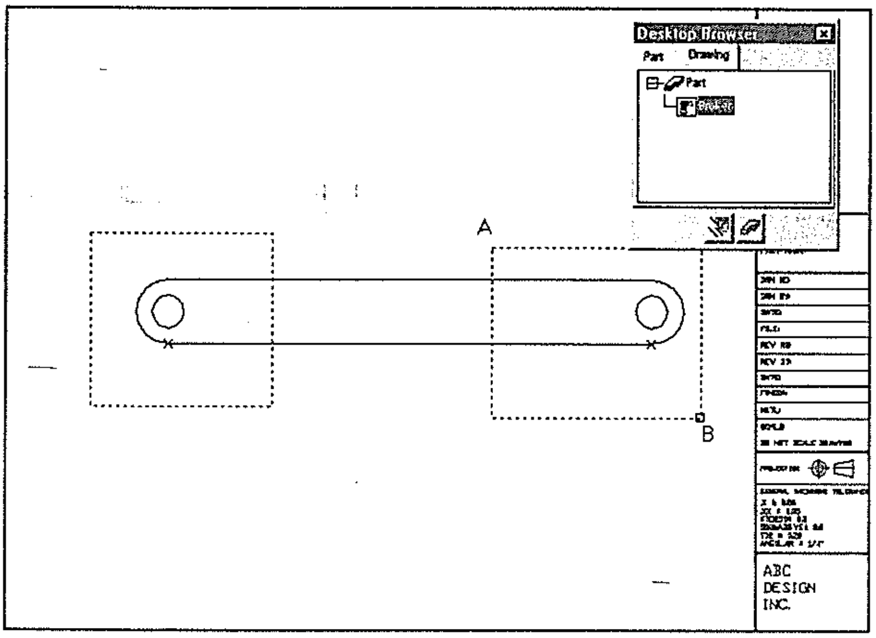


Figure 5.61 Second rectangle described

Because you do not need the third subview, press the [Enter] key to exit.

Select vertex in temporary view for next sub-view center (or Enter when done): **[Enter]**

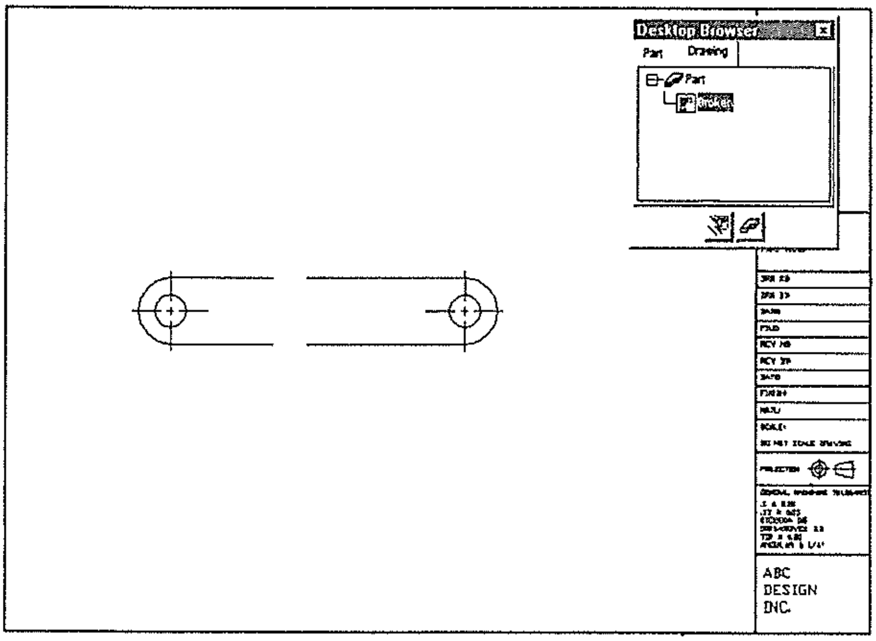


Figure 5.62 Broken view constructed

Add center lines and dimensions as shown in Figure 5.63. Note that the dimensioning is not affected by the broken view.

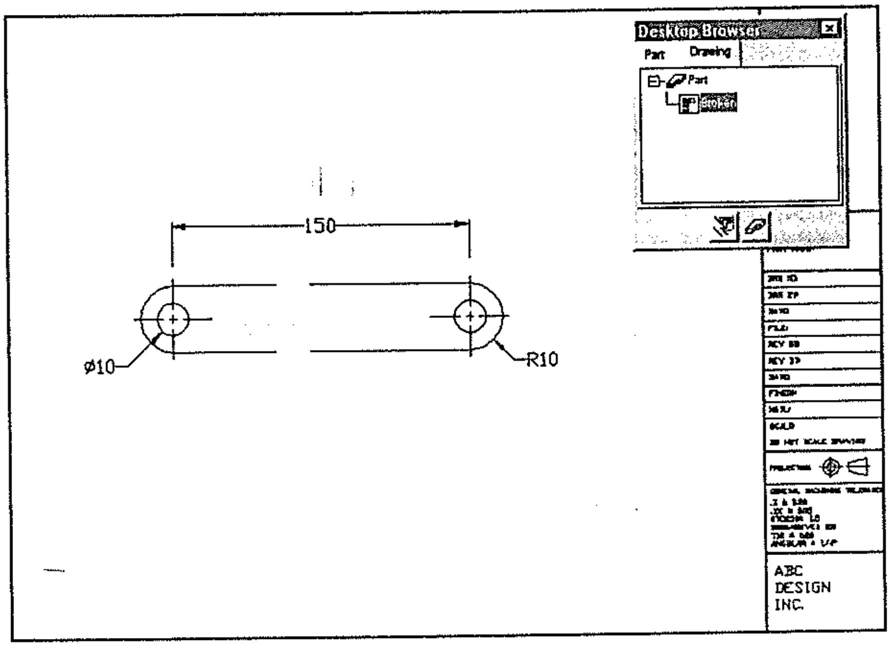


Figure 5.63 Completed drawing

The drawing view is complete. Add appropriate dimensions and center lines. Then save your drawing.

<File> **<Save>**

Detailed View

Open the drawing Ucase.dwg (see Figure 5.64) that you saved in Chapter 4. You will construct a top view, a front view, and a detailed view.

<File> **<Open...>**

File name: **Ucase.dwg**

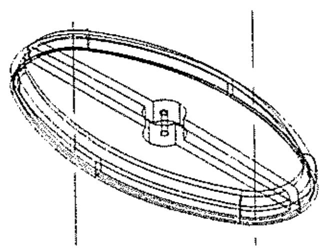


Figure 5.64 Upper casing of the remote control

Set drawing preferences, change to drawing mode, add a new layer called Title, and insert a drawing title block.

<Drawing> **<Preferences...>**

<Drawing> **<Drawing Mode>**

<Assist> **<Format>** **<Layer...>**

New current layer: **Title**

<Construct> **<Insert Block...>**

Use the AMDWGVIEW command to construct a top view and a front view. (See Figure 5.65.)

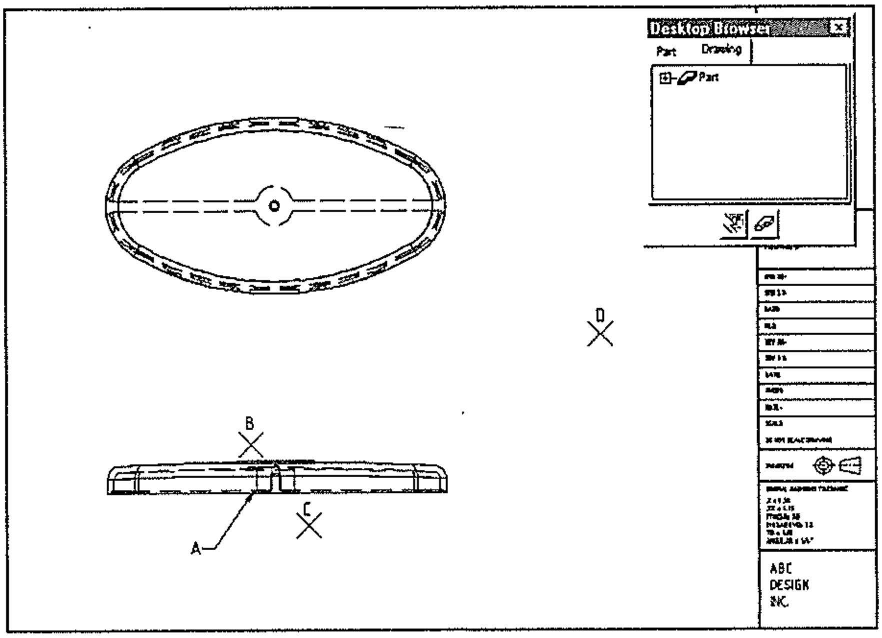


Figure 5.65 Top view and front view constructed

Repeat the AMDWGVIEW command to construct a detailed view. A detailed view is a drawing view showing a magnified portion of one of the drawing views. (See Figure 5.66.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Detail**
Detail View Symbol: **A**
Scale: **2** **Relative to Parent**
OK]

Select vertex in parent view for detail center: **[Select A (Figure 5.65).]**
Drag rectangle around detail: **[Select B (Figure 5.65).]**
Other corner: **[Select C (Figure 5.65).]**
Location for detail view: **[Select D (Figure 5.65).]**
Location for detail view: **[Enter]**

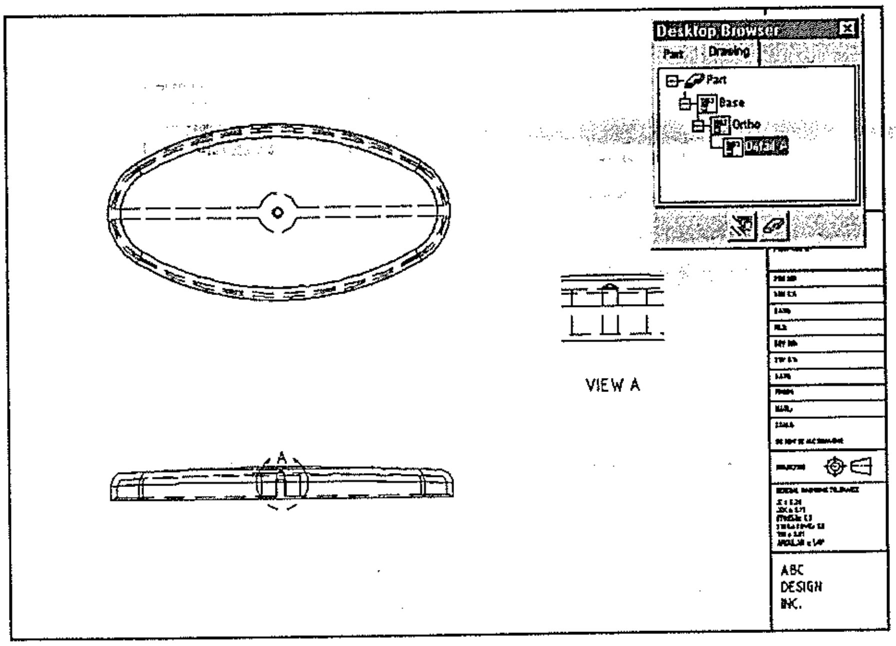


Figure 5.66 Detailed view constructed

The drawing views are complete. Add appropriate dimensions and center lines. Then save your drawing.

<File> **<Save>**

Exploded Assembly Drawing

In an assembly drawing, you can maintain a number of scenes in which you can tweak or explode the solid parts into separate pieces. By constructing drawing views from these scenes, you can have a drawing view that has no tweaking or explosion, or a tweaked or exploded drawing view.

Open the drawing 4bar.dwg. (See Figure 5.67.) You will construct an isometric view of the exploded scene, and a top view and front view of the scene with no explosion.

<File> **<Open...>**

File name: **4bar.dwg**

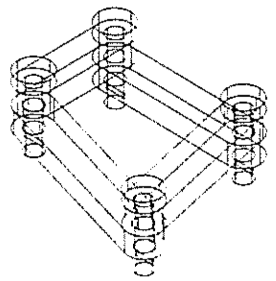


Figure 5.67 Four-bar linkage assembly

Set drawing preferences, enter drawing mode, add a new layer called Title, and insert an engineering drawing block. (See Figure 5.68.)

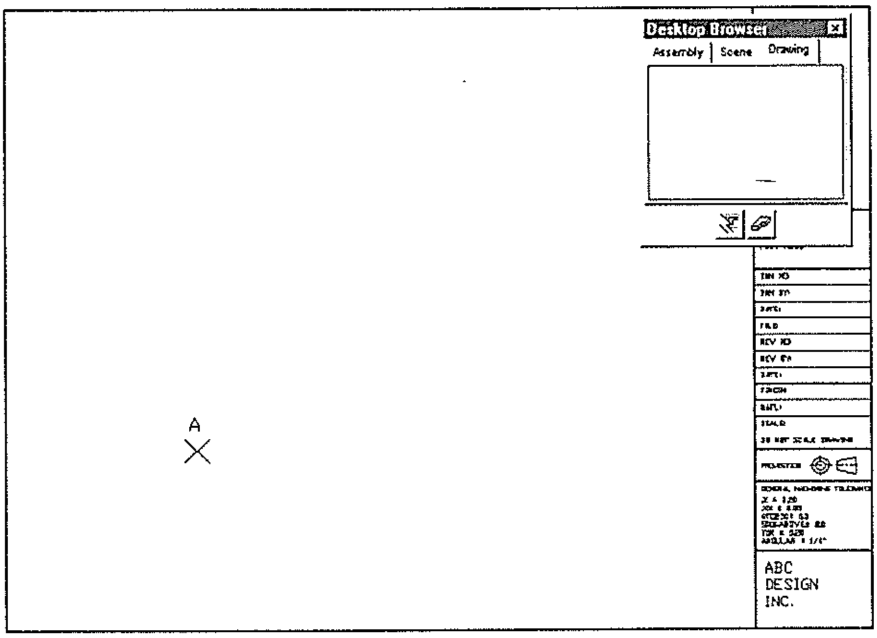


Figure 5.68 Title block inserted in drawing mode

Because an isometric drawing view needs to be constructed from a base view, you will construct a base view. Then you will construct an isometric view. After that, you will delete the base view.

Use the AMDWGVIEW command to construct a drawing view from the exploded scene of the drawing. (See Figure 5.69.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Scene**
Scale: **0.5**
OK]

[Select View
Select Scene: **4bar01**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **Z**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**
Location for base view: **[Select A (Figure 5.68).]**
Location for base view: **[Enter]**

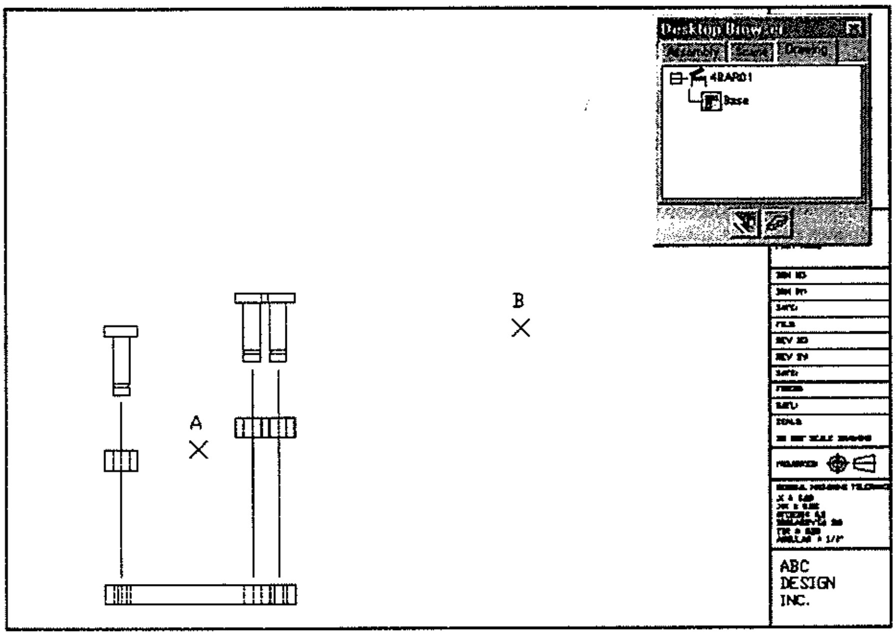


Figure 5.69 Base drawing view of the exploded scene

Repeat the AMDWGVIEW command to construct an isometric view. (See Figure 5.70.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Iso**
Scale: **1** **Relative to Parent**
OK]

Select parent view: **[Select A (Figure 5.69).]**
Location for isometric view: **[Select B (Figure 5.69).]**
Location for isometric view: **[Enter]**

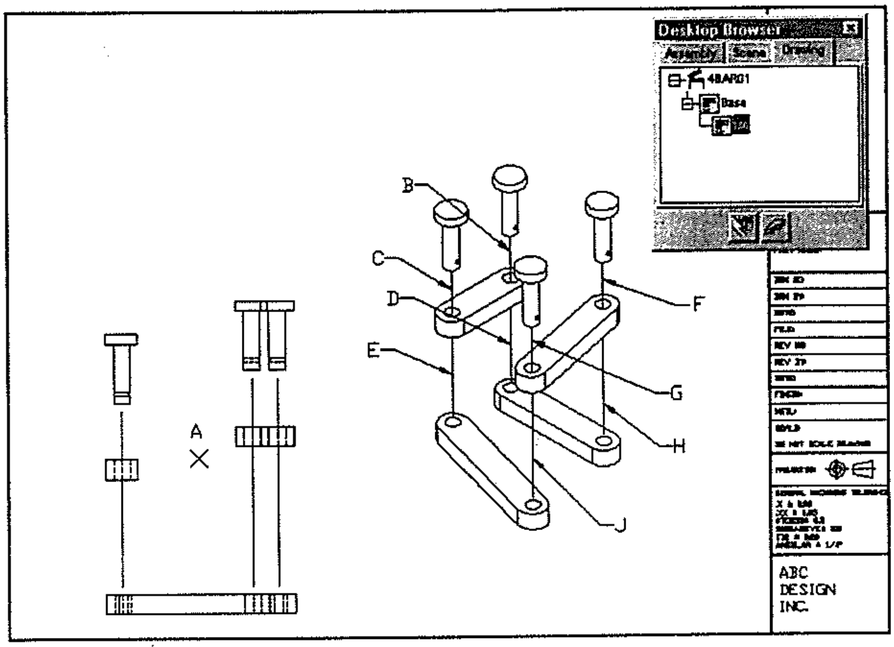


Figure 5.70 Isometric view of the exploded scene constructed

As we have said, the base view is not required in the drawing; the purpose of constructing it is to provide a base view for the isometric view. Therefore, you will now delete this view. After that, use the AMEDITVIEW command to change the linetype of the trailing lines. (See Figure 5.71.)

<Drawing> **<Edit View>** **<Delete>**

Command: **AMDELVIEW**
Select view to delete: [**Select A (Figure 5.70).**]
View has 1 dependent view. Delete it also? Yes/No/<Cancel>: **NO**

<Drawing> **<Edit View>** **<Attribute...>**

Command: **AMEDITVIEW**

[**Edge Properties...**]

Select view to edit: [**Select B (Figure 5.70).**]
Unhide all/<Select>: [**Enter**]
Select Edges: [**Select B, C, D, E, F, G, H, and J (Figure 5.70).**]
Select Edges: [**Enter**]

[Edge Properties
Linetype]

[Select Linetype
Set Linetype: **CENTER**
OK]

[**OK**]

[OK]

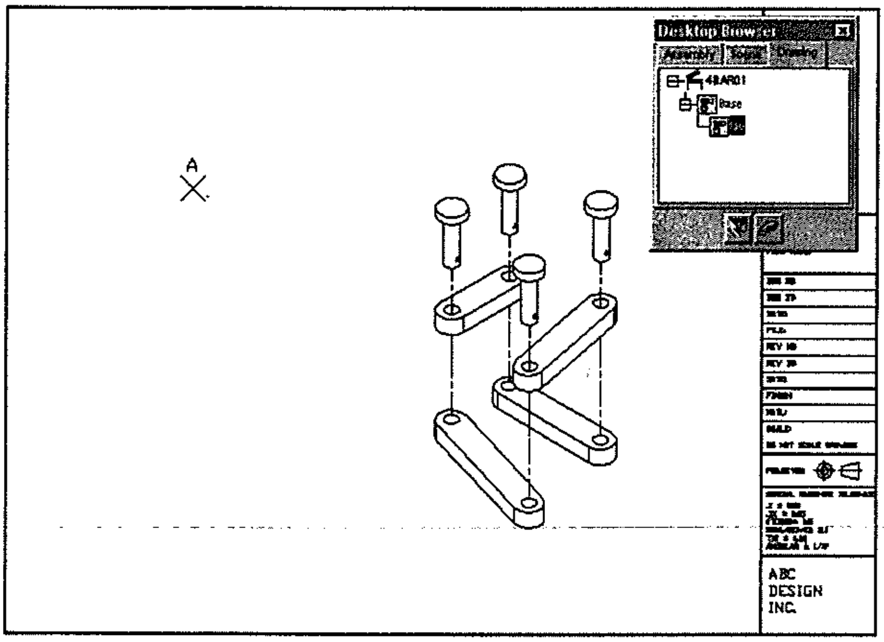


Figure 5.71 Base view deleted, trail line linetype changed

Now construct a top view of the scene with no explosion. (See Figure 5.72.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Scene**
Scale: **0.5**
OK]

[Select View
Select Scene: **4bar02**
OK]

worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **X**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**
Location for base view: **[Select A (Figure 5.71).]**
Location for base view: **[Enter]**

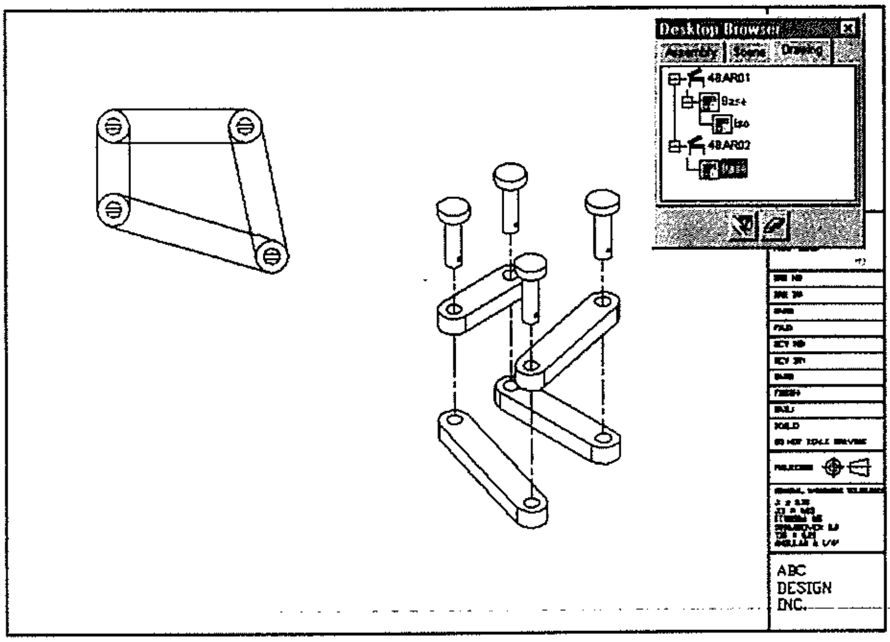


Figure 5.72 Top view of the scene with no explosion constructed

Construct the front view as shown in Figure 5.73.

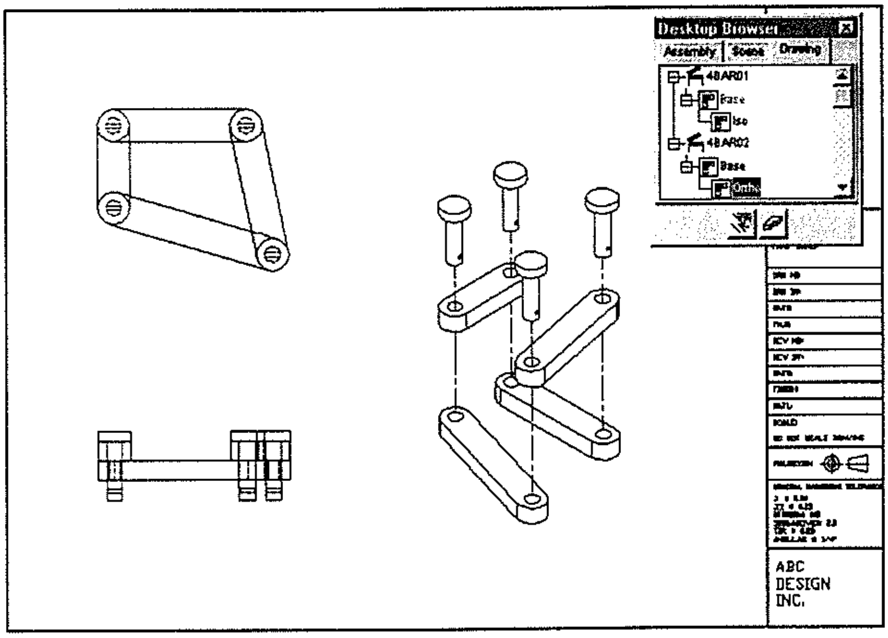


Figure 5.73 Front view constructed

The drawing views are complete. Save your drawing.

<File> <Save>

Sectional Assembly Drawing

Open the drawing Remote.dwg. (See Figure 5.74.) This assembly has two solid parts: the upper casing and the lower casing. You will construct a top view and a section front view of this assembly. To differentiate among various instances of an assembly in a sectional view, you have used the AMPATTERNDEF command to define two different kinds of hatch patterns for the solid parts in Chapter 4. In this drawing, there are two scenes, REMOTE01 and REMOTE02. In scene REMOTE01, the solid parts are tweaked apart. In scene REMOTE02, there is no tweaking.

<File> **<Open...>**

File name: **Remote.dwg**

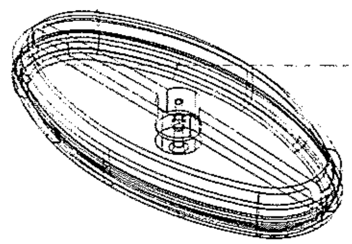


Figure 5.74 Remote control assembly

Set drawing preferences, enter drawing mode, create a layer called Title, and insert a title block. Then construct a top view as shown in Figure 5.75. When constructing the top view, choose scene REMOTE02 as the data set.

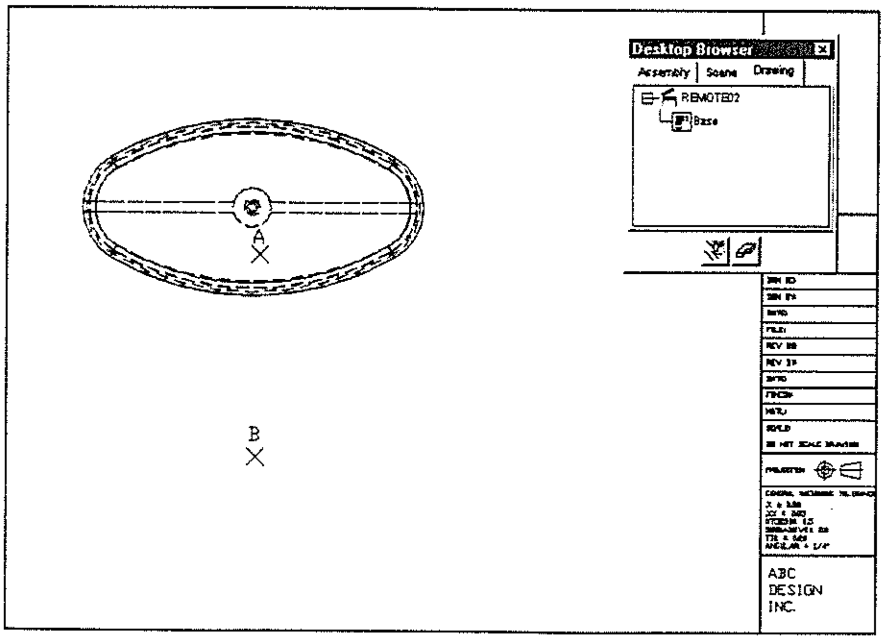


Figure 5.75 Top view of the assembly

Repeat the AMDWGVIEW command to construct a section front view. (See Figure 5.76.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Ortho**
Section Views...]

[Section View
Section Type **Full**
Hatching **Hatch**
Section Symbol: **A**
OK]

[**OK**]

Select parent view: **[Select A (Figure 5.75).]**
Location for orthographic view: **[Select B (Figure 5.75).]**
Location for orthographic view: **[Enter]**
Section through Point/Ucs/<Work plane>: **[Enter]**
Select work plane in parent view for the section: **[Select the horizontal work plane from the top view.]**

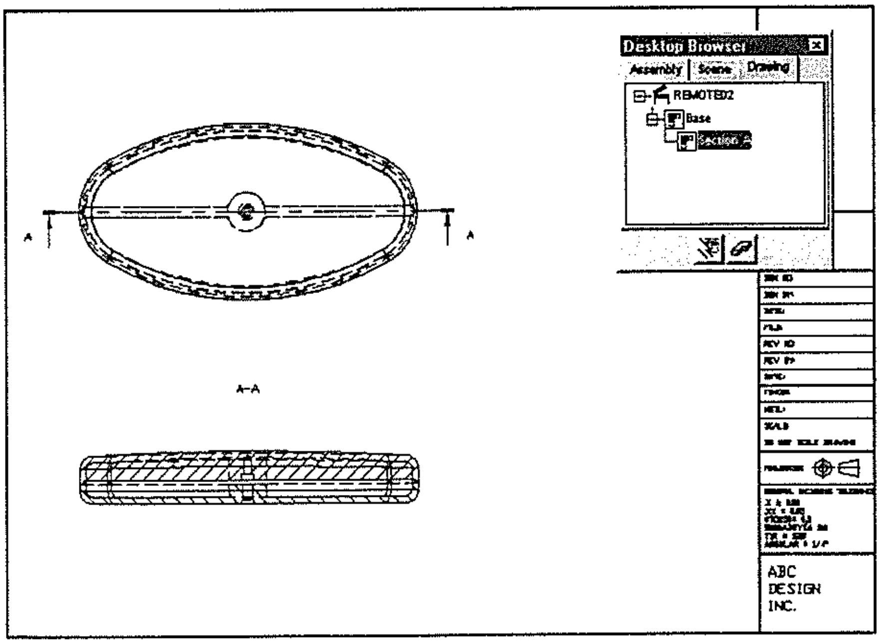


Figure 5.76 Sectional front view constructed

The drawing views are complete. Save your drawing.

<File> **<Save>**

Suppression in Scene Mode

You can suppress sectioning on selected solid parts in a section view of an assembly. If you have closed the file, open it again.

<File> **<Open...>**

File name: **Remote.dwg**

From the browser, select the Scene tab to enter scene mode. Then activate scene REMOTE02 (the scene without tweaking). (See Figure 5.77.)

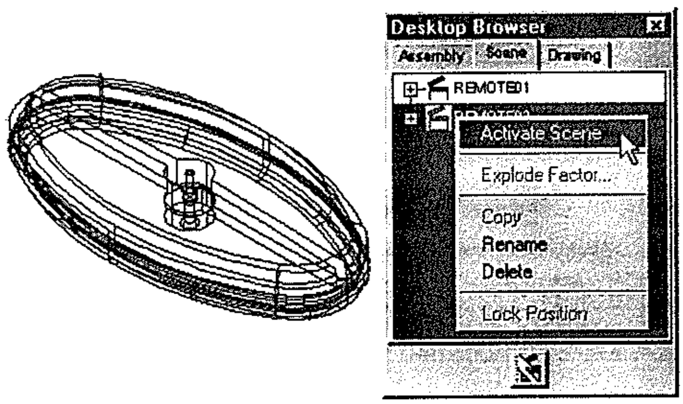


Figure 5.77 Activating a scene in the browser

To see how a solid part can be suppressed in a drawing view, use the AMSUPPRESS command to suppress the upper casing of this scene. (See Figure 5.78, the Instance Suppression dialog box.) Select Ucase_1 (or the name that you assigned to the solid part of the upper casing of the assembly). Then select the [Suppress] button. After that, select the [OK] button. This way, sectioning of the upper casing will be suppressed.

<Assembly> **<Scene>** **<Suppress Section...>**

Command: **AMSUPPRESS**

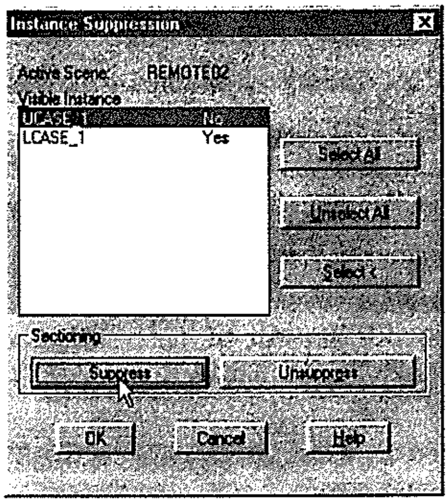


Figure 5.78 Instance Suppression dialog box

Now enter drawing mode by selecting the drawing tab of the browser. Then use the AMUPDATEDWGVIEW command to update the drawing views. (See Figure 5.79.)

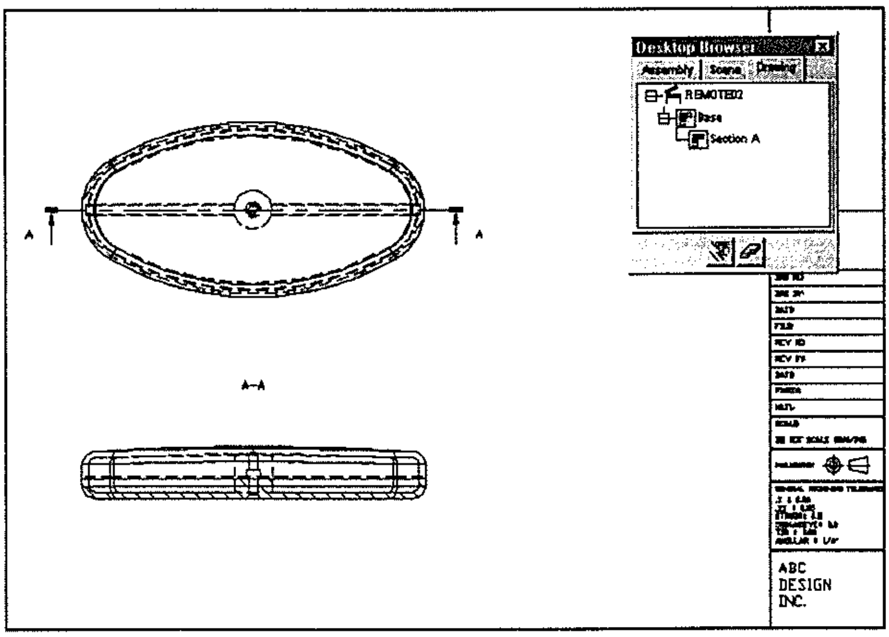


Figure 5.79 Upper casing suppressed in the scene

Now the upper casing is not sectioned in the section front view. To unsuppress sectioning, you can use the AMSUPPRESS command again. As shown in Figure 5.78, select the suppressed object. Then select the [Unsuppress] button. (See Figure 5.76 again.)

Save your drawing.

<File> **<Save>**

Bidirectional Associativeness

A solid part and its associated engineering drawing are bidirectionally associated to each other. If you edit a feature in model mode, the associative drawing follows the change. If you change a parametric dimension in drawing mode, the solid in model mode will change as well. Open the drawing Mblock.dwg. (See Figure 5.80.)

<File> **<Open...>**

File name: **Mblock.dwg**

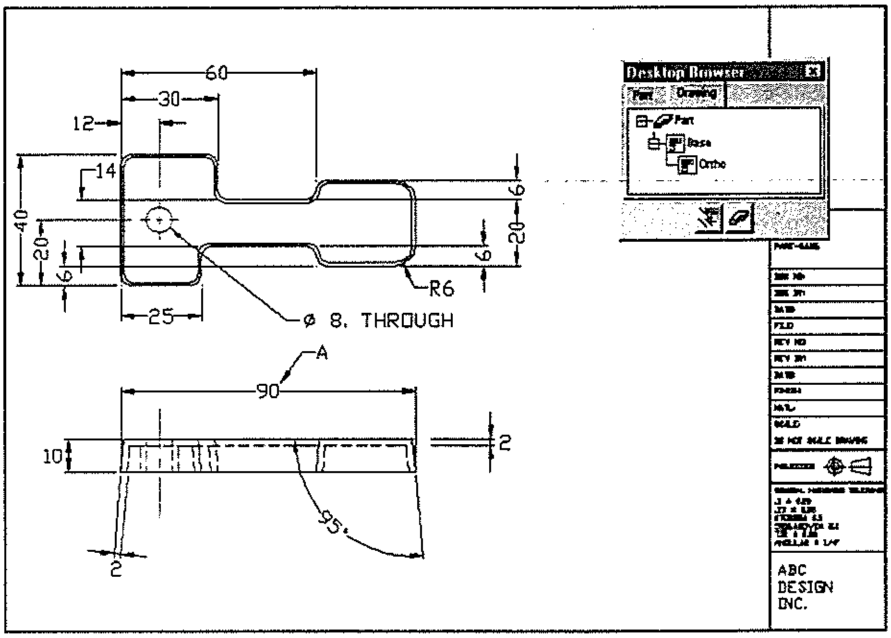


Figure 5.80 Drawing of the mounting block

On your screen, there are two kinds of dimensions: parametric dimensions and reference dimensions. By default, the color of the parametric dimensions is green and the color of the reference dimensions is cyan. Use the AMMODDIM command to change dimension A (Figure 5.80) to 120. Then use the AMUPDATE command to update the model. (See Figure 5.81.)

<Part> **<Change Dimension>**

Command: **AMMODDIM**
Select dimension to change: [**Select A (Figure 5.80).**]
New value for dimension: **120**
Select dimension to change: [**Enter**]

<Part> **<Update>**

Command: **AMUPDATE**

<Drawing> **<Update Drawing View>**

Command: **AMUPDATEDWGVIEW**
Select/<All>: **[Enter]**

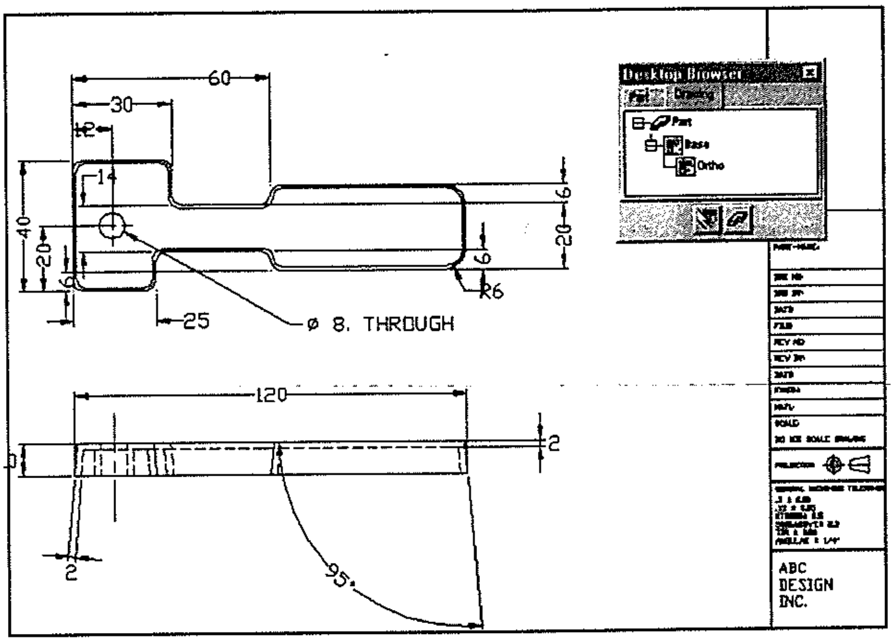


Figure 5.81 Drawing updated

After updating, change to model mode. (See Figure 5.82.) To see the bidirectional associativeness, use the AMEDITFEAT command to change dimension A (Figure 5.82) to 90.

<Part> **<Edit Feature>**

Command: **AMEDITFEAT**
Independent array instance/Sketch/surfCut/Toolbody/<select Feature>: **[Select the solid.]**
Select object: **[Select A (Figure 5.82).]**
Enter new value for dimension: **90**
Select object: **[Enter]**

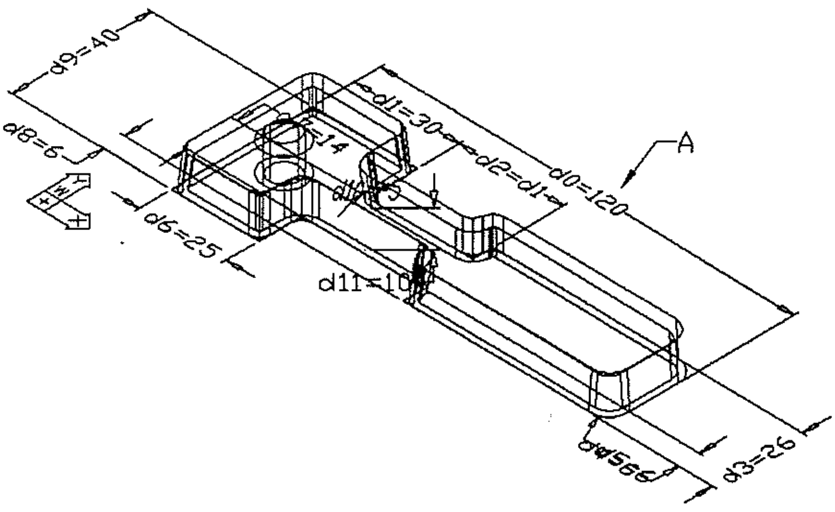


Figure 5.82 Part updated

Update the change. After you change to drawing mode, the drawing views should update automatically. If not, use the AMUPDATEDWGVIEW command. (See Figure 5.80 again.) Save your drawing.

<File> **<Save>**

Dimension Display and OLE Objects

As we said in Chapter 3, a parametric dimension can be displayed in one of three forms: as a number, as a parameter, or as an equation. In an associative engineering drawing, you can also display the parametric dimensions in one of the three formats. Open the drawing Bearing1.dwg (see Figure 5.83) that you saved in Chapter 3. Then save the drawing with a new file name.

<File> **<Open...>**

File name: **Bearing1.dwg**

<File> **<Save As...>**

File name: **Bearing3.dwg**

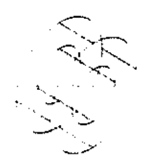


Figure 5.83 Bearing

Set drawing preferences, enter drawing mode, add a layer called Title, insert a drawing title block, and construct two drawing views as shown in Figure 5.84.

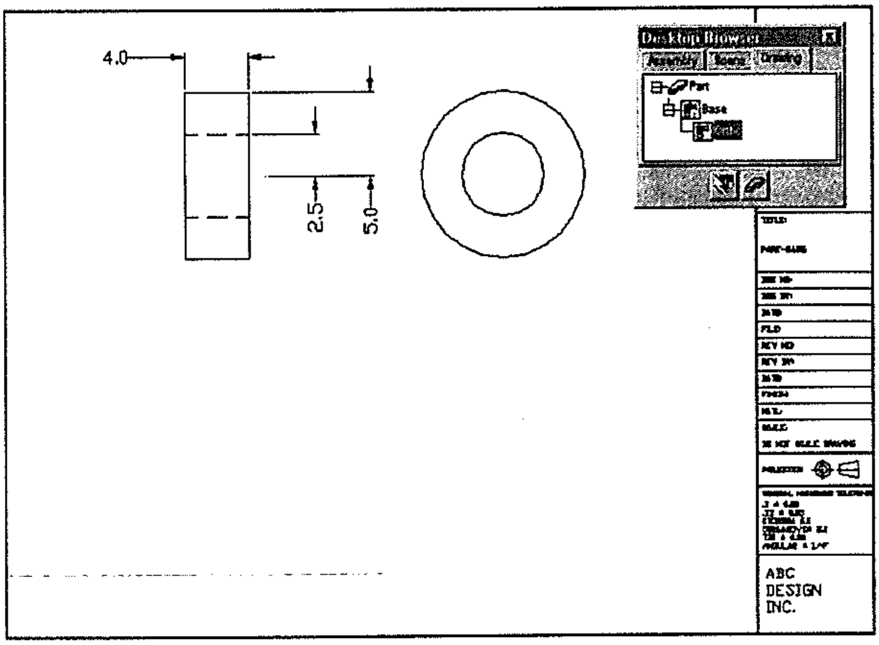


Figure 5.84 Drawing of the bearing

To display the parametric dimensions as parameters, select the Dimensions as Parameters of the Parametric Dim Display cascading menu of the Drawing pull-down menu. (See Figure 5.85.)

<Drawing> **<Parametric Dim Display>** **<Dimensions as Parameters>**

All/View/<Select dimensions>: **ALL**

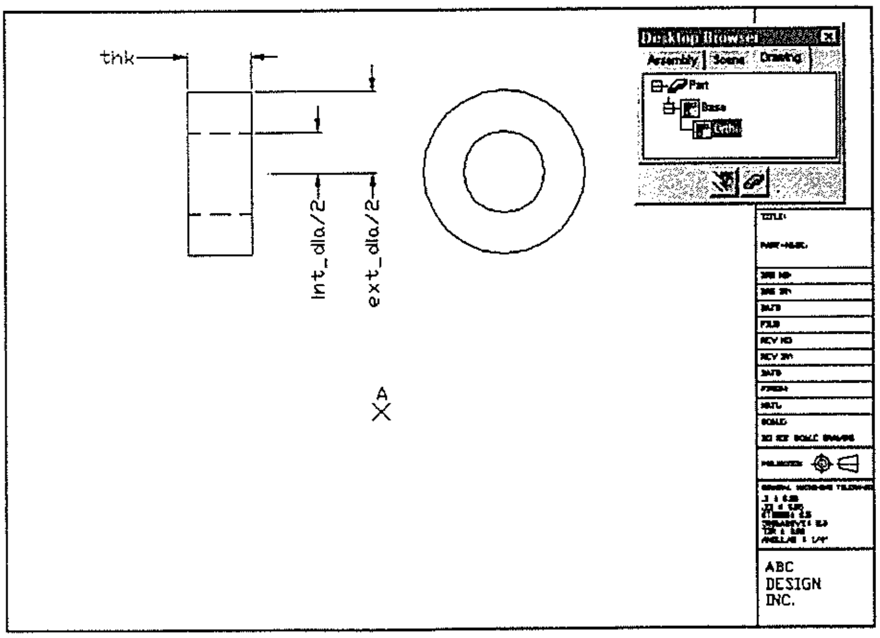


Figure 5.85 Parametric dimensions displayed as parameters

Now use the INSERTOBJ command to insert the Excel table that you constructed to define the design variables of this solid part. As in Figure 5.86, the Insert Object dialog box, select Microsoft Excel Worksheet in the Object Type list box. Then check the Create from File box. Then, as in Figure 5.87, select the Excel file that you saved in Exercise 3.4. If you cannot locate the file, select the [Browse] button to find the file. Check the [Link] box. Then select the [OK] button. After that, select the inserted Excel worksheet and drag it to A (Figure 5.85). (See Figure 5.88.)

Command: **INSERTOBJ**

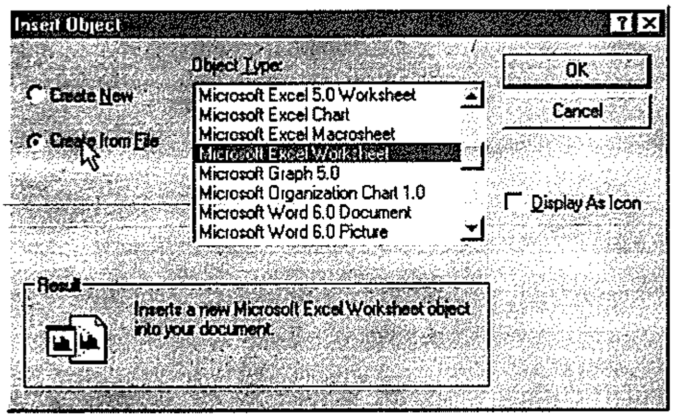


Figure 5.86 Insert Object dialog box

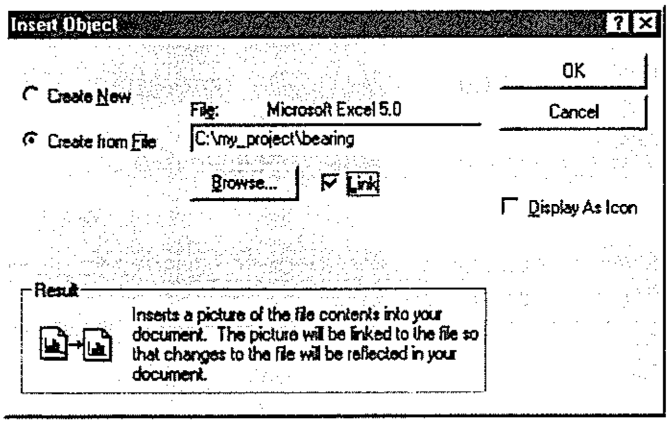
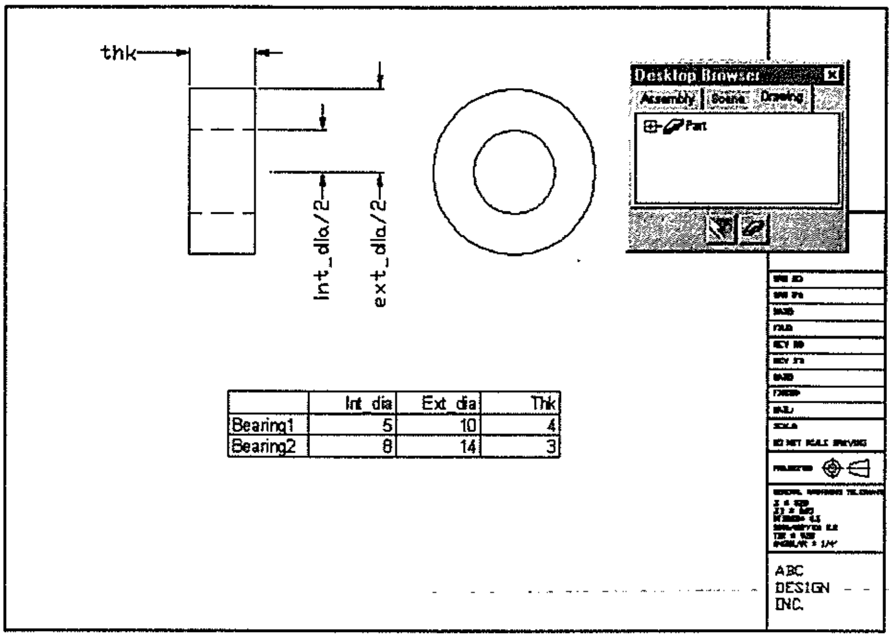


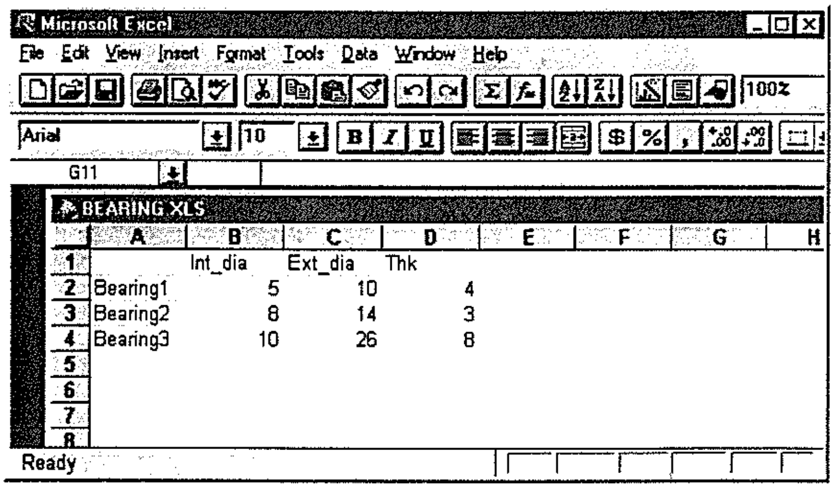
Figure 5.87 Microsoft Excel file selected



	Int_dia	Ext_dia	Thk
Bearing1	5	10	4
Bearing2	8	14	3

Figure 5.88 Excel worksheet inserted

Because the parametric dimensions are driven by the worksheet, change in the Excel worksheet will cause change in the solid part. Select the worksheet. Then double-click to activate the Excel worksheet file. Add a row as shown in Figure 5.89. Then save and close the Excel worksheet file. After that, change to model mode and use the AMVARS command to update the link to the Excel worksheet and select the Bearing3 item.



	A	B	C	D
1		Int_dia	Ext_dia	Thk
2	Bearing1	5	10	4
3	Bearing2	8	14	3
4	Bearing3	10	26	8

Figure 5.89 Excel worksheet activated

To update the drawing, change back to drawing mode, double-click on the Excel table, save the Excel table once again, and return to the drawing. (See Figure 5.90.)

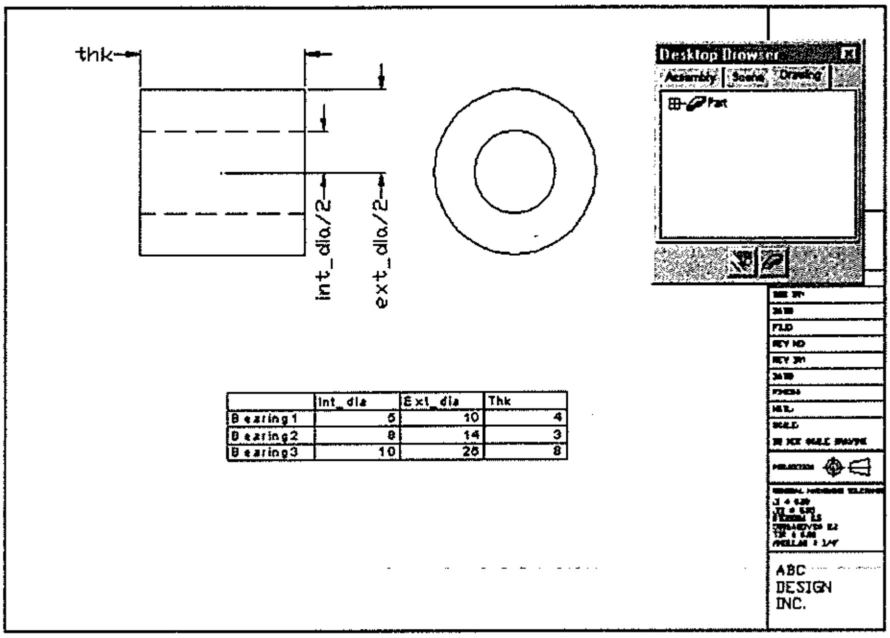


Figure 5.90 Solid part changed, drawing updated

Save your drawing.

<File> <Save>

NURBS Surfaces

A NURBS surface model can be used for many purposes. You can use it to manufacture the component depicted by the model or to create a photo-realistic rendering. Sometimes you might simply want to construct a 2D engineering drawing from it. Open the drawing Joypad.dwg that you saved in Chapter 2. (See Figure 5.91.)

<File> <Open...>

File name: **Joypad.dwg**

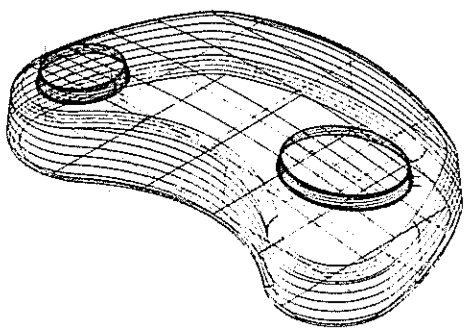


Figure 5.91 Joy pad

Set drawing preferences, enter drawing mode, add a new layer called Title, and insert a drawing title block. After that, use the AMDWGVIEW command. In the Create Drawing View dialog box, there is only one kind of Data Set option because there is no solid part and no assembly, and there are no entity groups. In the Hidden Lines dialog box, uncheck the Calculate hidden lines box to suppress hidden line calculation. After that, select the [OK] button. (See Figure 5.92.)

<Drawing> **<New View...>**

Command: **AMDWGVIEW**

[Create Drawing View
Type: **Base** Data Set: **Select**
Scale: **0.5**
Hidden Lines...]

[Hidden Lines
Uncheck the [Calculate hidden lines] button.
OK]

[OK]

Select entities to be included in view:
Select objects: **[Select all the surfaces.]**
Select objects: **[Enter]**
worldXy/worldYz/worldZx/Ucs/View/<Select work plane or edge>: **X**
worldX/worldY/worldZ/<Select work axis or straight edge>: **X**
Rotate/Z-flip/<Accept>: **[Enter]**
Location for base view: **[Select the upper left of the title block.]**
Location for base view: **[Enter]**

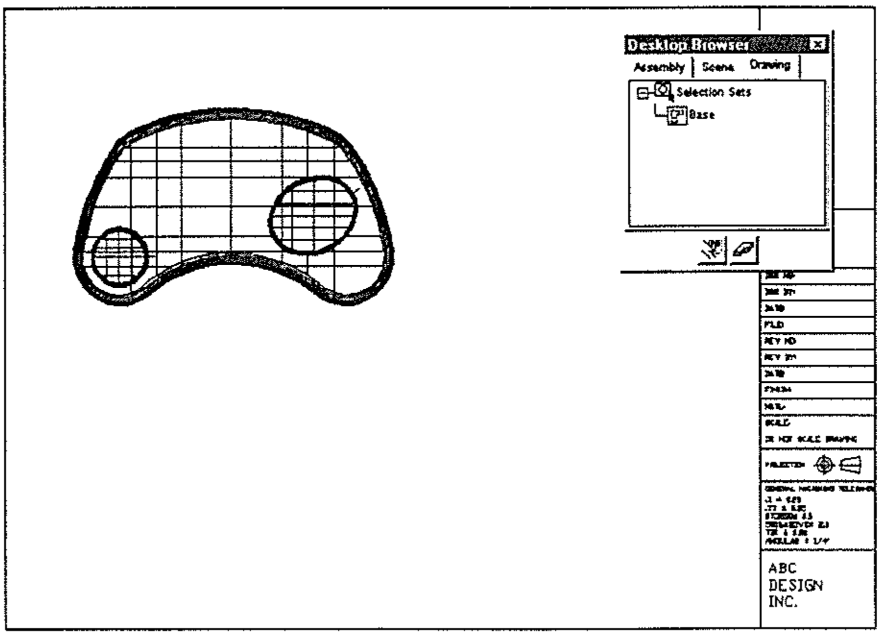


Figure 5.92 Top view constructed

Construct a front view and an isometric view as shown in Figure 5.93.

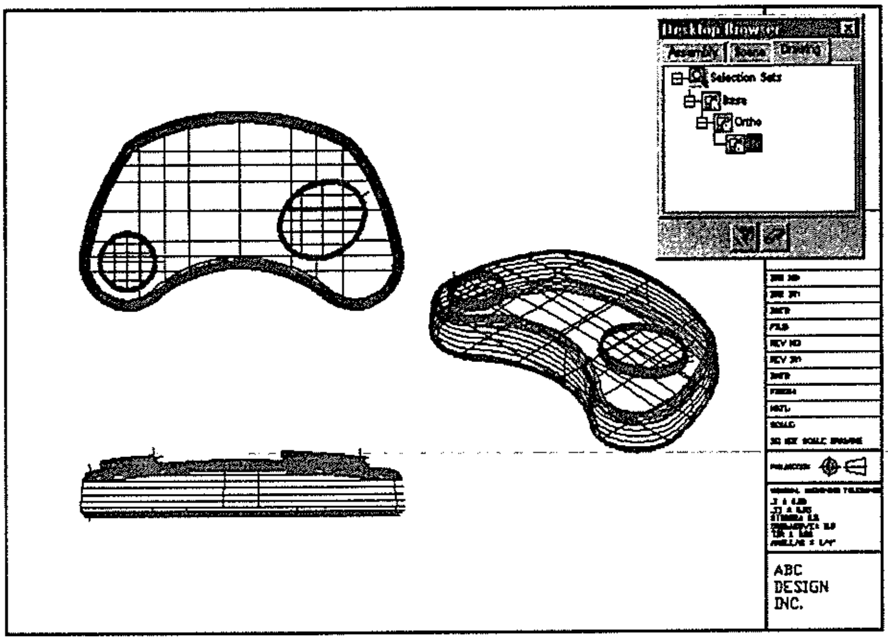


Figure 5.93 Front view and isometric view constructed

Although it is common practice to add dimensions to an engineering document, you will not add dimensions to this drawing, because the document shows a set of NURBS surfaces. Therefore, the drawing views are merely silhouettes of the surface model, each in a particular viewing direction. Dimensions set on these silhouettes do not seem to provide any real or useful engineering meaning. To use a surface model for manufacturing, we strongly advise that you use electronic data transmission. To output a surface model, you can use the IGESOUT command or the AMMODOUT command.

You have completed the 2D engineering document for the surface model. Save your drawing.

<File> **<Save>**

Outputting a 2D Engineering Drawing

Your drawing has two modes, drawing mode and model mode. If you wish to output a purely 2D drawing from the drawing mode of a 3D drawing, you can issue the AMDWGOUT command.

<Drawing> **<Utilities>** **<Drawing Out...>**

5.4 Annotations, Dimensions, and Symbols

In addition to generating orthographic views from the 3D objects (3D NURBS surfaces, 3D solids, and 3D assemblies), you have to include annotations, dimensions, tolerance symbols, surface texture symbols, and welding symbols in an engineering drawing.

Annotations

Open the drawing Mblock.dwg.

<File> **<Open...>**

File name: **Mblock.dwg**

There are two major kinds of text objects: single-line text and multi-line text. To add single-line text, you can use the DTEXT command. To add multi-line text, you can use the MTEXT command. Before adding text to a drawing, you have to set the text style by using the STYLE command.

<Drawing> **<Annotation Styles>** **<Text...>**

Command: **STYLE**

If you want to use the text style of an existing drawing but you do not want to insert the drawing, you can import the text style by using the AMSTYLEI command.

<Drawing> **<Annotation Styles>** **<Importer...>**

Command: **AMSTYLEI**

Use the DTEXT command to add single-line text as shown in Figure 5.94.

<Drawing> **<Annotate>** **<Line Text>**

Command: **DTEXT**
Justify/Style/<Start point>: **[Select A (Figure 5.94).]**
Height: **4**
Rotation angle: **0**
Text: **SHELL THICKNESS 3mm**
Text: **[Enter]**

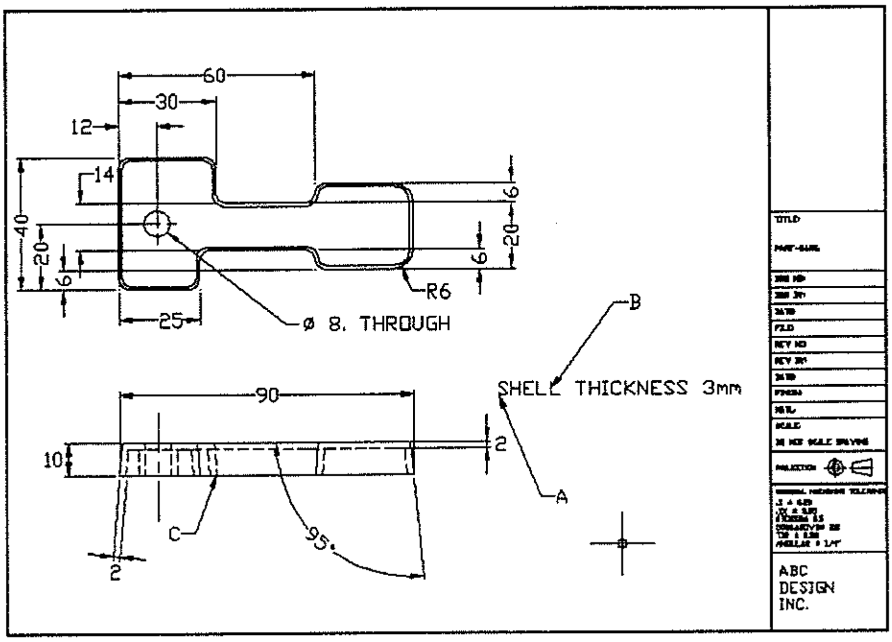


Figure 5.94 Single-line text added

For the text to move with the drawing view if the latter is moved, you can attach the text (single-line or multi-line) to a selected drawing view by using the AMANNOTE command. This command has four options: add, delete, move, and create. Now create a link for the text to be attached to a selected point on the drawing view.

<Drawing> **<Annotation Object>** **<Create>**

Command: **AMANNOTE**
Add/Delete/Move/<Create>: **[Enter]**
Select objects to associate to view.
Select objects: [**Select B (Figure 5.94).**]
Select objects: [**Enter**]
Select point in view to attach annotation: **[Select C (Figure 5.94).]**

The single-line text is now attached to the front view. To appreciate how the text is attached, you can use the AMMOVEVIEW command to move the view C (Figure 5.95) to a new position. To move the annotation relative to the attached drawing view, use the AMANNOTE command again.

<Drawing> **<Annotation Object>** **<Move>**

Command: **AMANNOTE**
Add/Delete/Move/<Create>: **MOVE**
Select annotation by selecting any of its objects: [**Select B (Figure 5.94).**]
Annotation location: [**Select a new position.**]
Annotation location: [**Enter**]

Now construct another line of text as shown in Figure 5.95. The ° symbol is constructed by typing %%D. After constructing the text, use the AMANNOTE command to add it to the attached text.

<Drawing> <Annotate> <Line Text>

Command: **DTEXT**
Justify/Style/<Start point>: **[Select A (Figure 5.94).]**
Height: **4**
Rotation angle: **0**
Text: **DRAFT ANGLE 5%%D**
Text: **[Enter]**

<Drawing> <Annotation Object> <Add To>

Command: **AMANNOTE**
Add/Delete/Move/<Create>: **ADD**
Select annotation by selecting any of its objects: **[Select C (Figure 5.95).]**
Select objects to add to annotation.
Select objects: **[Select B (Figure 5.95).]**
Select objects: **[Enter]**

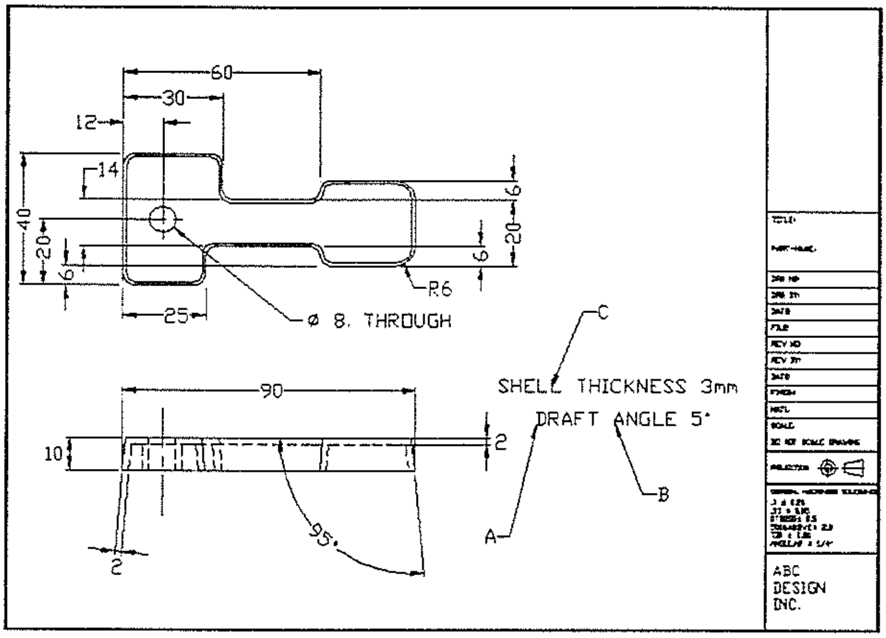


Figure 5.95 Second text object added to the first

There are two more options: Detach and Remove.

Command: **AMANNOTE**
Add/Remove/Delete/deTach/Move/<Create>: **T or R**

To edit text, use the DDEDIT command.

<Drawing> **<Annotate>** **<Edit Text...>**

Command: **DDEDIT**

To check the spelling of text, use the SPELL command.

<Drawing> **<Annotate>** **<Spelling>**

Command: **SPELL**

To add a hole note to a hole feature of a solid part, you can use the AMHOLENOTE command.

<Drawing> **<Annotate>** **<Hole Note...>**

Command: **AMHOLENOTE**

To edit the template for making a hole note, you can use the AMTEMPLATE command.

<Drawing> **<Annotate>** **<Template...>**

Command: **AMTEMPLATE**

To edit the text context of an attached hole note, you can use the DDEDIT command.

Dimensioning

The way a dimension is displayed depends on the dimension style. To set the dimension style, you can use the DDIM command.

<Drawing> **<Annotation Styles>** **<Dimension...>**

Command: **DDIM**

Parametric dimensions that are used to construct a sketched solid feature appear in the drawing views. These dimensions, though valid in terms of constraining the sketch and the solid part, are sometimes wrongly placed or are irrelevant to manufacturing. Before you can release the drawing to the workshop, you have to make appropriate changes to the dimensions and add annotations. To hide a parametric dimension, you can use the Drawing tab of the AMVISIBLE command.

<Drawing> **<Visibility...>**

Command: **AMVISIBLE**

As in model mode, you can display the dimensions in terms of parameter name, numeric value, or an equation.

<Drawing> **<Parametric Dim Display>** **<Dimensions As Numbers>**

Command: **AMDWGDIMDSP**

To move a dimension within a drawing view, you can manipulate its grip points. To move a dimension from one view to another, you can use the AMMOVEDIM command.

<Drawing> **<Move Dimension>**

Command: **AMMOVEDIM**

There are two types of dimensions: parametric dimensions and reference dimensions. The parametric dimensions are dimensions that you use to create the parametric solid part. The reference dimensions are dimensions associated with the parametric solid part. You can modify the parametric dimensions to change the solid part. The reference dimensions only follow the change; they cannot drive the change. To create a reference dimension, you can use the AMREFDIM command.

<Drawing> **<Add Ref Dimension>**

Command: **AMREFDIM**

To align the dimension lines of one or more dimensions, you can use the AMDIMALIGN command.

<Drawing> **<Edit Dimensions>** **<Align>**

Command: **AMDIMALIGN**

To join one or more continue dimensions to create a single reference dimension, you can use the AMDIMJOIN command.

<Drawing> **<Edit Dimensions>** **<Join>**

Command: **AMDIMJOIN**

To replace a dimension with two continue dimensions, you can use the AMDIMINSERT command.

<Drawing> **<Edit Dimensions>** **<Insert>**

Command: **AMDIMINSERT**

To hide a portion of the extension lines or dimension line of a dimension, you can use the AMDIMBREAK command.

<Drawing> **<Edit Dimensions>** **<Break>**

Command: **AMDIMBREAK**

Symbol Standards

In addition to text, hole notes, and dimensions, you can add surface texture symbols, welding symbols, and geometric tolerance symbols to a drawing. Before using these symbols, you have to use the AMSYMSTD command. By using the AMSYMSTD command, you can set the properties in accordance with your drafting requirements. Select the Symbol Standards... item of the Drawing pull-down menu to use the AMSYMSTD command. (See Figure 5.96.)

<Drawing> **<Symbol Standards...>**

Command: **AMSYMSTD**

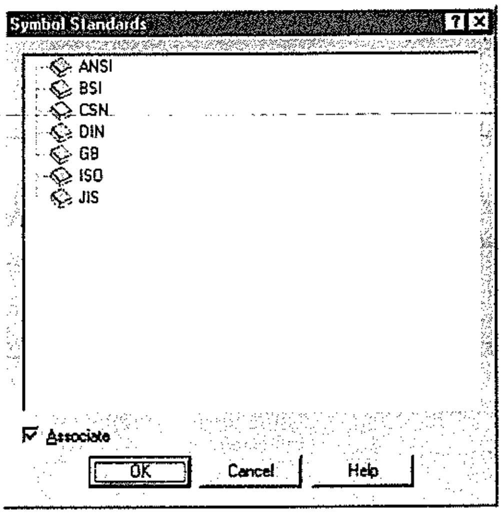


Figure 5.96 Symbol Standards dialog box

In the Symbol Standards dialog box, there are seven standards: ANSI, BSI, CSN, DIN, GB, ISO, and JIS. Select ANSI and double-click to expand the graphic tree. (See Figure 5.97.)

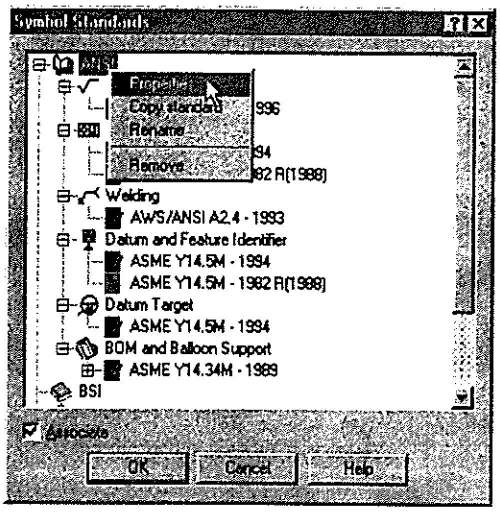


Figure 5.97 ANSI standards

Select ANSI and press the right mouse button. Then select the Properties item. (See Figure 5.98, the Standard Properties for ANSI dialog box.) Here you can set the attributes that are common to all symbols: overall scale, text style, and leader properties.

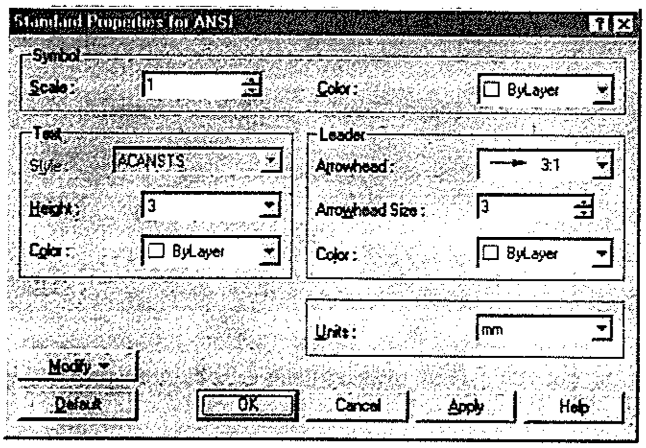


Figure 5.98 Standard Properties for ANSI dialog box

To set the preferences for a standard, select the standard, press the right mouse button, and select Properties.

Welding Symbols

Open the drawing Tarm.dwg.

<File> **<Open...>**

File name: **Tarm.dwg**

Set drawing preferences, enter drawing mode, add a layer called Title, and insert an engineering title block. Then construct a top view and a front view of a scene with no tweaking or explosion, with 0.2 scale, as shown in Figure 5.99.

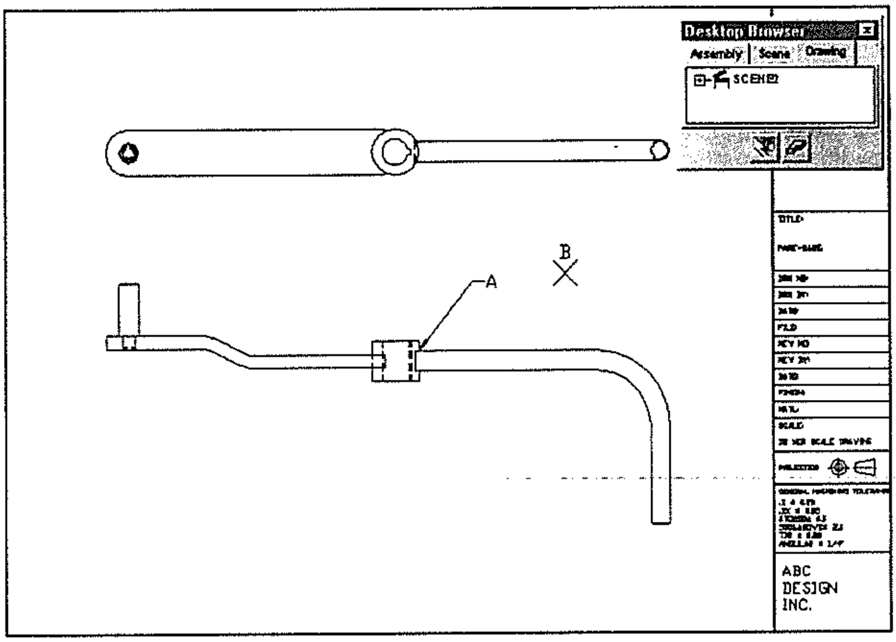


Figure 5.99 Top view and front view constructed

Use the AMSYMSTD command to set the welding symbol standards. (See Figure 5.100.)

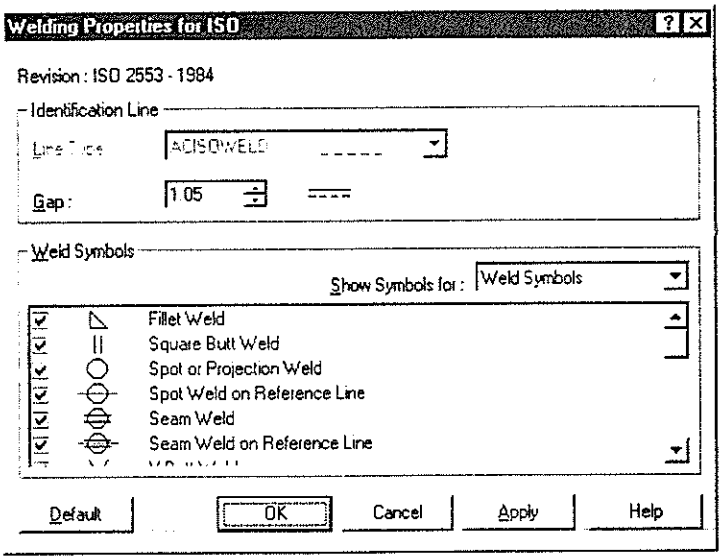


Figure 5.100 Welding Properties for ISO dialog box

Now use the AMWELDSYM command to add a welding symbol to your drawing. In Figure 5.101, the Weld Symbol dialog box, there are four tabs: General, Arrow Side, Other Side, and Leader. Select Other Side and then clear the symbol. After that, select Arrow Side and then select the fillet weld symbol. Then select the [OK] button. A welding symbol is attached. (See Figure 5.102.)

<Drawing> **<Annotate>** **<Welding...>**

Command: **AMWELDSYM**
Select object to attach: [**Select A (Figure 5.99).**]
Start Point: **END** of [**Select A (Figure 5.99).**]
Next Point: [**Select B (Figure 5.99).**]
Next Point <Symbol>: [**Enter**]
Next Point <Symbol>: [**Enter**]

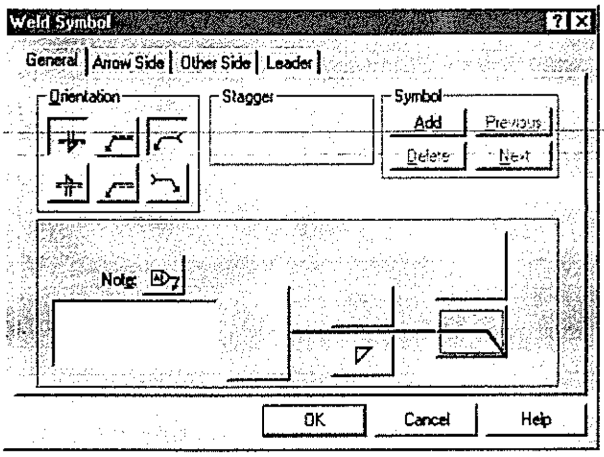


Figure 5.101 Weld Symbol dialog box

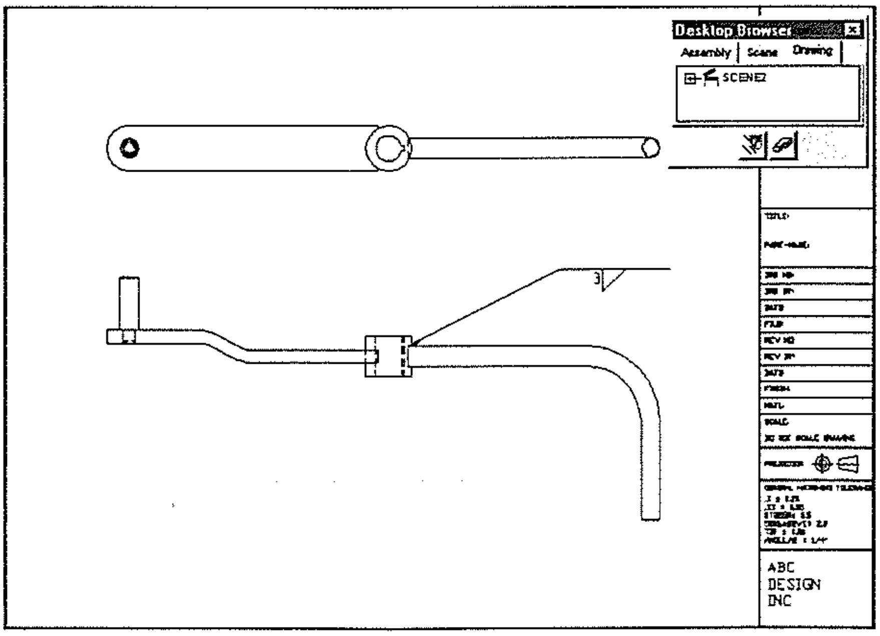


Figure 5.102 Welding symbol inserted

If you want to modify an attached symbol, you can use the AMEDIT command.

<Drawing> **<Annotate>** **<Edit Symbols...>**

The welding symbol is complete. Save your drawing.

<File> **<Save>**

Surface Texture Symbols

Open the drawing Intersm.dwg.

<File> **<Open...>**

File name: **Intersm.dwg**

Set drawing preferences, enter drawing mode, add a new layer called Title, and insert an engineering title block. Then construct a base view and two orthographic views as shown in Figure 5.103.

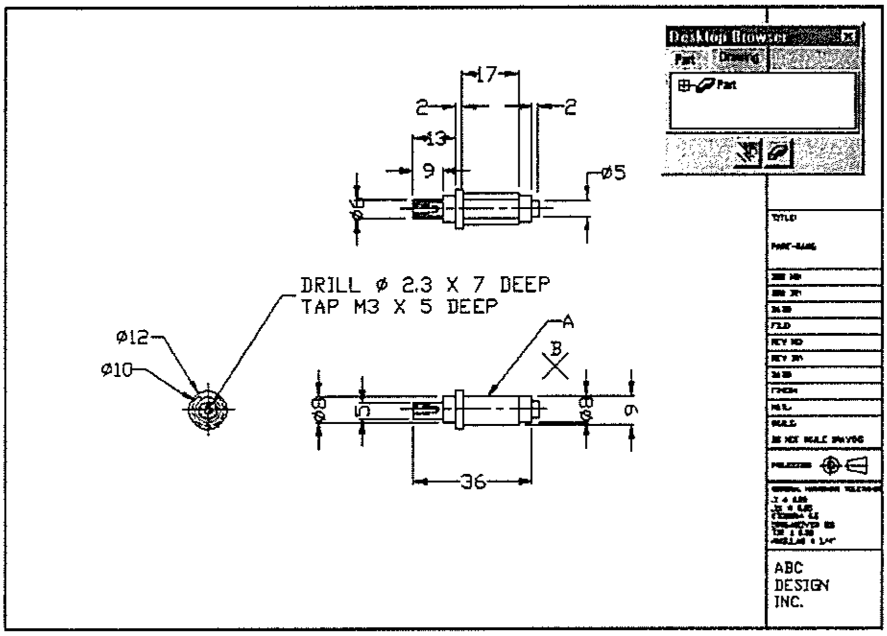


Figure 5.103 Base view and two orthographic views

Use the AMSYMSTD command to set the surface texture symbol standards. (See Figure 5.104.)

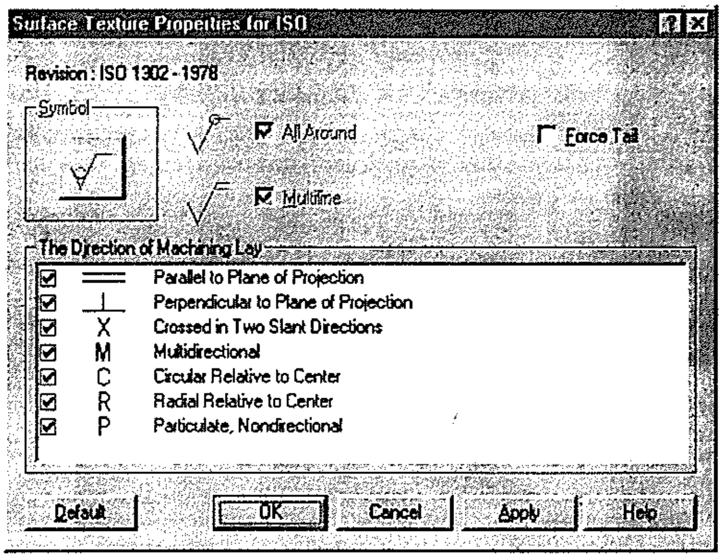


Figure 5.104 Surface Texture Properties for ISO dialog box

Now use the AMSURFSYM command to add a surface texture symbol to your drawing. (See Figure 5.105.)

<Drawing> **<Annotate>** **<Surface Texture...>**

Command: **AMSURFSYM**
Select object to attach: [**Select A (Figure 5.103).**]
Start Point: **MID** of [**Select A (Figure 5.103).**]
Next Point <Symbol>: [**Select B (Figure 5.103).**]
Next Point <Symbol>: [**Enter**]

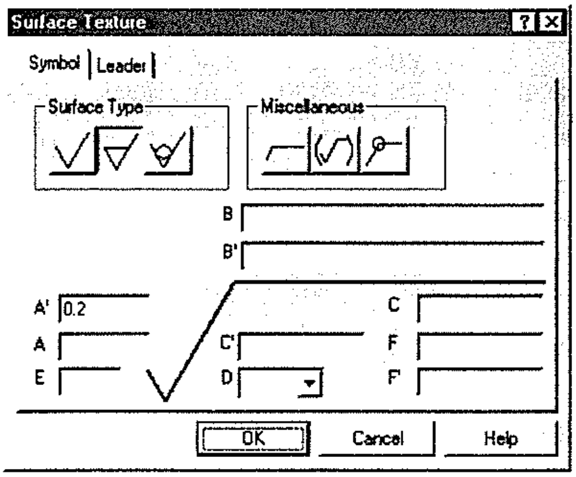


Figure 5.105 Surface Texture dialog box

Set the surface texture symbol as shown in Figure 5.105. Then select the [OK] button. (See Figure 5.106.)

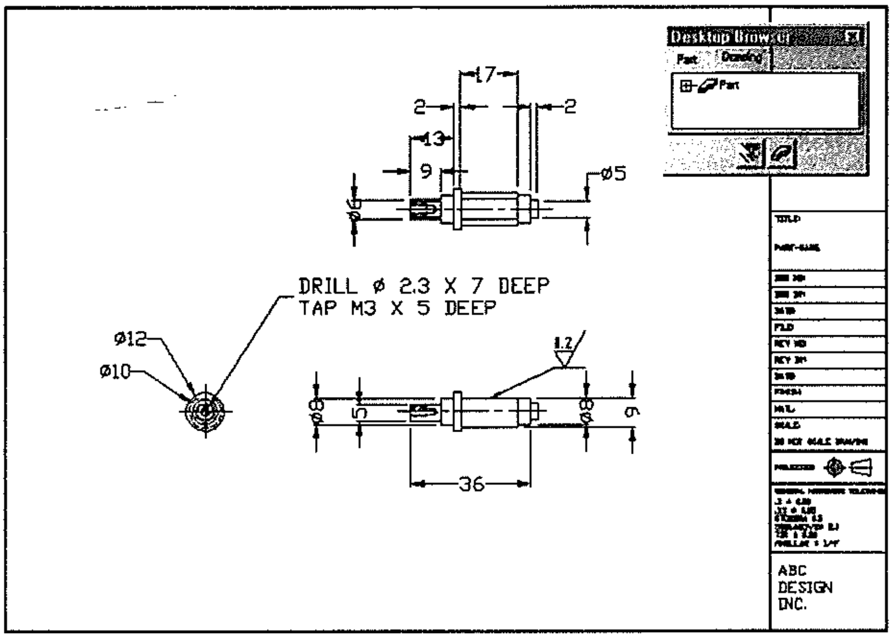


Figure 5.106 Surface texture symbol attached

To modify a surface texture symbol, you can use the AMEDIT command.

<Drawing> **<Annotate>** **<Edit Symbols...>**

The surface texture symbol is complete. Save your drawing.

<File> **<Save>**

Lateral and Geometric Tolerance

You will continue to work on the drawing Arm_l.dwg. (See Figure 5.107.)

<File> **<Open...>**

File name: **Arm_l.dwg**

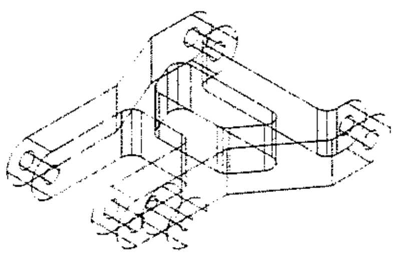


Figure 5.107 Lower suspension arm of a scale model car

Set drawing preferences, enter drawing mode, add a new layer Title, and insert a title block. Then construct a base view and two orthographic views. After that, add dimensions and center lines. (See Figure 5.108.)

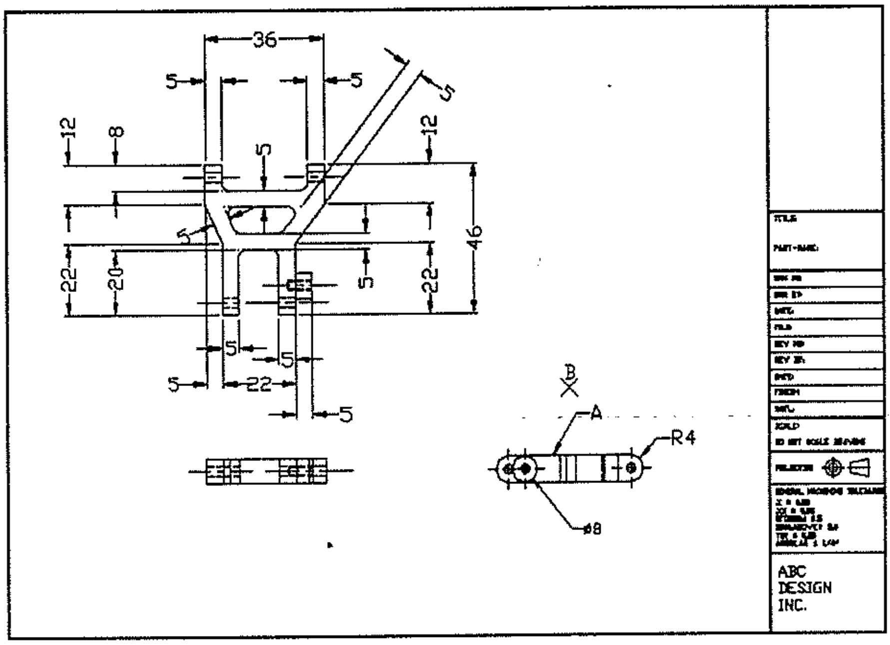


Figure 5.108 Base view and two orthographic views constructed

Lateral tolerance is the allowable deviation of a dimension from its theoretically perfect size. To set lateral tolerance, you can set up a dimension style by using the DDIM command.

<Drawing> **<Annotation Styles>** **<Dimension...>**

Command: **DDIM**

To govern the geometric aspects of a 3D solid part, you can apply geometric tolerance. Set the related preferences by using the AMSYMSTD command. (See Figures 5.109 and 5.110.)

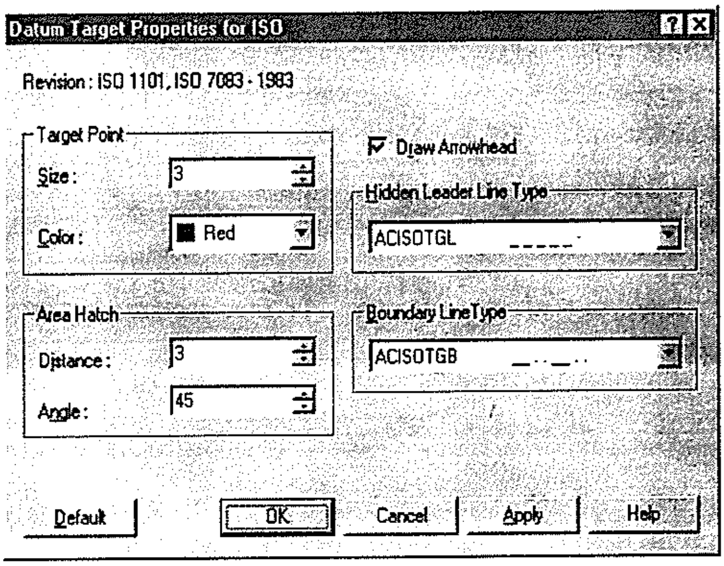


Figure 5.109 Datum Target Properties for ISO dialog box

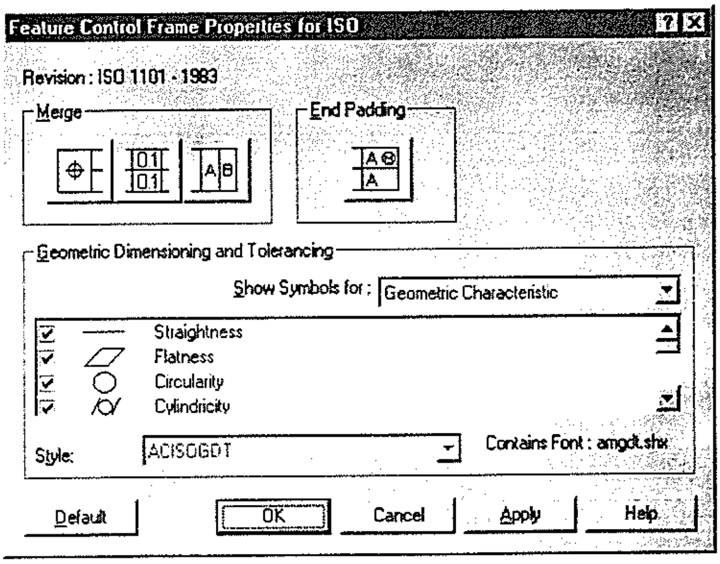


Figure 5.110 Feature Control Frame Properties for ISO dialog box

Use the AMDATUMID command to establish a datum target. (See Figure 5.111, the Datum Identifier dialog box.) Type A and then select the [OK] button. (See Figure 5.112.)

<Drawing> **<Annotate>** **<Datum Identifier...>**

Command: **AMDATUMID**
Select object to attach: [**Select A (Figure 5.108)**.]
Start Point: **NEA** to [**Select A (Figure 5.108)**.]
Next Point <Symbol>: [**Enter**]

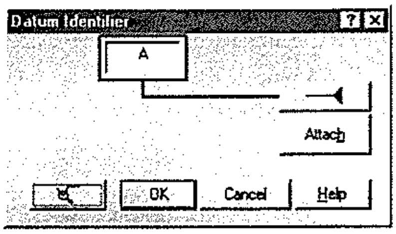


Figure 5.111 Datum Identifier dialog box

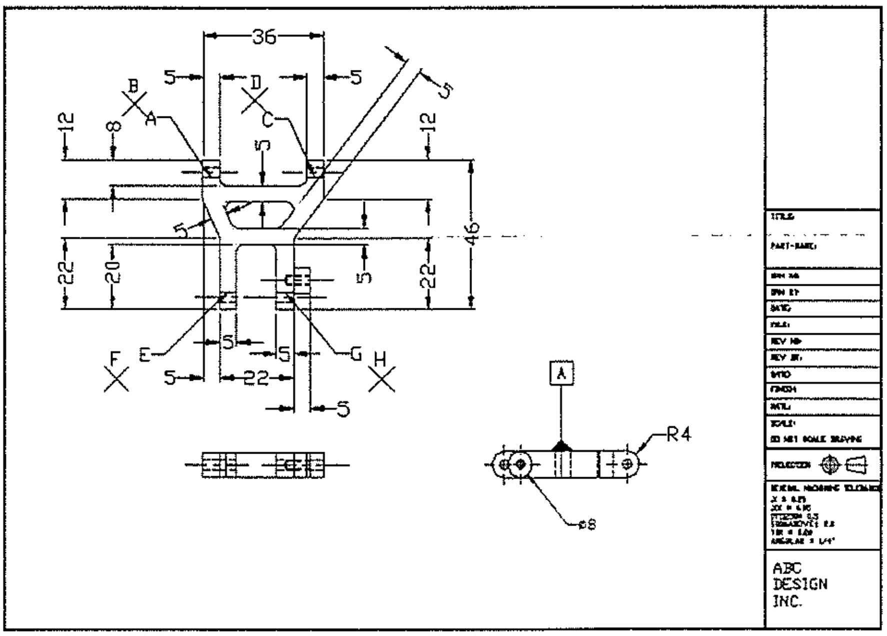


Figure 5.112 Datum identifier attached

Suppose that the component shown in the drawing is produced by casting. Because surfaces that are produced by casting or forging may be subject to warping or twisting, surface A (Figure 5.112) needs to be defined by a number of datum targets and datum target symbols. To add a datum target, use the AMDATUMTGT command. (See Figure 5.113.)

<Drawing> **<Annotate>** **<Datum Target...>**

Command: **AMDATUMTGT**
Select object to attach: [**Select A (Figure 5.112).**]

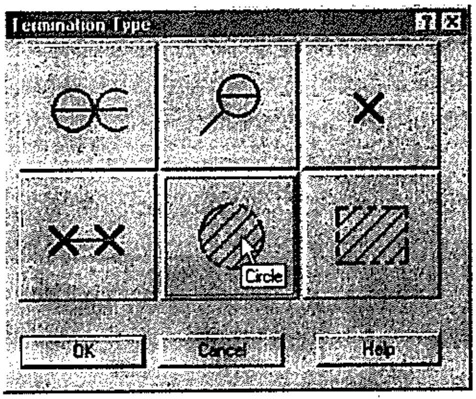


Figure 5.113 Termination Type dialog box

In the Termination Type dialog box, select the [Circle] button. Now select a location for the target, specify a radius for the target, and select a location for the symbol. (See Figure 5.114.) Now the Datum Target dialog box appears. Set the Dimension to %%C2 ("%%C" means diameter), and make the datum name A1.

Center point: **MID** of [**Select A (Figure 5.112).**]
Diameter/<Radius> **2**
Next Point: [**Select B (Figure 5.112).**]
Next Point: [**Enter**]

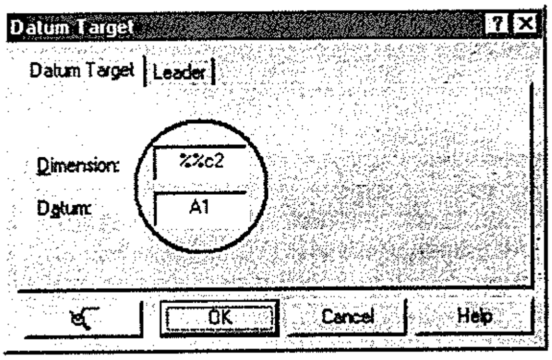


Figure 5.114 Datum Target dialog box

Now a datum target is attached. Repeat the AMDATUMTGT command three more times to set three more datum targets A2, A3, and A4, of diameter 2 units, located at the midpoints of C, E, and G (Figure 5.112), and with symbols placed at D, F, and H (Figure 5.112). (See Figure 5.115.)

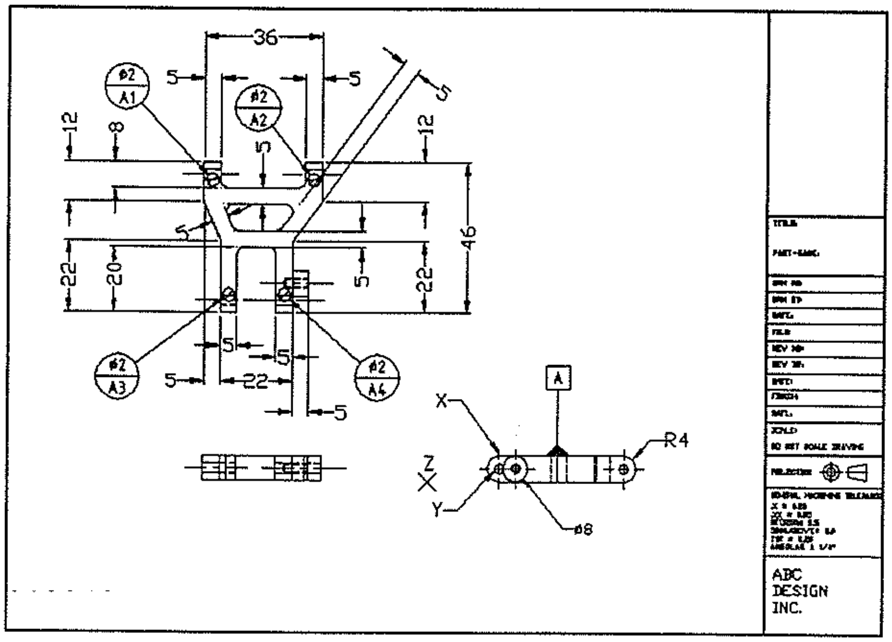


Figure 5.115 Datum targets placed

Together, the datum targets A1, A2, A3, and A4 make up a datum plane A. Now use the AMREFDIM command to add a base dimension. (See Figure 5.116.) Note that a base dimension is placed inside a rectangle. It does not carry any tolerance and it defines the theoretically perfect size.

<Drawing> **<Add Ref Dimension>**

Command: **AMREFDIM**
Select first object: [**Select X (Figure 5.115).**]
Select second object or place dimension: [**Select Y (Figure 5.115).**]
Specify dimension placement: [**Select Z (Figure 5.115).**]
Undo/Hor/Ver/Align/Par/aNgle/Ord/reF/Basic/pLacement point: **B**
Undo/Hor/Ver/Align/Par/aNgle/Ord/reF/Basic/pLacement point: [**Enter**]

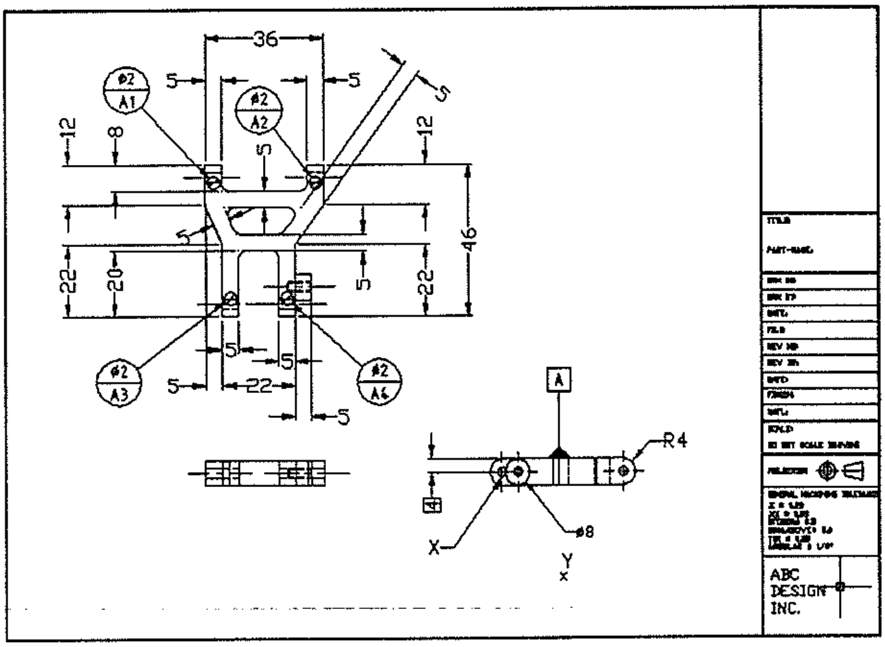


Figure 5.116 Base dimension constructed

Now use the AMFCFRAME command to add a position tolerance. (See Figure 5.117, the Feature Control Frame dialog box.) Set the tolerance value to diameter 0.2 unit and datum 1 to A. Then select the [Geometric Symbol Palette] button. (See Figure 5.118.) Select Position tolerance. Then select the [OK] button. (See Figure 5.119.)

<Drawing> **<Annotate>** **<Feature Control Frame...>**

Command: **AMFCFRAME**
Select object to attach: [**Select X (Figure 5.116).**]
Start Point: **NEA** to [**Select X (Figure 5.116).**]
Next Point <Symbol>: [**Select Y (Figure 5.116).**]
Next Point <Symbol>: [**Enter**]

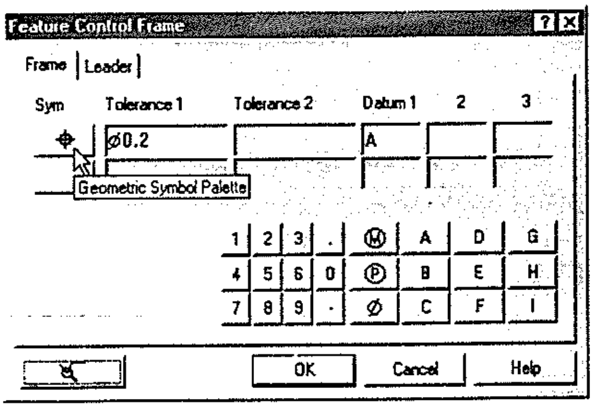


Figure 5.117 Feature Control Frame dialog box

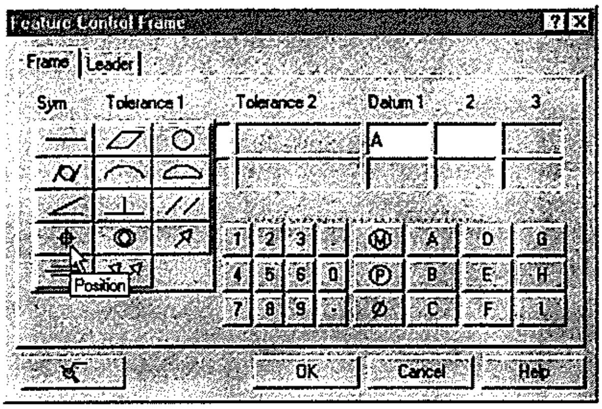


Figure 5.118 Position tolerance selected

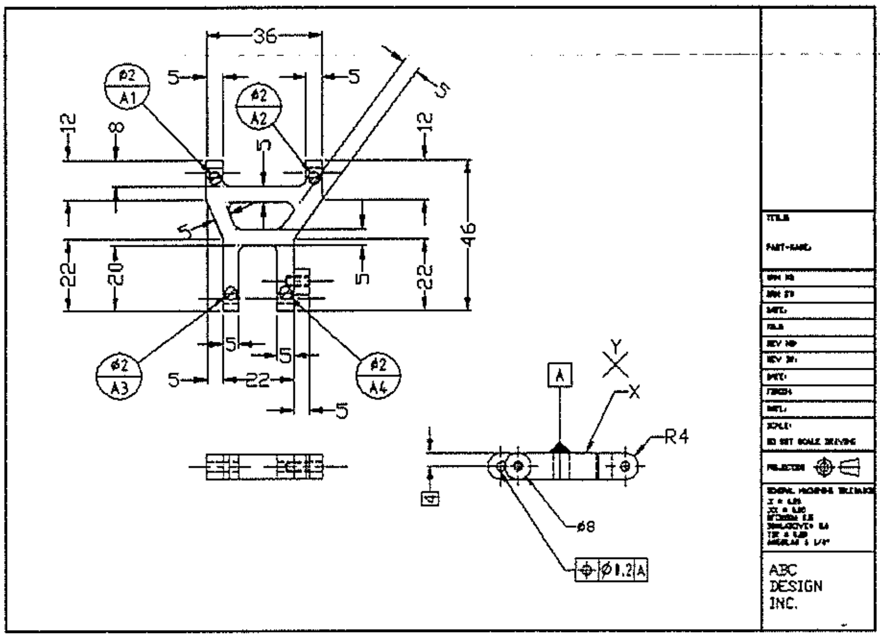


Figure 5.119 Position tolerance constructed

A position tolerance is placed. It governs the position of the specified hole in relation to the datum plane A that is defined by datum targets A1, A2, A3, and A4.

Turn on ORTHO mode. Then repeat the AMFCFRAME command to place a parallelism tolerance. (See Figure 5.120.)

Command: **ORTHO**

<Drawing> **<Annotate>** **<Feature Control Frame...>**

Command: **AMFCFRAME**
Select object to attach: **[Select X (Figure 5.119).]**
Start Point: **NEA** to **[Select X (Figure 5.119).]**
Next Point <Symbol>: **[Select Y (Figure 5.119).]**
Next Point <Symbol>: **[Enter]**

[Feature Control Frame
Geometric Symbol Palette
Parallelism
OK]

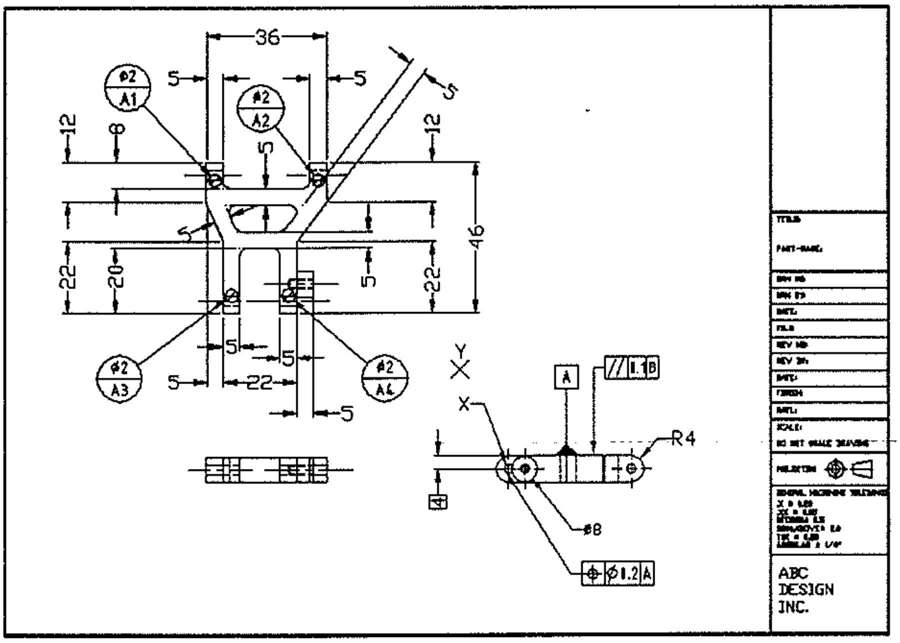


Figure 5.120 Parallelism tolerance added

Now use the AMFEATID command to add a feature identifier. (See Figures 5.121 and 5.122.)

<Drawing> **<Annotate>** **<Feature Identifier...>**

Command: **AMFEATID**
Select object to attach: [**Select X (Figure 5.120).**]
Start Point: [**Select X (Figure 5.120).**]
Next Point <Symbol>: [**Select Y (Figure 5.120).**]
Next Point <Symbol>: [**Enter**]

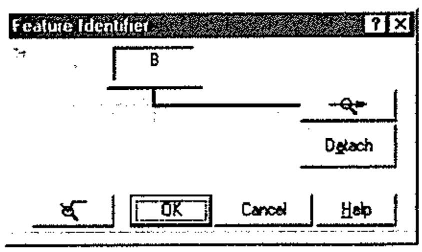


Figure 5.121 Feature Identifier dialog box

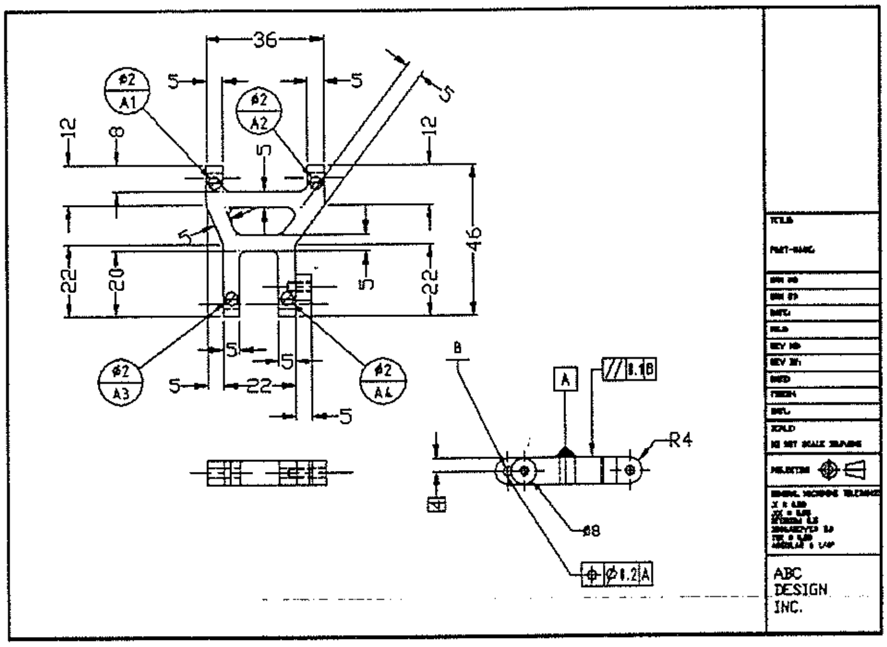


Figure 5.122 Feature identifier attached

To modify a geometric tolerance symbol, use the AMEDIT command.

<Drawing> <Annotate> <Edit Symbols...>

The geometric tolerance symbols are complete. Save your drawing.

<File> <Save>

Attributes and Bill of Materials

Open the drawing 4bar.dwg that you saved earlier in this chapter. You will add a BOM (bill of materials) and appropriate balloons to the drawing.

<File> <Open...>

File name: **Remote.dwg**

Use the AMSYMSTD command to set standards for constructing the BOM. Expand the standards list. (See Figure 5.123, the Symbol Standards dialog box.)

To set the BOM standard, double-click on the Parts List (Figure 5.123). (See Figure 5.124, the Parts List Properties for ISO dialog box.) Set the properties for ISO and then select the [OK] button.

To set the Balloon standard, double-click on the Balloon item (Figure 5.123). (See Figure 5.125, the Balloon Properties for ISO dialog box.) After setting the properties, select the [OK] button.

<Drawing> **<Symbol Standards...>**

Command: **AMSYMSTD**

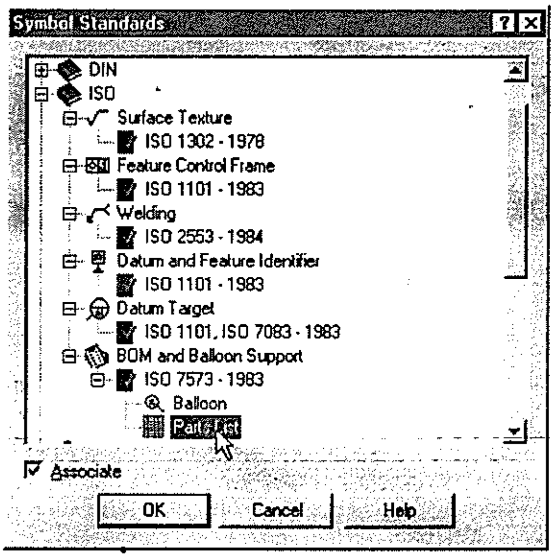


Figure 5.123 Symbol Standards dialog box

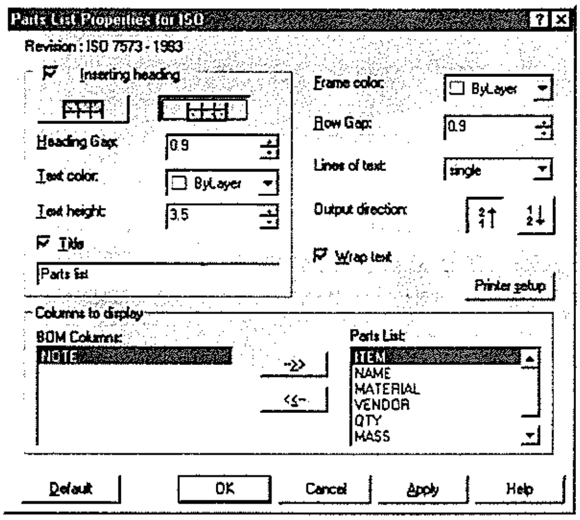


Figure 5.124 Parts List Properties for ISO dialog box

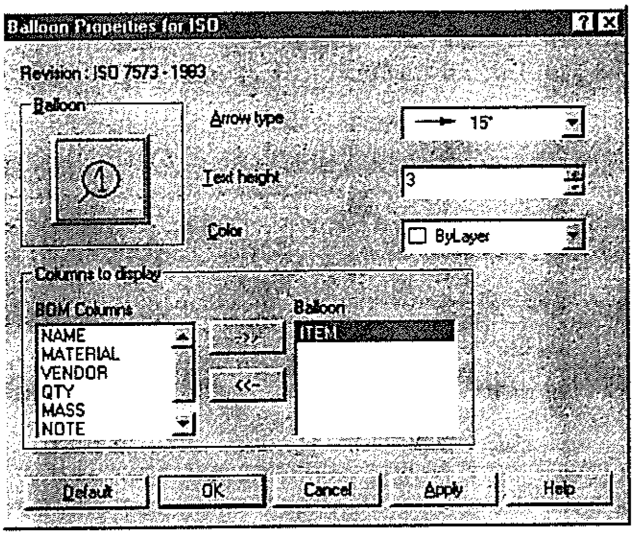


Figure 5.125 Balloon Properties for ISO dialog box

Now use the AMBOM command to edit the BOM database. (See Figure 5.126.)

<Drawing> **<Balloons and BOMs>** **<Edit BOM Database...>**

Command: **AMBOM**

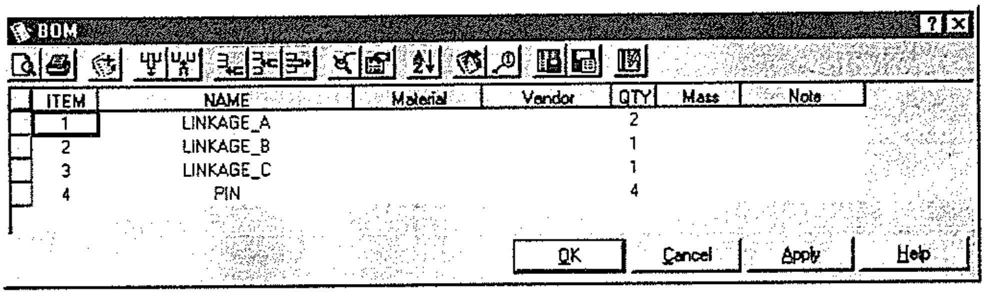


Figure 5.126 BOM dialog box

The BOM dialog box shows a database of attributes assigned to each solid part of the drawing. The BOM and balloons are referenced to this database. Use the AMPARTLIST command to add a BOM parts list to the drawing. (See Figure 5.127.)

<Drawing> **<Balloons and BOMs>** **<Place Part List...>**

Command: **AMPARTLIST**
Select type of Parts List: Parts/Range/View/<All>: **ALL**
Specify location or ENTER to Right direction: **[Select A (Figure 5.127).]**

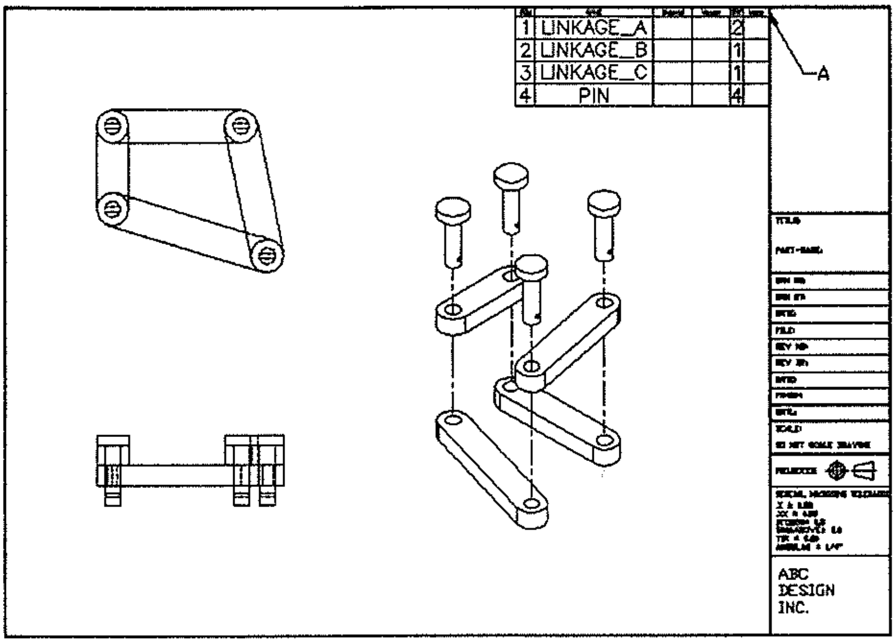


Figure 5.127 Parts list inserted

To edit a parts list, you can use the AMEDIT command.

<Drawing> **<Balloons and BOMs>** **<Edit Part List...>**

Command: **AMEDIT**
Select object: **[Select the parts list.]**

After you select the parts list, the BOM dialog box (Figure 5.126) appears. You can modify the attributes in the database.

Now use the AMBALLOON command to place four balloons. (See Figure 5.128.)

<Drawing> **<Balloons and BOMs>** **<Place Balloon...>**

Command: **AMBALLOON**

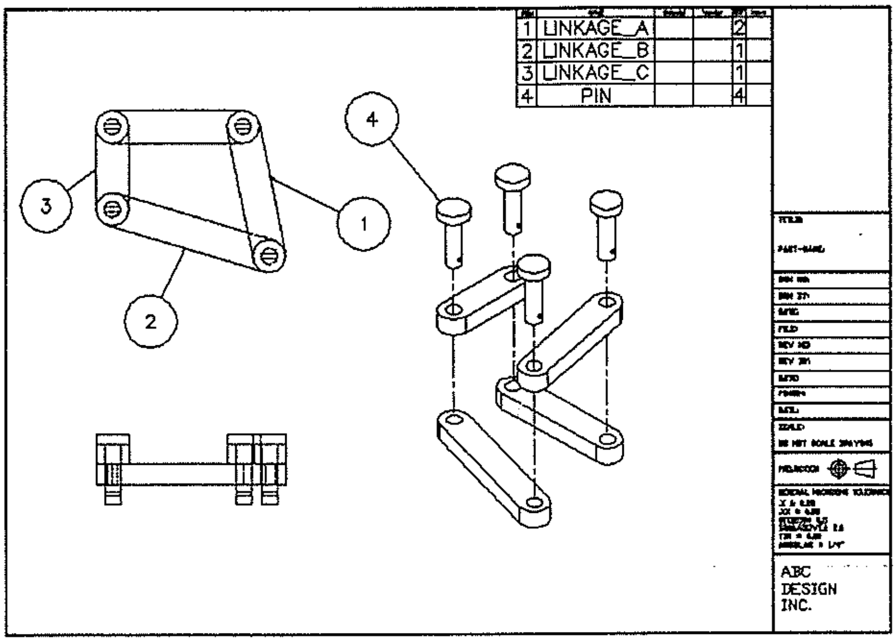


Figure 5.128 Balloons added

To edit the balloons, you can use the AMEDIT command.

<Drawing> **<Balloons and BOMs>** **<Edit Balloon...>**

Command: **AMEDIT**

In addition to solid parts of the assembly, you can include block references to the BOM by using the AMPARTREF command.

<Drawing> **<Balloons and BOMs>** **<Place Part Reference>**

Command: **AMPARTREF**

To edit a referenced block associated to the BOM, you can use the AMPARTREFEDIT command.

<Drawing> **<Balloons and BOMs>** **<Edit BOM Attributes...>**

Command: **AMPARTREFEDIT**

The drawing is complete. Save your file.

<File> **<Save>**

5.5 Key Points and Exercises

There are two working modes in a drawing file: drawing mode and model mode. You construct 3D objects in model mode and construct engineering drawings in drawing mode.

In drawing mode, you perform two major tasks: insert an engineering drawing title block and construct drawing views. The drawing views are projected from the data set constructed in model mode. There are four kinds of data set: active part, scene, select, and group. The active part is the active solid part of the drawing. The scene is the assembly scene. In addition to choosing the active part or an assembly scene, you can determine the data set by selecting or by specifying the entity group name. For the drawing of a NURBS surface model, you can use either the Select option or the Group option in specifying the data set.

You can hide dimensions and move dimensions within a drawing view and across drawing views. In addition to parametric dimensions, you can add reference dimensions to depict the 3D objects. In the assembly drawing, you can set up a bill of materials and add balloons.

The parametric dimensions in an engineering drawing associate bidirectionally to the solid part. Change in the parametric dimensions in drawing mode modifies the solid part in model mode. Editing the solid part in model mode causes an automatic update of the parametric dimensions, as well as the reference dimensions, in drawing mode. As for the reference dimensions, they follow the change but cannot drive the change.

Exercise 5.1

Outline the steps to construct an associative engineering drawing from a 3D parametric solid, an assembly of solid parts, and a set of NURBS surfaces.

Exercise 5.2

Open the drawings constructed in Exercises 3.3 through 3.28 one by one. Then construct associative engineering drawings. In each engineering drawing, include an engineering title block and add appropriate center lines and dimensions.

Exercise 5.3

Open the drawings constructed in Exercises 4.4 through 4.9 one by one. Then construct associative engineering drawings. In each engineering drawing, include an engineering title block and add a bill of materials and appropriate balloons.

Appendix

Quick Command Reference

Brief descriptions of Mechanical Desktop commands, together with the appropriate menus and toolbars, are given here for your quick reference. They are divided into five categories: surface modeling, parametric solid modeling, assembly modeling, associative drafting, and visualization.

A.1 Surface Modeling

AM2SF	converts an object into a set of surfaces.
AMABOUT	displays version and copyright information.
AMAUGMENT	constructs augmented lines from surface edges, internal trim lines, and display lines.
AMBLEND	constructs a surface connecting two, three, or four wires or surfaces.
AMBREAK	breaks one surface into two surfaces along a specified direction.
AMCHECKFIT	measures the distance between objects.
AMCORNER	constructs a blended fillet corner surface from three intersecting fillet surfaces.
AMDIRECTION	displays the direction of an object and reverses its direction.
AMDISPSF	controls the way surfaces are displayed.
AMDIST	measures the minimum 3D distance between two sets of objects.
AMEDGE	untrims, copies, and extracts edges of surfaces and shows edge nodes of surfaces and faces.
AMEDITAUG	resizes, rotates, copies, and corrects vectors on augmented lines and constructs augmented lines from lines and polylines.
AMEDITSF	changes the density of surface grip points, the grip span, and the direction of the normal and truncates surfaces.
AMEXTRUDESF	constructs a surface by extruding a line, arc, circle, ellipse, spline, or polyline.
AMFILLET3D	constructs a smoothly fitted arc of constant radius between wires.
AMFILLETSF	constructs a fillet surface along the intersection of two surfaces.
AMFITSPLINE	smoothes wire objects by fitting them with a smooth NURBS spline.
AMFLOW	constructs flow wires in the U and V directions.
AMINTERSF	intersects two surfaces and constructs a 3D polyline at the intersection.
AMJOIN3D	joins wires to form one 3D polyline, spline, or augmented line.

AMJOINSF	joins two or more surfaces at their untrimmed base edges into one continuous surface.
AMLENGTHEN	extends or shortens base surface edges by a percentage or distance.
AMLOFTU	constructs a surface through a set of wires.
AMLOFTUV	constructs a surface through two sets of wires.
AMOFFSET3D	constructs a copy of a 3D polyline that is set off from the original.
AMOFFSETSF	constructs a surface offset from a selected surface.
AMPARTLINE	constructs a 3D polyline on the profile of the surface in the current view.
AMPLANE	constructs one or more planar surfaces.
AMPREFS	manages part, assembly, surface, drawing, and desktop preferences from a single dialog box.
AMPRIMSF	constructs a cone, cylinder, torus, or sphere.
AMPROJECT	projects a wire onto a surface.
AMREFINE3D	changes the point density of lines and 3D polylines.
AMREFINESF	redefines the selected surface with a less complex approximation.
AMREVOLVESF	constructs a surface by rotating a path wire around a selected axis.
AMRULE	constructs a straight element surface between two wires.
AMSCALE	increases or decreases one or all surface axes by a specific scale factor.
AMSECTION	constructs section cuts through one or more surfaces.
AMSOLCUT	cuts an AutoCAD solid model with a Mechanical Desktop surface.
AMSURFPROP	calculates the mass properties of surfaces.
AMSWEEPSF	constructs a surface by sweeping cross sections along one or two rails.
AMTUBE	constructs a tubular surface around a selected wire that becomes the axis of the tube.
AMUNSPLINE	changes splines to fit points or polylines.
AMVISIBLE	controls the visibility of parts, assemblies, scenes, drawings, and geometric objects.

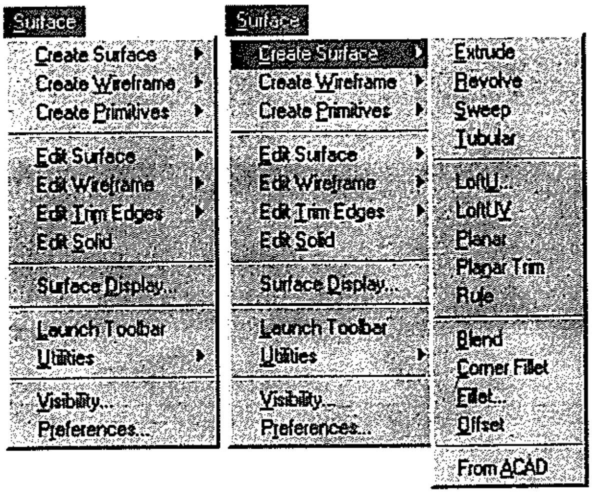


Figure A.1 Surface pull-down menu and Create Surface cascading menu

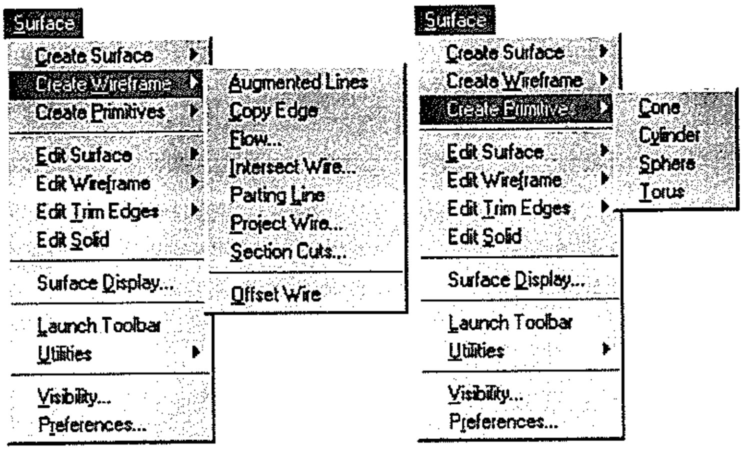


Figure A.2 Create Wireframe and Create Primitives cascading menus

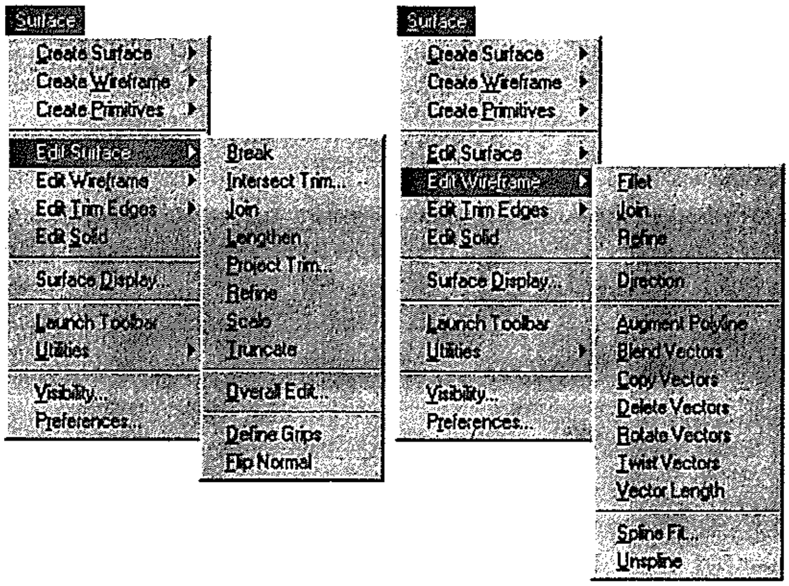


Figure A.3 Edit Surface and Edit Wireframe cascading menus

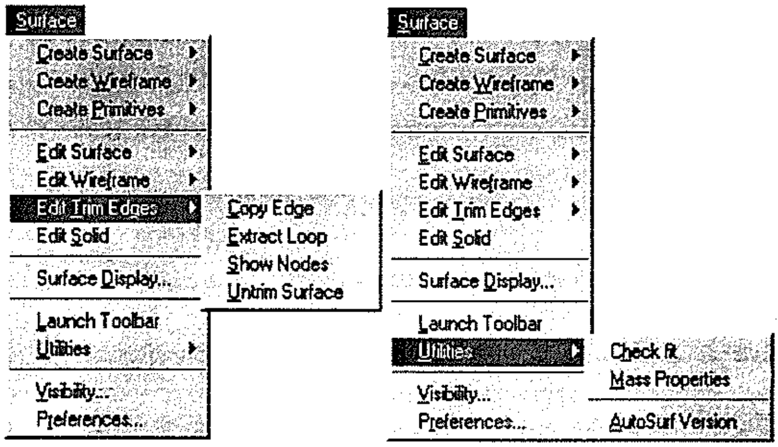


Figure A.4 Edit Trim Edges and Utilities cascading menus

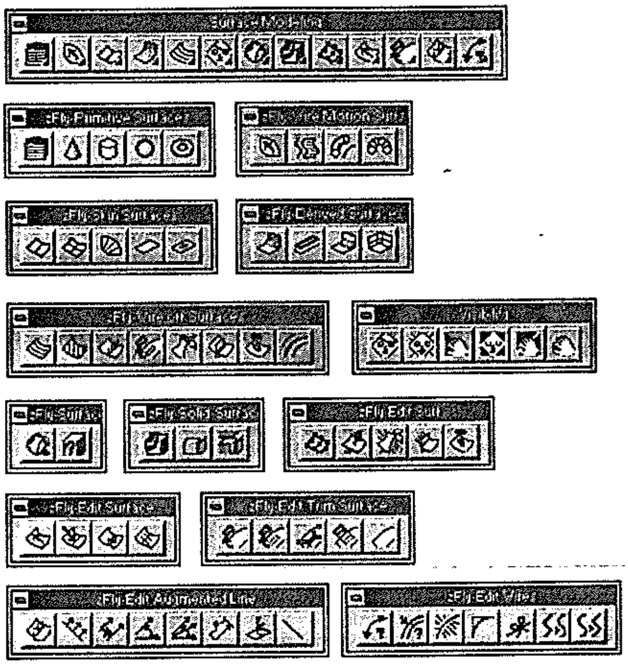


Figure A.5 Surface Modeling toolbars

A.2 Parametric Solid Modeling

AMACTIVATE	selects and activates a solid part, assembly, or scene in the Part/Assembly environment.
AMADDCON	adds parametric constraints to the selected profile, path, cutting line, or split line.
AMARRAY	constructs rectangular and polar arrays of sketched and placed solid features, and combines part features.
AMASSIGN	assigns a value to a part attribute or adds a new user-defined attribute to a part definition.
AMBROWSER	provides a visual structure for a part, assembly, scene, or drawing with icons that represent a part, feature, constraint, or attribute as it is added to a model.
AMCHAMFER	constructs a chamfer on a selected edge or edges of the active part.
AMCOPYFEAT	copies sketched and hole features or feature sets within the same part or from one part to another.
AMCOPYSKETCH	selects single or multiple sketches to copy and specifies placement of the sketch center.
AMDELCON	removes constraints from a selected sketch.
AMDELFEAT	deletes features from a selected part.
AMDIMDSP	changes the display mode for dimensions.
AMEDITFEAT	displays and modifies dimension values for features of the active part.
AMEXTRUDE	constructs an extrude solid feature.

AMFACEDRAFT	constructs a draft angle on a part face.
AMFILLET	constructs and displays an example of a specified fillet type.
AMFIXPT	fixes a point on a sketch in XYZ space.
AMHOLE	constructs drilled, counterbored, or countersunk holes that can be terminated on planar faces.
AMIDFIN	reads printed circuit board (PCB) data in the Intermediate Data Format (IDF) and converts it to AutoCAD and Mechanical Desktop objects.
AMLISTPART	displays part, feature, and view information.
AMLOFT	lofts a solid from a series of profiles.
AMMAKEBASE	converts an active part to a static base feature.
AMMIRROR	generates a mirror copy of a part in the Single Part and Part/Assembly environments.
AMMODDIM	modifies parametric dimensions in sketches and drawings.
AMNEW	constructs new instances of parts, scenes, and subassemblies.
AMPARDIM	constructs parametric dimensions in the model environment.
AMPARTEDGE	constructs a line on a selected edge to use in a new profile.
AMPARTPROP	lists the mass properties for a selected part.
AMPATH	constructs a constrained sketch used as a path for a sweep feature.
AMPREFS	manages part, assembly, surface, drawing, and desktop preferences from a single dialog box.
AMPROFILE	solves 2D geometry to construct a constrained sketch used as a profile for extrude, revolve, or sweep features.
AMREORDFEAT	changes the order of a feature in the part creation history.
AMREPLAY	redisplays, in sequence, the steps used to construct features and sketches of the selected part.
AMREVOLVE	constructs a revolved solid feature from a selected profile.
AMSHELL	constructs a shell with an assigned wall thickness or multiple wall thicknesses.
AMSHOWACT	highlights the active part or sketch plane.
AMSHOWCON	displays the constraint symbols on a selected sketch.
AMSHOWINST	highlights selected part instances in the Desktop Browser.
AMSKPLN	sets the sketch plane location and XY axis orientation.
AMSPLITLINE	splits a face of a solid part into two.
AMSURFCUT	cuts free-form surface shapes on solid models.
AMSWEEP	constructs a solid feature from a profile moved along a path.
AMUPDATE	regenerates the active part and drawing with new values, dimensions, and constraints.
AMVARS	constructs and manages active part and global design variables and provides table-driven parts by linking to Microsoft Excel spreadsheets.
AMVISIBLE	controls the visibility of parts, assemblies, scenes, drawings, and geometric objects.
AMVRMLOUT	converts selected objects to VRML (Virtual Reality Modeling Language) so they can be viewed on a Web page.

AMWORKAXIS constructs a work axis at the center line of a cylindrical, conical, or toroidal surface.

AMWORKPLN constructs a construction plane on a selected part.

AMWORKPT constructs work points on the active sketch plane.

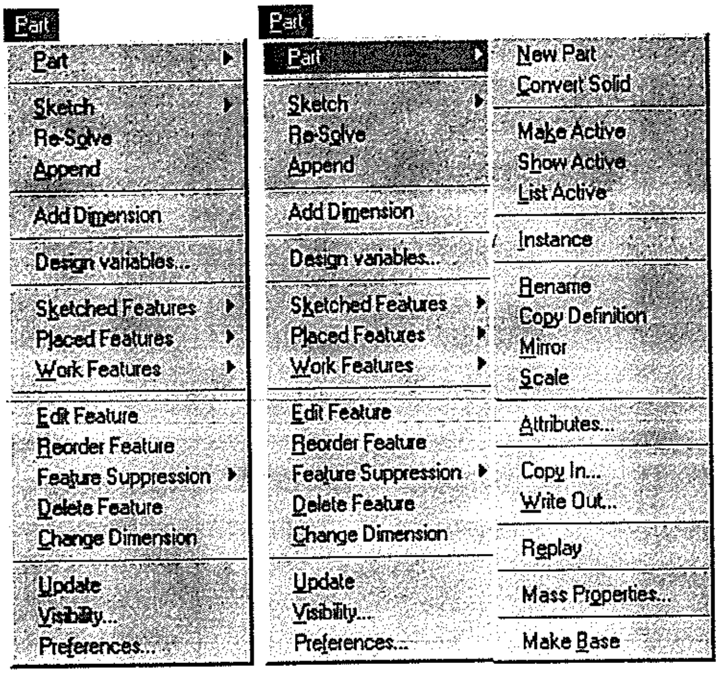


Figure A.6 Part pull-down menu and Part cascading menu

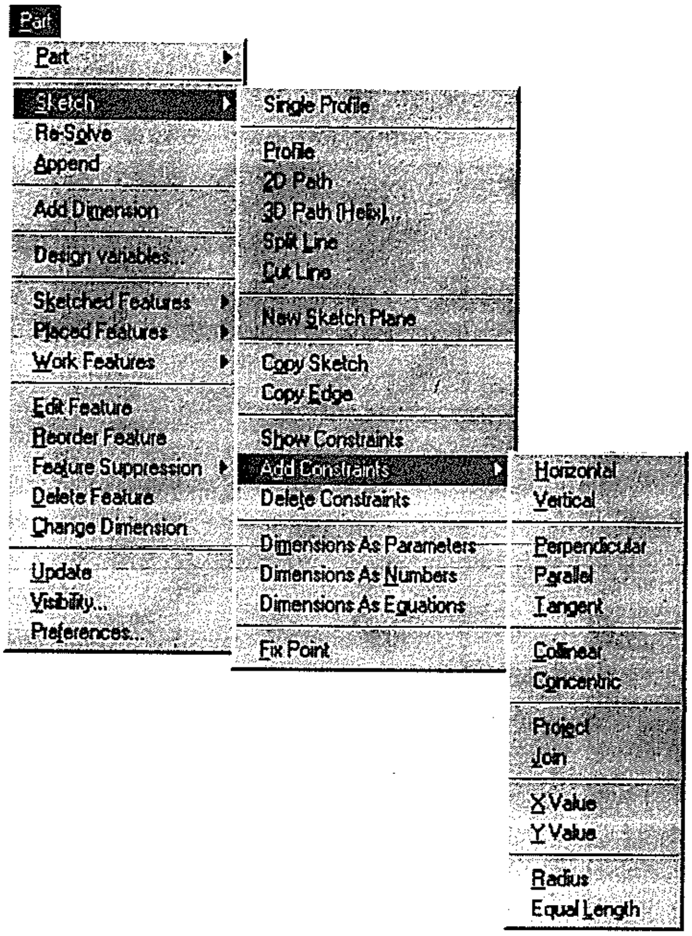


Figure A.7 Add Constraints cascading menu

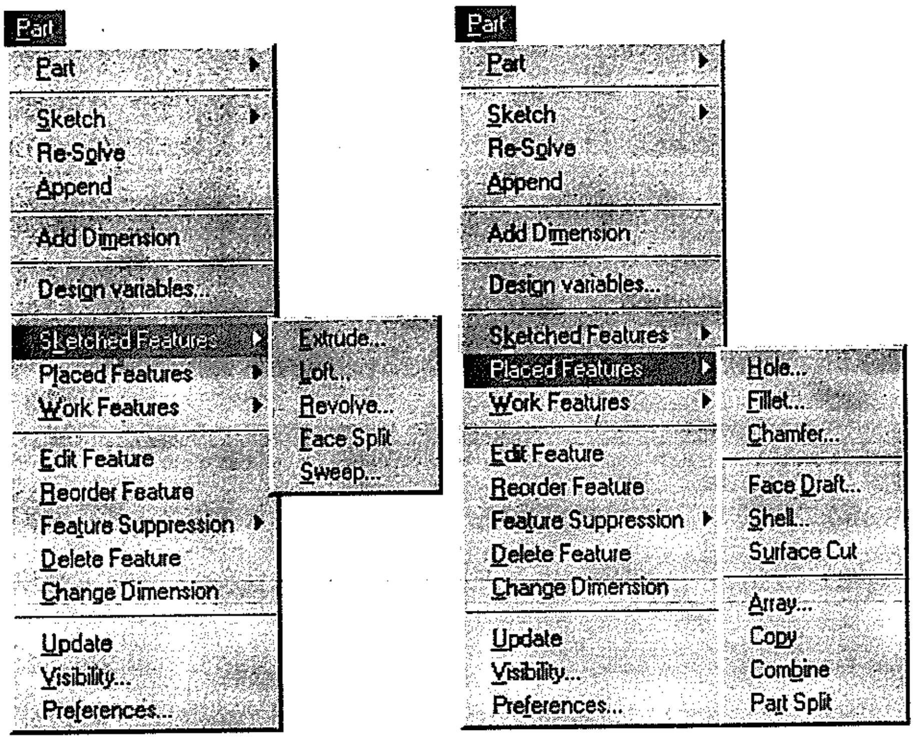


Figure A.8 Sketched Features and Placed Features cascading menus

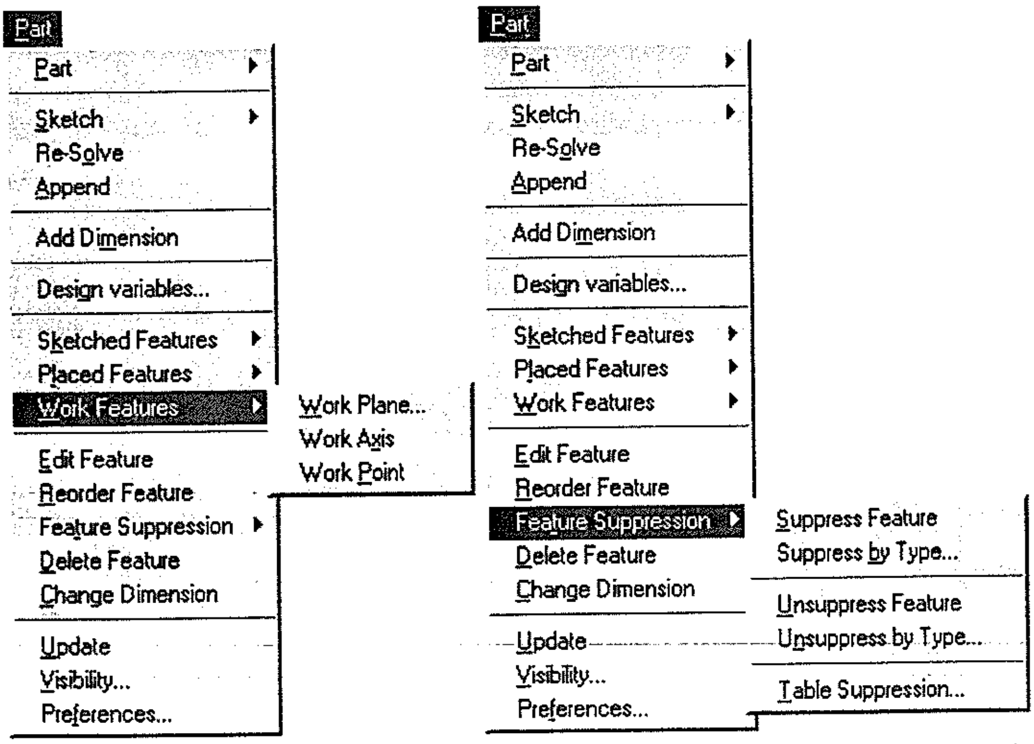


Figure A.9 Work Features and Feature Suppression cascading menus

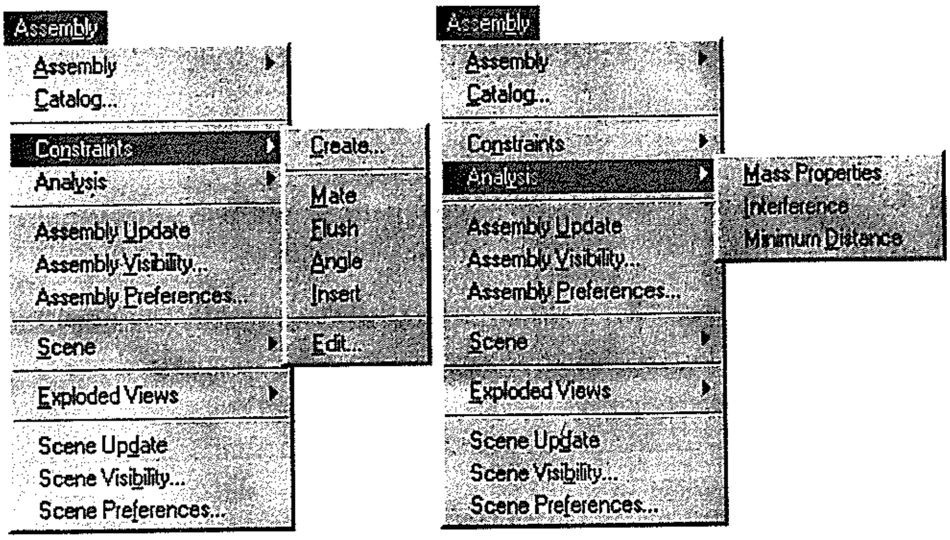


Figure A.12 Constraints and Analysis cascading menus

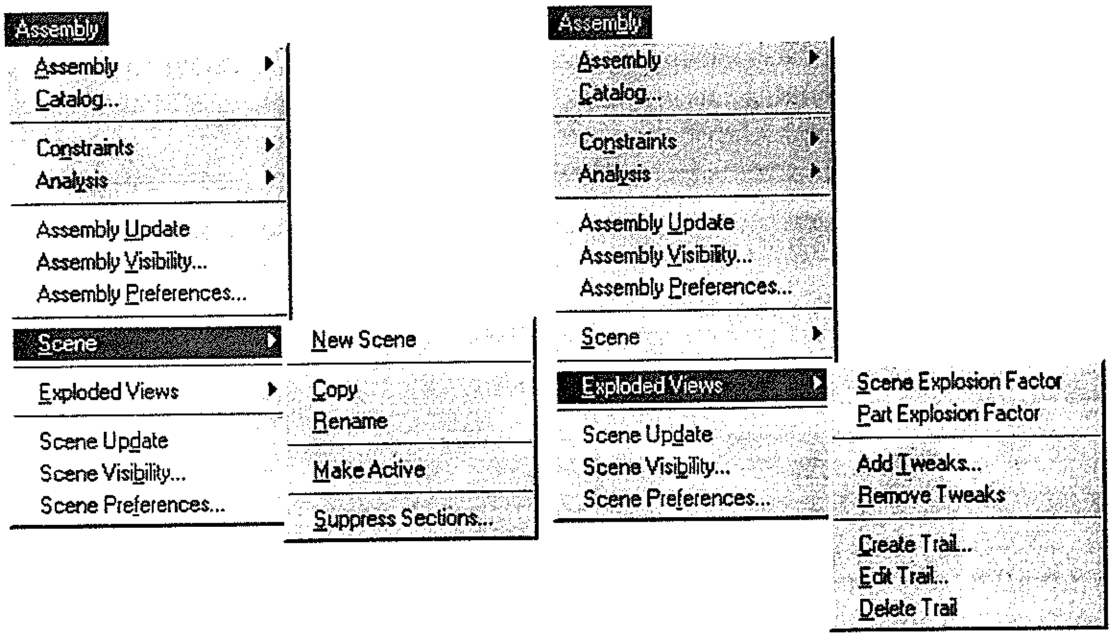


Figure A.13 Scene and Exploded Views cascading menus

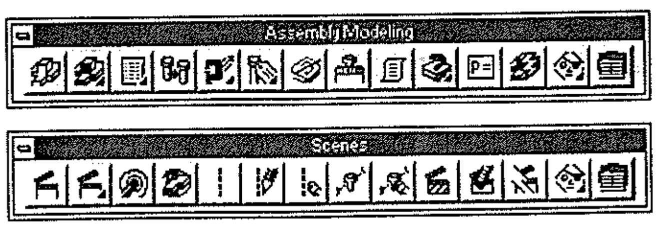


Figure A.14 Assembly Modeling and Scenes toolbars

A.4 Associative Drafting

AMANNOTE	constructs, adds, moves, or deletes drawing annotations.
AMBALLOON	constructs and places balloons in drawing mode.
AMBOM	constructs a bill of materials.
AMBROWSER	provides a visual structure for a part, assembly, scene, or drawing with icons that represent a part, feature, constraint, or attribute as it is added to a model.
AMCENLINE	constructs a parametric center line in drawing mode.
AMCUTLINE	constructs a cutting-line sketch for an offset section view.
AMDATUMID	constructs and edits datums represented in frames containing an alphabetic character.
AMDATUMTGT	constructs and edits datum target symbols.
AMDELTRAIL	deletes a selected trail.
AMDELTWEAKS	deletes all tweaks associated with the selected parts.
AMDELVIEW	deletes specified views and all views dependent on them.
AMDIMALIGN	aligns linear dimensions so they are parallel and angular dimensions so they are concentric.
AMDIMBREAK	constructs a gap (break) between two points on an existing associative dimension or extension line.
AMDIMDSP	changes the display mode for dimensions.
AMDIMFORMAT	edits individual dimensions with a dialog box.
AMDIMINSERT	edits a dimension and inserts a new dimension simultaneously.
AMDIMJOIN	joins two or more linear or angular dimensions.
AMDWGDIMDSP	changes the display of parametric dimensions in drawing mode.
AMDWGOUT	converts Mechanical Desktop data to AutoCAD 2D data.
AMDWGVIEW	constructs base, orthographic, auxiliary, isometric, detail, section, and broken views.
AMEDIT	edits symbols for welding, surface textures, feature control frames, datum targets, and datum and feature identifiers.
AMEDITTRAIL	modifies the explosion path start and end offsets in an assembly scene.
AMEDITVIEW	modifies the scale, text, associativity, and hidden line display of a drawing view.
AMFCFRAME	constructs a feature control frame.
AMFEATID	constructs a feature identifying symbol.
AMHOLENOTE	constructs custom diameter, depth, and angle information for standard hole notes.
AMLISTDWG	lists view information in a text window.
AMMODE	controls whether model or drawing mode is in effect.
AMMOVEDIM	controls locations of dimensions in drawing mode.
AMMOVEVIEW	moves drawing views in drawing mode.
AMPREFS	manages part, assembly, surface, drawing, and desktop preferences from a single dialog box.
AMREFDIM	constructs reference dimensions in drawing mode.

AMSTYLEI	imports existing dimension styles.
AMSURFSYM	adds surface texture finish symbols to drawings.
AMSYMSTD	edits and defines drafting standards for symbols.
AMTEMPLATE	constructs, renames, and edits templates for hole notes.
AMUPDATEDWGVIEW	updates drawing views.
AMVIEW	controls viewing in model space.
AMWELDSYM	constructs a welding symbol.

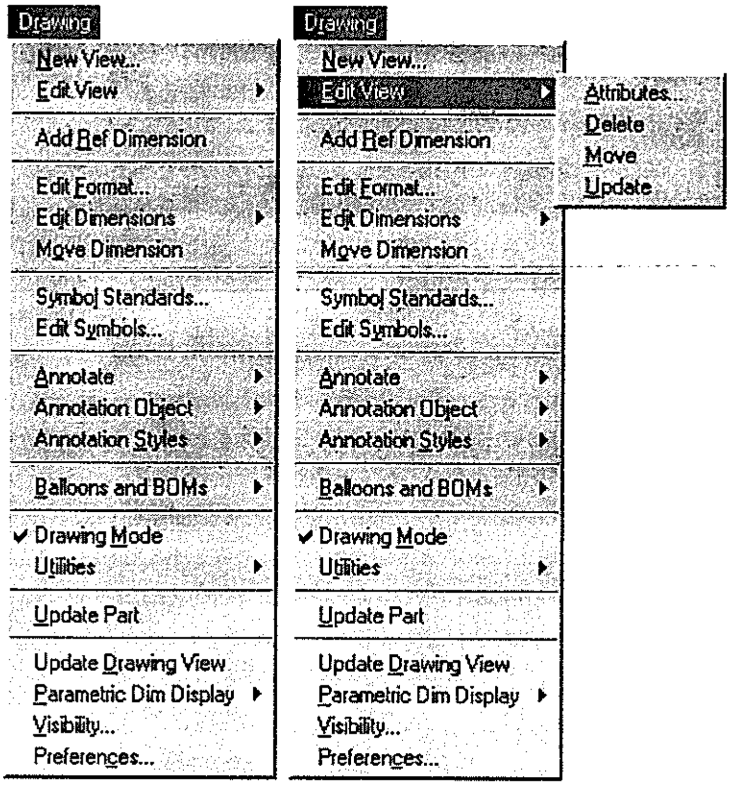


Figure A.15 Drawing pull-down menu and Edit View cascading menu

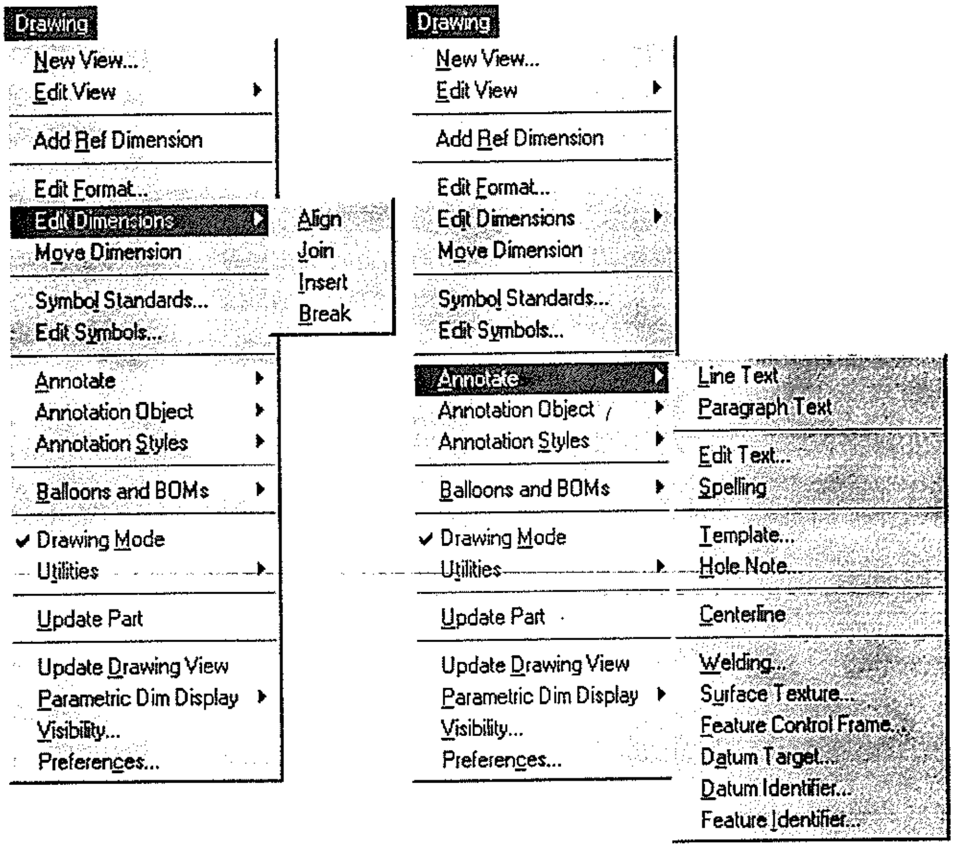


Figure A.16 Edit Dimensions and Annotate cascading menus

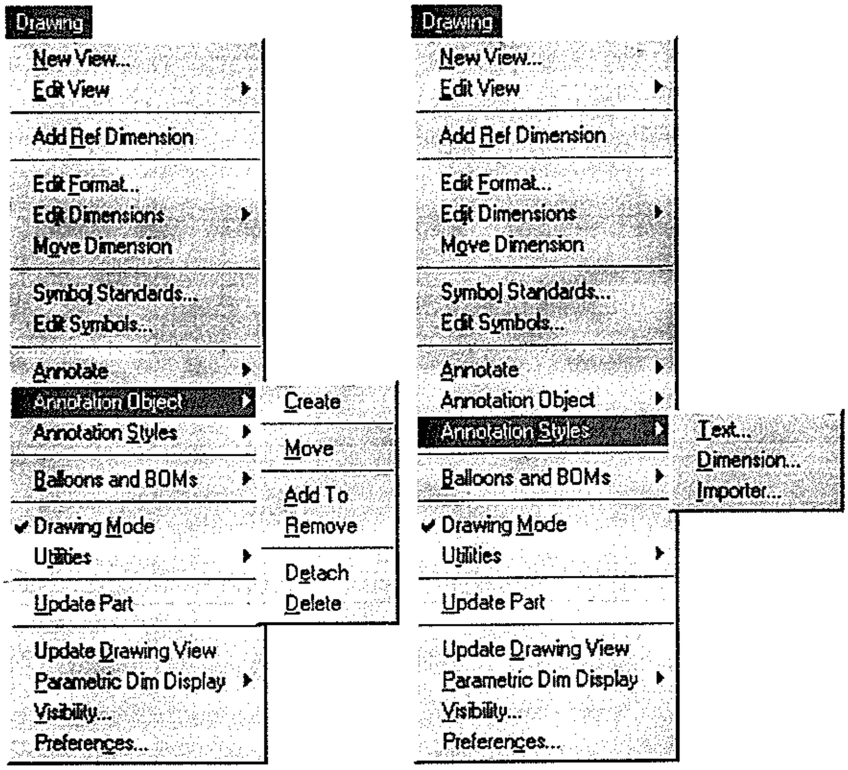


Figure A.17 Annotation Object and Annotation Styles cascading menus

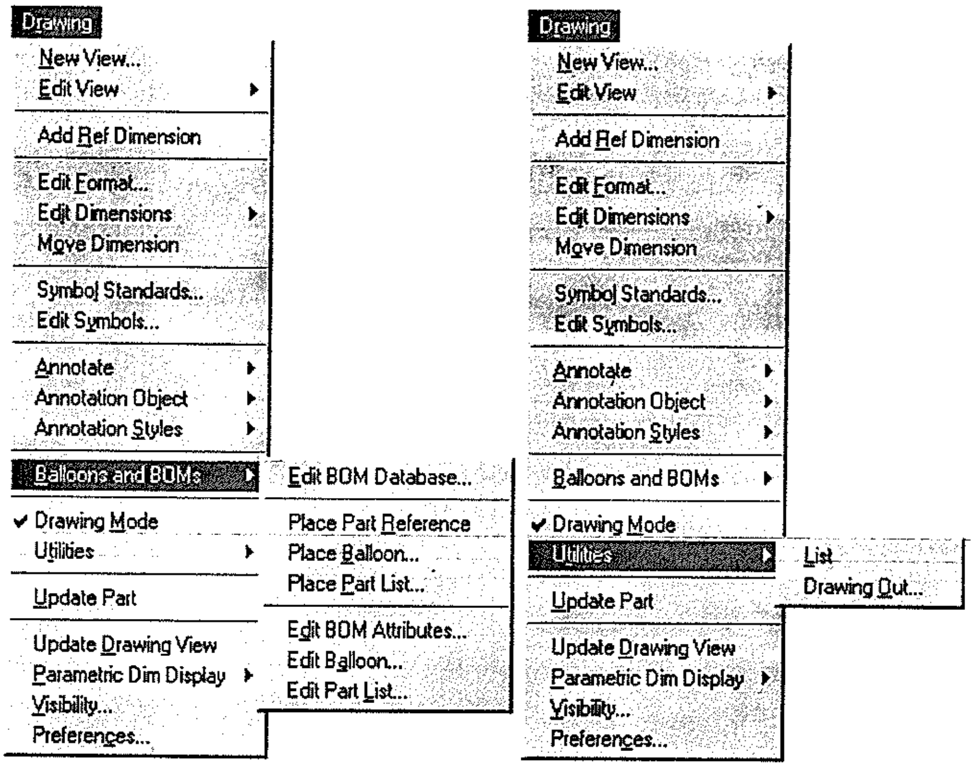


Figure A.18 Balloons and BOMs and Utilities cascading menus

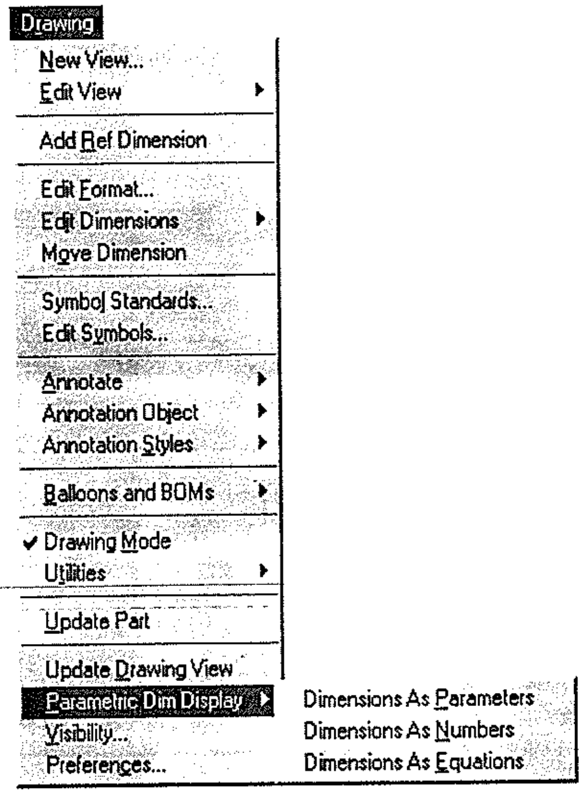


Figure A.19 Parametric Dim Display cascading menu

Figure A.20 Drawing Layout toolbar

A.5 Visualization

AV3DVARS	changes the user-configurable 3D display options.
AVEDGES	displays edges on shaded graphics.
AVHIDE	sets the 3D display mode to hidden line.
AVMAT	displays the 3D Graphics Material Values dialog box.
AVPAN	moves the 3D display in the current viewport along the view plane.
AVRENDER	sets the 3D display for rendering.
AVROTATE	performs 3D view rotation of a model about its center of geometry.
AVROTATEPT	rotates a model about a user-specified point.
AVSELECT	activates the cursor for selecting the items to be included in a 3D display.
AVWIREFRAME	sets the 3D display mode to wireframe.
AVZOOM	increases or decreases the 3D display magnification of the model.

Figure A.21 Desktop View toolbar

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