

A 3D CAD model of a mechanical assembly, possibly a bracket or a part of a machine. The model is rendered in a light gray color. A yellow wireframe is overlaid on the model, highlighting the underlying geometry. The wireframe is composed of lines that define the surfaces and edges of the part. The background is white.

Chapter 9

Working with Wireframe and Surface Design Workbench

Learning Objectives

After completing this chapter you will be able to:

- Create wireframe geometry.
- Create extruded surfaces.
- Create revolved surfaces.
- Create spherical surfaces.
- Create offset surfaces.
- Create swept surfaces.
- Create fill surfaces.
- Create loft surfaces.
- Create blend surfaces.
- Split surfaces.
- Trim surfaces.
- Join surfaces.

NEED OF SURFACE MODELING

The product and industrial designers these days are giving importance to product styling and providing a unique shape to components. Generally, this is done to make sure that the product looks attractive and presentable to the customer. The shape of products are managed using the surface modeling techniques. Surface models are three-dimensional models with no thickness and unlike solid models, they do not have mass properties. CATIA V5 provides a number of surface modeling tools to create complex three-dimensional surface models. Various workbenches in CATIA V5 with surface creation tools are:

1. Wireframe and Surface Design.
2. Generative Shape Design.
3. FreeStyle.

In this textbook, you will learn about the surface modeling tools available in the **Wireframe and Surface Design** workbench.

WIREFRAME AND SURFACE DESIGN WORKBENCH

The **Wireframe and Surface Design** workbench provides the tools to create wireframe construction elements during preliminary design and enrich existing 3D mechanical part design with wireframe and basic surface features.

Starting Wireframe and Surface Design Workbench

Start a new session of CATIA and close the new product file, which is opened by default. Next, choose **Start > Mechanical Design > Wireframe and Surface Design** from the menu bar to start a new file in the **Wireframe and Surface Design Workbench**.

CREATING WIREFRAME ELEMENTS

The wireframe construction elements aid in creating surfaces. The sketches drawn in sketcher workbench can also be used to create surfaces. The tools available for constructing the wireframe geometries are discussed in the following section.

Creating Circles

Menu:	Insert > Wireframe > Circle
Toolbar:	Circle-Corne... > Circle



The **Circle** tool is used to create circular arcs and circles. Choose the **Circle** button from the **Wireframe** toolbar; the **Circle Definition** dialog box is displayed, as shown in Figure 9-1. The **Center and radius** option is selected by default in the **Circle type** drop-down list; you are prompted to select the center point. You can select a predefined point or create a point by choosing any one of the options from the contextual menu, which is available when you right click on the **Center** selection area of the **Circle Definition** dialog box. Next, you are prompted to select the support surface. Select a plane as the support surface. Specify the required radius value in the **Radius** spinner. You can set the angular limits of the arc from the **Circle Limitation** area and finally choose the **OK** button to complete the arc .

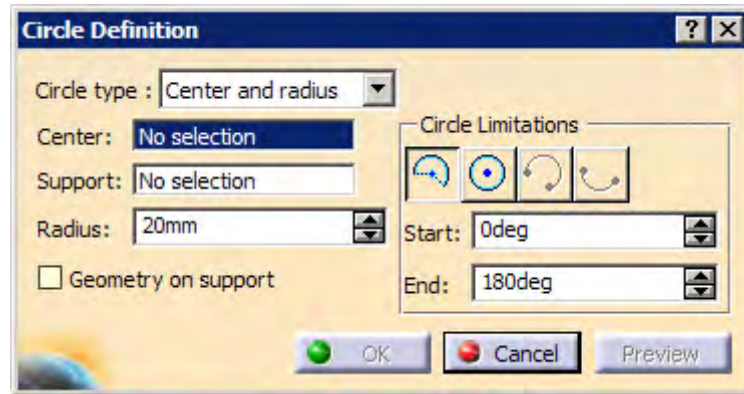


Figure 9-1 The Circle Definition dialog box

The other tools such as **Corner**, **Connect Curve** were discussed in the earlier chapters.

Creating Splines

Menu: Insert > Wireframe > Spline
Toolbar: Curve > Spline



The **Spline** tool is used to draw a spline in three dimensional space by selecting the connecting points. Choose the down arrow on the right of the **Spline** button to invoke the **Curves** toolbar, as shown in Figure 9-2, and then choose the spline button.

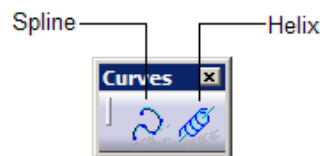


Figure 9-2 The Curve toolbar

The **Spline Definition** dialog box, as shown in Figure 9-3, is displayed and you are prompted to select a point. You can select a predefined point or create a point using the options from the contextual menu, which will be displayed when you right-click in the **Points** selection area of the dialog box. Once you have selected a point, you are further prompted to select a point or a direction (line or plane) or a curve. You can choose a number of points to draw the spline.

In the **Spline Definition** dialog box, **Geometry on Support** check box is provided. On selecting this check box you are prompted to select a support element. Select a plane or a surface such that the point defined for spline creation lies on it. The spline thus created will lie completely on the defined support element. Choose **OK** button from the dialog box to complete the spline.

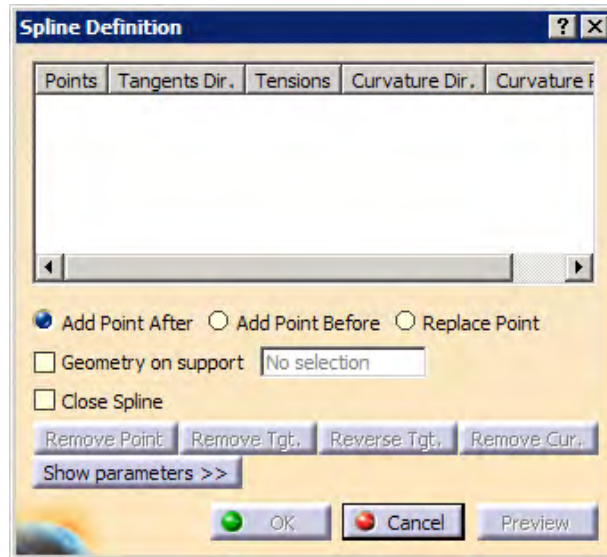


Figure 9-3 The Spline definition dialog box

Creating Helix

Menu: Insert > Wireframe > Helix
Toolbar: Curve > Helix



The **Helix** tool is used to create a helical curve. When you invoke this tool, the **Helix Curve Definition** dialog box will be displayed, as shown in Figure 9-4, and you are prompted to select the helix starting point. Select a predefined point, or create a point using the options from the contextual menu, which will be displayed when you right-click on the **Starting point** selection area of the **Helix Curve Definition** dialog box. Next, you are prompted to select a line as the helix axis. Select a predefined line or draw a line using the options from the contextual menu, which will be displayed when you right-click in the **Axis** selection area. You can set the pitch, height, orientation, and start angle values in the respective spinners. You can also add a taper angle to the helix by specifying a value in the **Taper Angle** spinner available in the **Radius variation** area of the dialog box. Figure 9-5 shows a helix without a taper angle and Figure 9-6 shows a helix with a taper angle.

CREATING SURFACES

The tools provided in **Wireframe and Surface Design** workbench to create simple and complex surfaces are discussed in the following section.

Creating Extruded Surfaces

Menu: Insert > Surfaces > Extrude
Toolbar: Surfaces > Extrude



The extruded surfaces are created by extruding a profile and specifying the extrusion depth and direction vector. The basic parameters that are required to

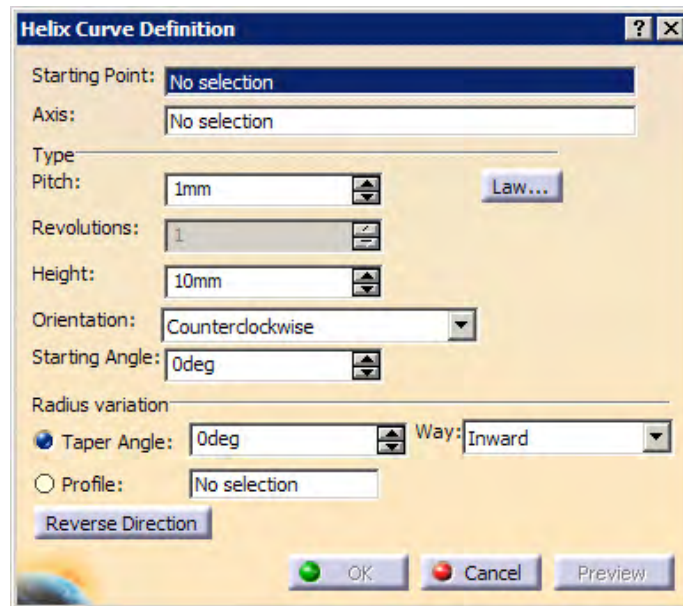


Figure 9-4 The Helix Curve definition dialog box

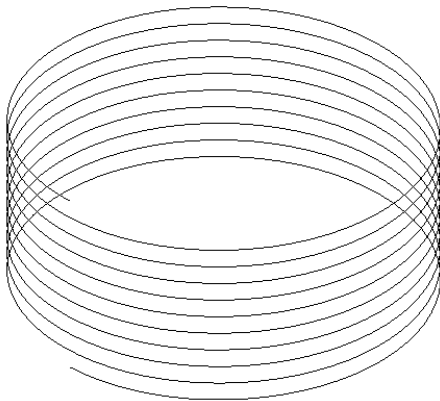


Figure 9-5 The helix without specifying the taper angle

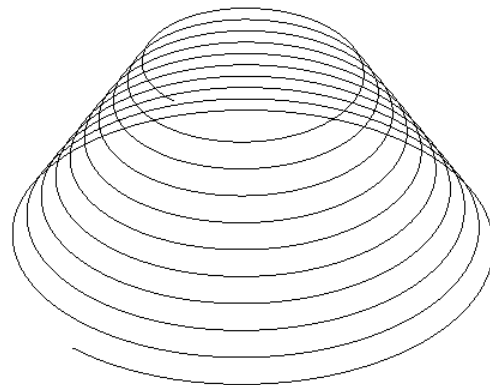


Figure 9-6 The helix with specified taper angle

create an extruded surface are profile, direction for extrusion, and extrusion limits. To create an extruded surface, you first need to draw the profile to be extruded using the **Sketcher** workbench or by using the tools available in the **Wireframe** toolbar. Once you have drawn the profile, choose the **Extrude** button from the **Surfaces** toolbar; the **Extrude Surface Definition** dialog box is displayed, as shown in Figure 9-7.

If the profile is selected before invoking this tool, the preview of the extruded surface is displayed in the geometry area. Otherwise you are prompted to select the profile to be extruded. Select a profile to be extruded. If you draw the profile using the tools from the **Wireframe** toolbar, then you are prompted to specify the direction for extrusion. Specify the

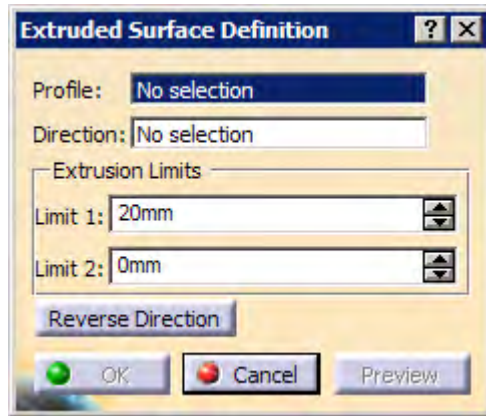


Figure 9-7 The *Extruded Surface Definition* dialog box

direction by selecting a plane normal to the profile. You can also specify a line, or an axis for specifying the direction for extrusion. Set the extrusion limits in the **Limit** spinners. Figure 9-8 shows the profile to be extruded and Figure 9-9 shows the resulting extruded surfaces.

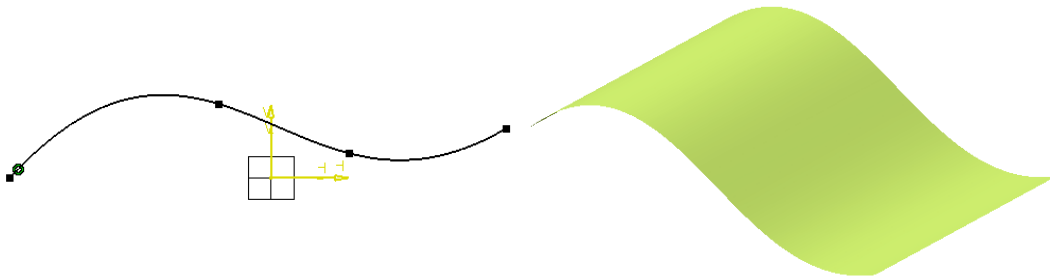


Figure 9-8 The profile to be extruded

Figure 9-9 The resulting extruded surface



Tip. You can also select an edge of an existing surface or a solid body as the profile to create an extruded surface.

Creating Revolved Surfaces

Menu: Insert > Surface > Revolve
Toolbar: Surface > Revolve



Revolved surfaces are created by revolving a profile about a revolution axis. To create a revolved surface, first sketch the profile and revolution axis around which the profile is to be revolved. Choose the **Revolve** button from the **Surfaces** toolbar; the **Revolution Surface Definition** dialog box is displayed, as shown in Figure 9-10.

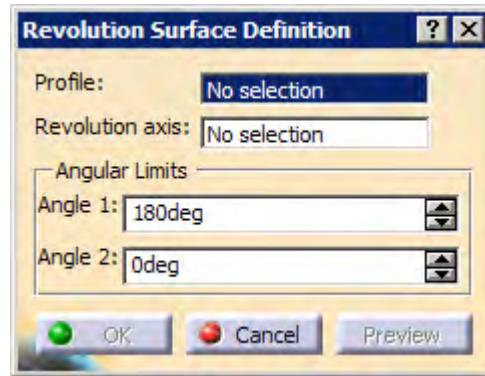


Figure 9-10 The Revolution Surface Definition dialog box

Select the profile to be revolved. By default, the axis you sketched, with the profile in the sketcher workbench, is selected as the axis of revolution. You can also select some other axis of revolution. Now, set the required angular limits in the **Angle** spinners. Figure 9-11 shows a profile and an axis of revolution to create the revolve surface. The resulting surface, revolved through an angle of 180-degree, is shown in Figure 9-12.

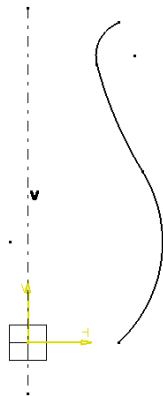


Figure 9-11 The profile and revolution axis



Figure 9-12 Surface revolved through an angle of 180-degree

Creating Spherical Surfaces

Menu: Insert > Surfaces > Sphere
Toolbar: Surfaces > Sphere



This tool is used to create the spherical surfaces. When you invoke this tool, the **Sphere Surface Definition** dialog box is displayed, as shown in Figure 9-13. You need to select the center point and an axis system as the sphere axis. You can select an existing point as the center point or create a point by using the options from the contextual menu, which will be displayed on right-clicking in the **Center** selection area. The **Default(xyz)** axis system is automatically selected. You can also select any previously created axis system. The preview of the spherical surface is displayed in the geometry area. You can

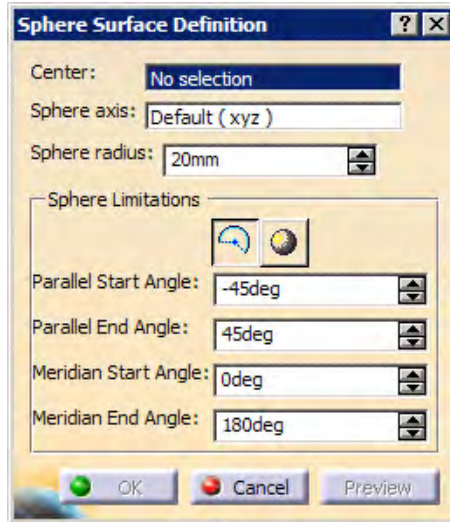


Figure 9-13 The Sphere Surface Definition dialog

vary the angle values using the options available in the **Sphere Limitations** area or by directly dragging the limiting arrows in the geometry area. Figure 9-14 shows the spherical surface created by defining the origin as the center. Also, this surface has the default axis system and sphere limitation values.



Figure 9-14 A spherical surface



Tip. You can create complete sphere using the **Sphere** button available in the **Sphere Limitations** area of the **Sphere Surface Definition** dialog box.

Creating Cylindrical Surfaces

Menu: Insert > Surfaces > Cylinder
Toolbar: Surfaces > Cylinder



This tool is used to create cylindrical surfaces. Choose the **Cylinder** button from the **Surfaces** toolbar; the **Cylinder Surface Definition** dialog box is displayed and you are prompted to select the center of the cylinder. You can select an existing point as the center point or create a point by using the options from the contextual menu, which will be displayed on right-clicking in the **Center** selection area. Next, you are prompted to specify the direction for the cylinder. Select a plane, normal to which the cylinder will be extruded. You can also select a direction vector from the contextual menu, which can be invoked by right-clicking in the **direction** selection area. Set the parameters using the spinners in the **Parameters** area in the **Surface Definition** dialog box. Choose **OK** to create the cylindrical surface.

Creating Offset Surfaces

Menu: Insert > Surfaces > Offset
Toolbar: Surfaces > Offset



The **Offset** tool is used to create a surface that is at an offset distance from a reference surface. To do so, choose the **Offset** tool from the **Surfaces** toolbar. The **Offset Surface Definition** dialog box is displayed, as shown in Figure 9-15, and you are prompted to select a reference surface.

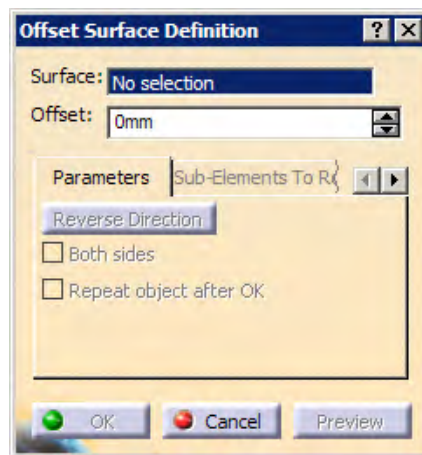


Figure 9-15 The **Offset Surface Definition** dialog box

Select the reference surface from the geometry area and specify the offset value in the **Offset** spinner. Choose the **Reverse Direction** button available in the dialog box to reverse the offset direction. The **Both sides** check box is selected to create the offset surface on both sides of the reference surface. The **Repeat object after OK** check box is used to create multiple offset surfaces. Select the **Repeat object after OK** check box and exit the **Offset Surface Definition** dialog box. The **Object Repetition** dialog box is displayed, as shown in Figure 9-16.

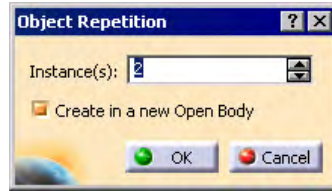


Figure 9-16 The *Object Repetition* dialog box

In this dialog box specify the required number of instance(s). Choose the **OK** button to create the offset surfaces. Figure 9-17 shows a reference surface and an offset surface.

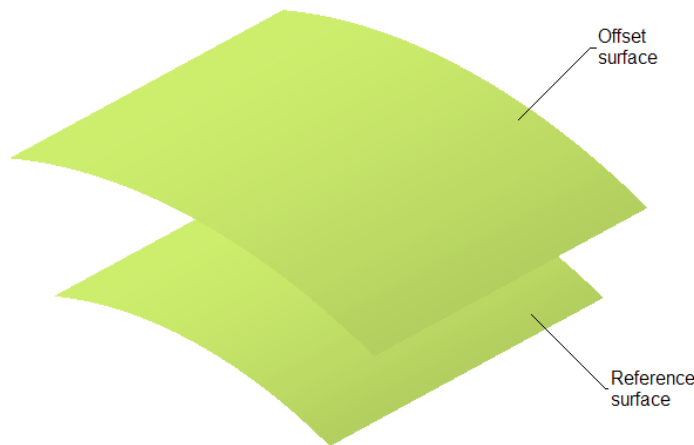


Figure 9-17 An offset surface



Note

Sometime for complex reference surfaces, the offset surface may not be created. In such cases, you need to reduce the offset value or modify the initial geometry.

Creating Swept surfaces

Menu: Insert > Surface > Sweep
Toolbar: Surface > Sweep



The sweep tool is provided to create surfaces by sweeping a profile along a guide curve in the **Wireframe and Surfaces Design** workbench of CATIA V5. To create a swept surface, you first need to draw a profile and a guide curve as two separate sketches. Next, choose the **Sweep** button from the **Surfaces** toolbar. The **Swept Surface Definition** dialog box is displayed, as shown in Figure 9-18, and you are prompted to select a profile. Select the profile from the geometry area; you are prompted to select a guide curve. Select the guide curve from the geometry area. Now, choose the **OK** button from the **Swept Surface Definition** dialog box. Figure 9-19 show a profile and a guide curve and Figure 9-20 shows the resulting swept surface.

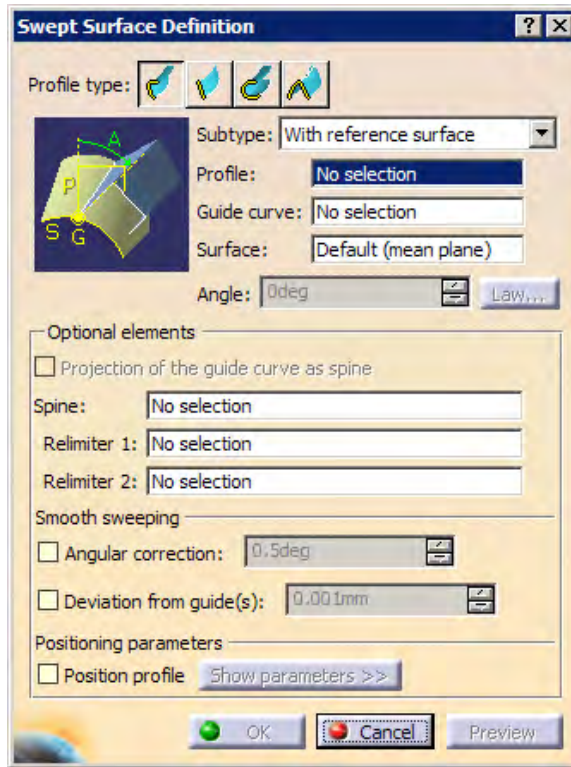


Figure 9-18 The Swept Surface Definition dialog box

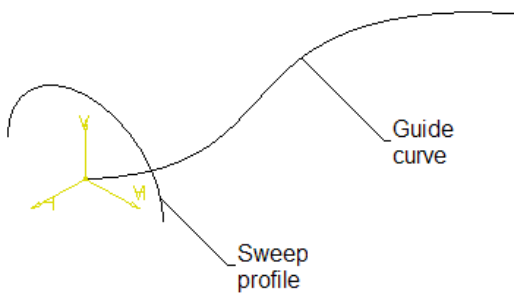


Figure 9-19 The sweep profile and guide curve



Figure 9-20 The resulting swept surface



Tip. Sometimes the swept surface may not be created, as the created geometry forms a cusp. In such a case, reduce the curvature of the guide curves.

Various other tools to create swept surfaces are discussed in the following section.

Swept Surface with two Guide Curves

You can also create a swept surface using more than one guide curve. First draw a profile and two guide curves as separate sketches. Now, select the **Sweep** button from the **Surfaces** toolbar; the **Swept Surface Definition** dialog box is displayed. Select **With two guide curve** option from the **Subtypes** drop-down list; you are prompted to select a profile. After you select the profile, you are prompted to select a guide curve. Select the first and second guide curves. Now, select the anchor point for the respective guide curves. Choose the **Preview** button from the **Swept Surface Definition** dialog box to preview the surface created. Choose the **OK** button from the **Swept Surface Definition** dialog box. Figure 9-21 shows a profile and guide curves. The swept surface created using the two guide curve is shown in Figure 9-22.

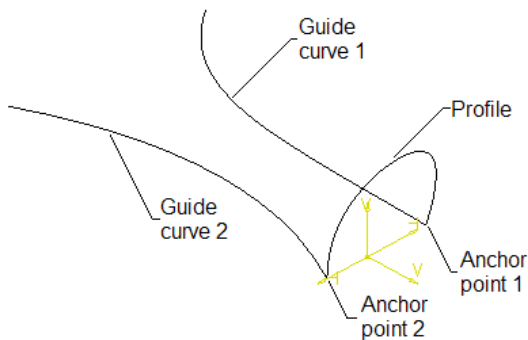


Figure 9-21 The sweep profile and guide curves



Figure 9-22 The resulting swept surface

Swept Surface with Two Limits

In CATIA V5, you can create a swept surface by defining the two limit curves. The limit curves can be in the same or different planes. To create swept surface with two limits, you need to draw two limit curves. Once you have drawn the curves, choose the **Sweep** button from the **Surfaces** toolbar; the **Swept Surface Definition** dialog box will be displayed. Select the **Line** button from the **Profile Type** area in the dialog box. The parameters in the **Swept Surface Definition** dialog box change and will appear as shown in Figure 9-23. You are prompted to select the first guide curve. After selecting the first guide curve, you are prompted to select a second guide curve. Select the second guide curve and choose the **Preview** button to display the swept surface created between the limiting curve. Note that in the **Optional elements** area, guide curve 1 is selected by default in the **Spine** selection area. You can select another curve to be define as spine. Choose the **OK** button from the dialog box to create the swept surface. Figure 9-24 shows the limit curves and Figure 9-25 shows the resulting surface.

Swept Surface with Three Curve

You can also create a circular swept surface using three guide curves. To create a surface using this tool, you first need to draw three guide curves, which should lie in different planes. After drawing the curves, invoke the **Swept surface Definition** dialog box. Choose the **Circle** button from the **Profile type** area in the dialog box. The parameters in the **Swept surface**

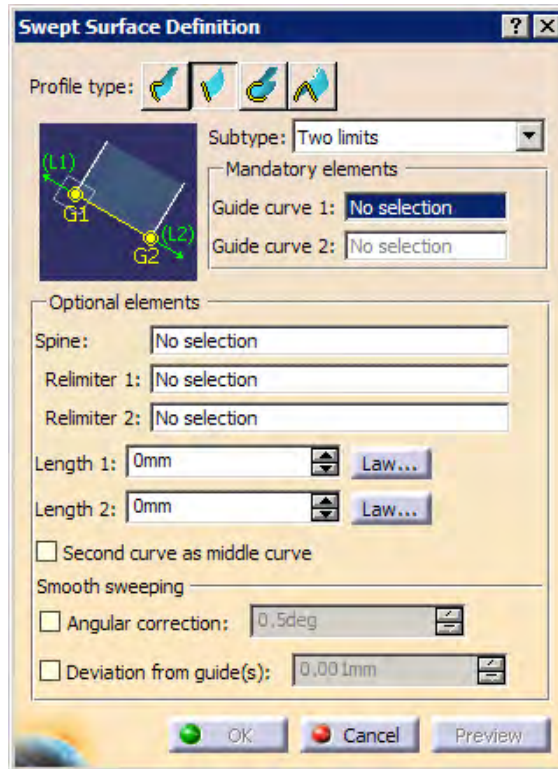


Figure 9-23 The Swept Surface Definition dialog box

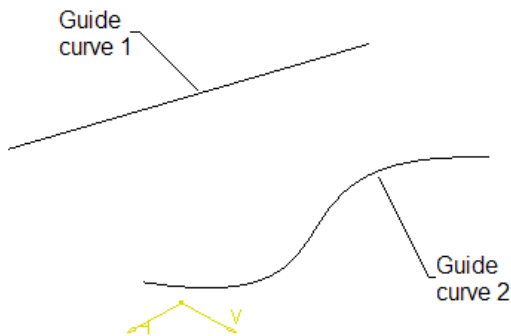


Figure 9-24 The sweep profile and guide curves

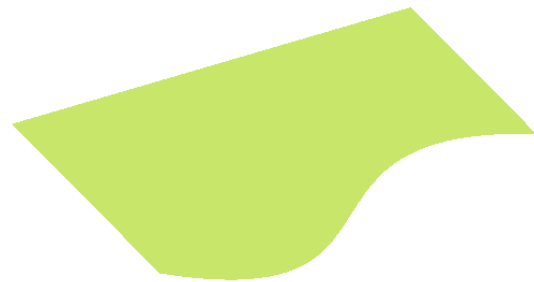


Figure 9-25 The resulting swept surface

Definition dialog box will change, as shown in Figure 9-26. You are prompted to select a first guide curve that define the first extremity of the circular arc. Select the first guide curve. Next you are prompted to select the second guide curve. After selecting the second guide curve, you are prompted to select a guide curve that defines the second extremity of the circular arc. Select the guide curve and choose the OK button from the dialog box to complete the swept surface. Figure 9-27 shows the guide curves and Figure 9-28 shows the resulting swept surface.

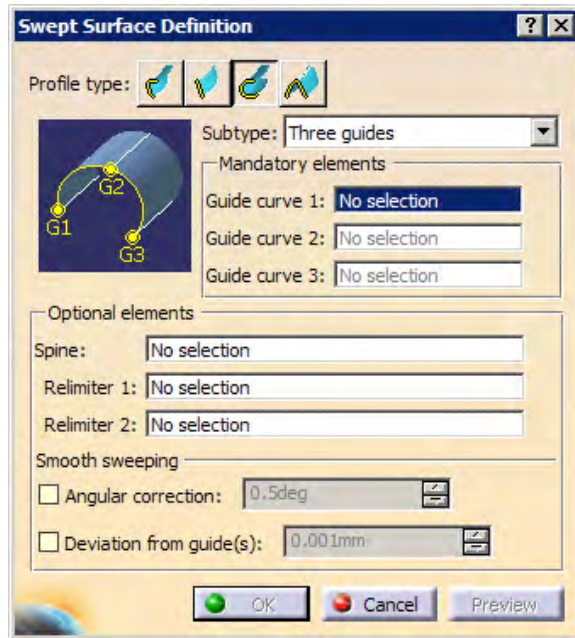


Figure 9-26 The Swept Surface Definition dialog box

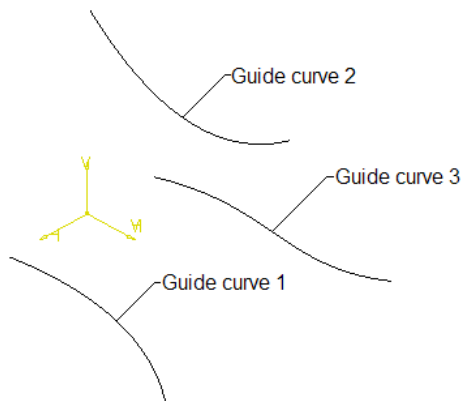


Figure 9-27 The guide curves



Figure 9-28 The resulting swept surface

Creating Fill Surfaces

Menu: Insert > Surfaces > Fill
Toolbar: Surfaces > Fill



The **Fill** tool allows you to create fill surfaces between a number of boundary segments. These may be planar or non-planar, but there should not be a large gap between the consecutive boundary segments. Before creating a fill surface, draw the boundary curves. Next, choose **Fill** button from the **Surface** toolbar. The **Fill Surface Definition** dialog box is displayed, as shown in Figure 9-29.

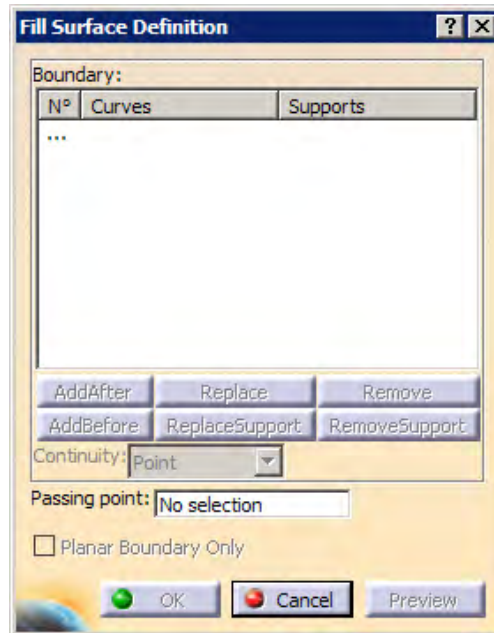


Figure 9-29 The *Fill Surface Definition* dialog box

Next, you are required to select the boundary segments. You need to make sure that while selecting the boundary segments, the sequence of selection should be such that a closed loop is formed. Once you have selected the boundary curves, choose the **OK** button. Figure 9-30 shows the curves drawn to create the fill surface and Figure 9-31 shows the resulting fill surface.

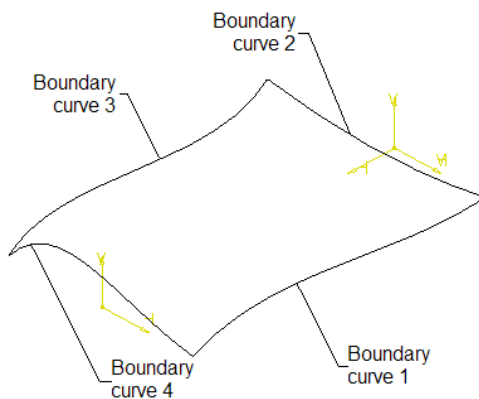


Figure 9-30 The *Boundary curves*

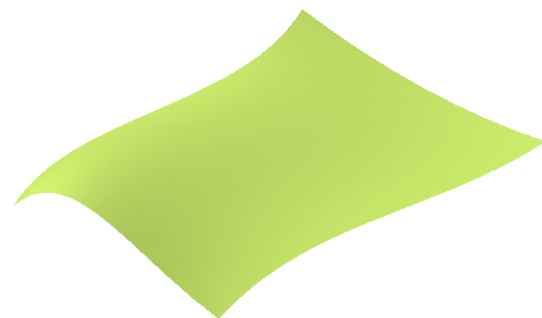


Figure 9-31 The *resulting fill surface*

Support surfaces may be selected with the respective curve to ensure the continuity between the fill surface and the support surface. Similarly, if you select a passing point, the fill surface will be created, such that it passes through the selected point.

Creating Lofted Multisection Surfaces

Menu: Insert > Surface > Multisections surface
Toolbar: Surface > Multisections surface



This tool allows you to create lofted multisection surfaces. The surface is created between the sections along the computed or user-defined spine. To create a multisection surface, you first need to create sections and guide curves. Next, choose the **Multisections surface** button from the **Surface** toolbar; the **Multi-sections Surface Definition** dialog box is displayed, as shown in Figure 9-32.

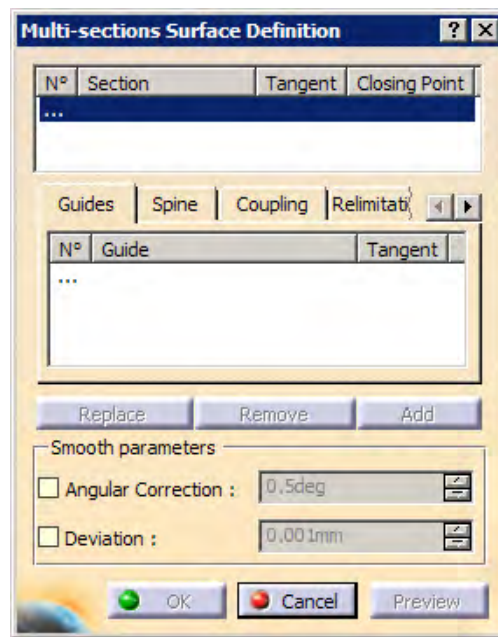


Figure 9-32 The *Multi-sections Surface Definition* dialog box

You are prompted to select a curve. Select the first section curve; you are prompted to select new curve or select a tangent surface. Select the second section curve from the geometry area and click on the **Guide** selection area to activate it. You are prompted to select a curve. One by one, select the guide curves that were drawn earlier. Choose the **OK** button to exit the **Multi-sections Surface Definition** dialog, box and complete the multi section surface. Figure 9-33 shows sections and guide curves to create the multisection surface and Figure 9-34 shows the resulting surface.



Note

While selecting the section curve, make sure the arrow associated with each section curve is pointing in the same direction. Else, the surface will result in a cusp and will not be created. In some cases, with arrows pointing in opposite direction, a twisted surface may be formed.

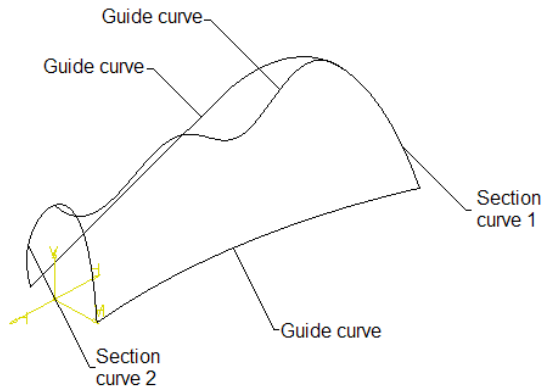


Figure 9-33 Sections and guide curves

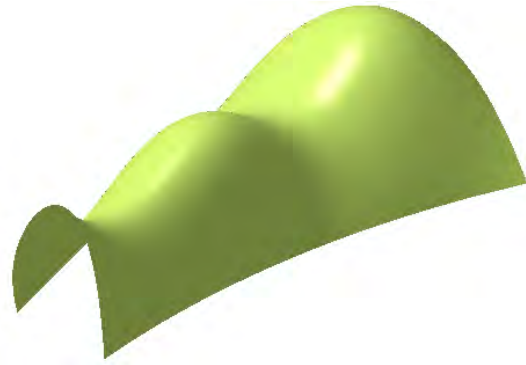


Figure 9-34 The resulting multisection

Creating Blended Surfaces

Menu: Insert > Surfaces > Blend
Toolbar: Surfaces > Blend



This tool allows you to create a surface by blending two curves. These curves can be sketched curves, wireframe geometries, or edge of existing surfaces. If you select support surfaces with curves, the resulting blend surface will be tangent to the support surfaces. To create a blend surface, draw some curves and create support surfaces. Choose the **Blend** button from the **Surfaces** toolbar. The **Blend Definition** dialog box is displayed, as shown in Figure 9-35.

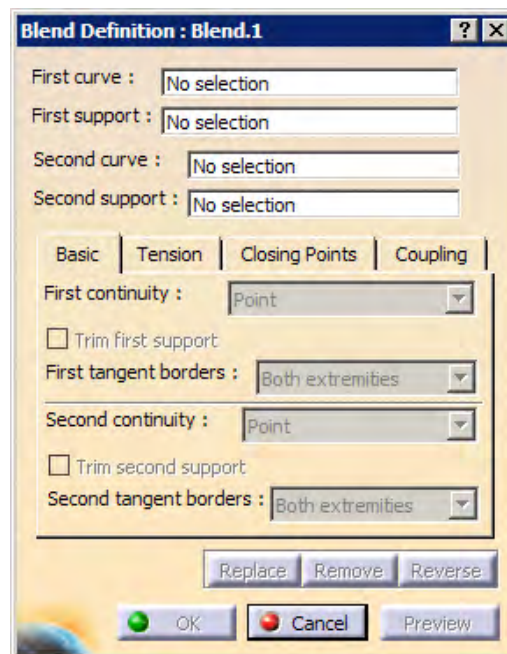


Figure 9-35 The Blend Definition dialog box

You are now prompted to select the first curve and first support. Select the curve and support. Next, you are prompted to select the second curve and second support. Select them choose the **OK** button from the **Blend Definition** dialog box. Figure 9-36 shows the curves and support surfaces to create the blend surface and Figure 9-37 shows the resulting blend surface.

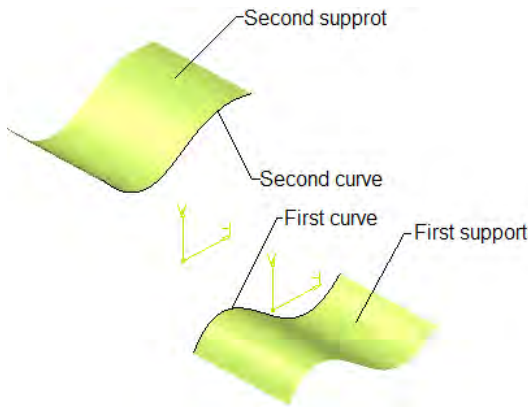


Figure 9-36 The curves and support surfaces



Figure 9-37 The resulting blend surface

OPERATION ON SHAPE GEOMETRY

Generally, the surface models are a combination of various surfaces. You need to join, trim, split, or translate the surfaces to manage multiple surfaces. CATIA provides a number of such operation tools that can be used on the surfaces created using the tools discussed earlier in this chapter. Some of these operations are discussed in the following section.

Joining Surfaces

Menu: Insert > Operations > Join
Toolbar: Operations > Join



The **Join** tool is used to join two adjacent surfaces or two adjacent curves. Choose **Join** button from the **Operation** toolbar; the **Join Definition** dialog box is displayed, as shown in Figure 9-38. You are prompted to select the elements (curves or surfaces) to be joined. Select the elements that you need to join. Remember that there should not be a large gap between the entities to be joined. Choose the **OK** button from the **Join Definition** dialog box to complete the join operation. The surfaces or curves that are joined together will behave as a single entity.

Splitting Surfaces

Menu: Insert > Operations > Split
Toolbar: Operations > Split



The **Split** tool is used to split a surface or a wireframe element using a cutting element. A wireframe element can be split using a point, another wireframe element, or a surface. A surface can be split using another surface or a wireframe element. To

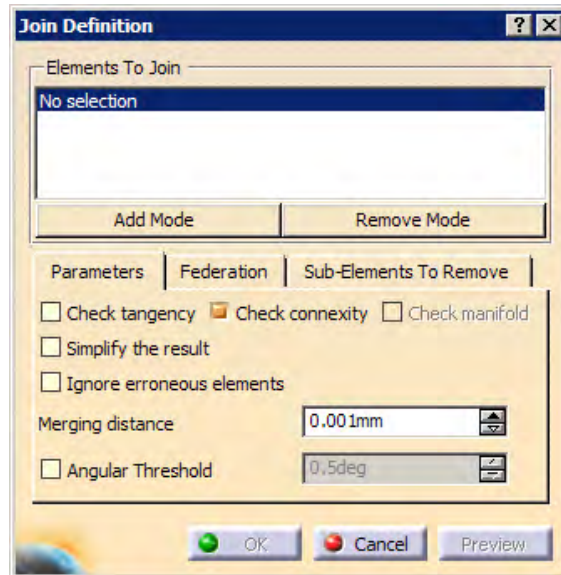


Figure 9-38 The *Join Definition* dialog box

understand the concept of this tool, consider a case in which two intersecting surfaces are created, as shown in Figure 9-39. Choose the **Split** button from the **Operations** toolbar to invoke the **Split Definition** dialog box; you are prompted to select the curve or surface to split. Select the cylindrical surface and then select the other surface, as the cutting element. The side of the cylindrical surface that is to be removed will be displayed as transparent. You can choose the **Other side** buttons from the **Split Definition** dialog box to reverse the side of the surface to be removed. You can also retain both sides of the split surface by selecting the **Keep both sides** check box from the **Split Definition** dialog box. Choose the **OK** button to split the cylindrical surface. The transparent side will be removed. Figure 9-40 shows the split surface.

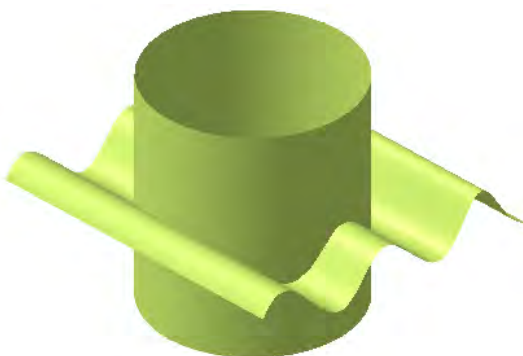


Figure 9-39 The split surface and the cutting surfaces

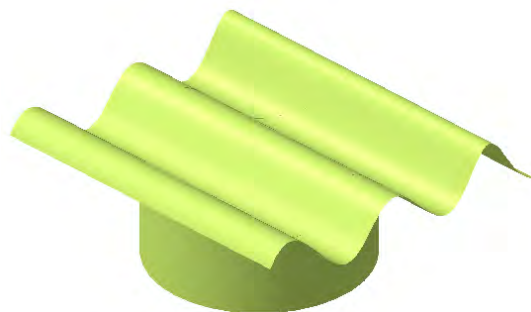


Figure 9-40 The resulting split surface

Trimming Surfaces

Menu: Insert > Operations > Trim

Toolbar: Operations > Trim



The **Trim** tool allows you to trim two intersecting surfaces or curves with respect to each other. When you invoke this tool, the **Trim Definition** dialog box will be displayed, as shown in Figure 9-41. Choose a surface as the first element and the other surface as the second element. One side of each surface will be displayed as transparent. You can choose the **Other side** button to reverse the sides of surfaces to be trimmed. Figure 9-42 shows the surface Trimmed using the **Trim** tool. Figure 9-43 shows the trimmed surface when the **Other side** buttons are selected for both surfaces from the **Trim Definition** dialog box.

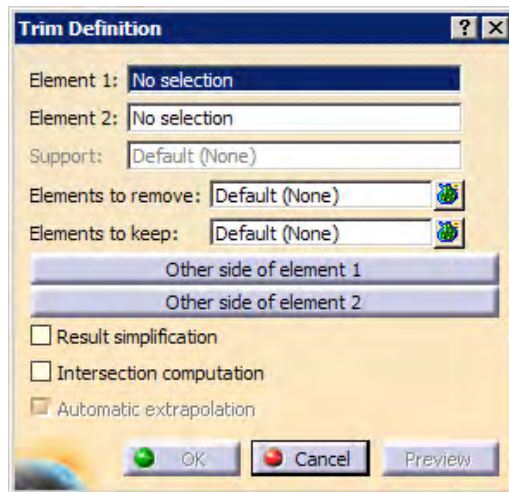


Figure 9-41 The Trim Definition dialog box

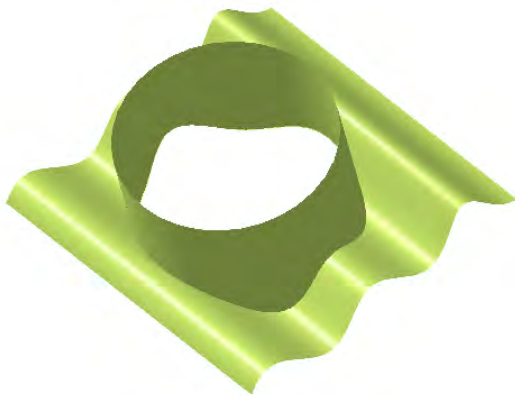


Figure 9-42 The trimmed surfaces

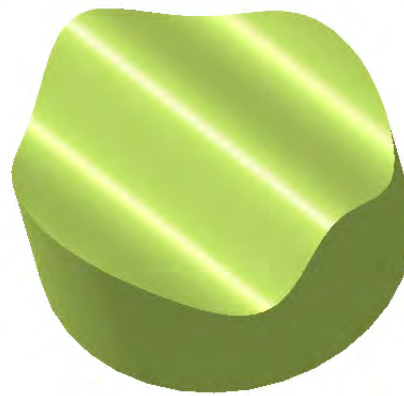


Figure 9-43 The trimmed surface

TUTORIALS

Tutorial 1

In this tutorial, you will create the model shown in Figure 9-44. The views and dimensions of the model are shown in Figure 9-45. **(Expected time: 45min)**



Figure 9-44 The isometric view of the model

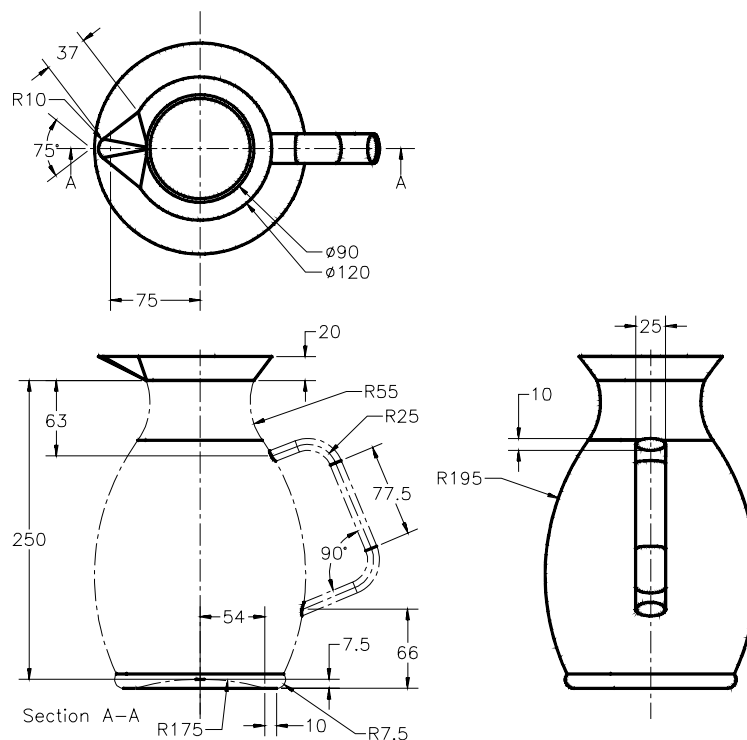


Figure 9-45 The views and dimensions of the model

The following steps are required to complete this tutorial:

- Start CATIA V5 and start a new file in the **Wireframe and Surface Design** workbench.
- Draw the sketches for the loft surface, refer to Figures 9-46 through 9-48.
- Create the loft surface, refer to Figures 9-49 and 9-50.
- Draw the sketch to create the revolve surface, refer to Figure 9-51.
- Create the revolved surface, refer to Figure 9-52.
- Draw the sketch to create the sweep profile, refer to Figures 9-53 and 9-54.
- Create the swept surface, refer to Figure 9-55.
- Split the swept surface with the revolved surface.

Starting CATIA V5R13 and Opening a New Part File

- Start CATIA V5 and choose **Close** from the **File** menu. The start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Wireframe and Surface Design** from the menu bar to start a new file in **Wireframe and Surface Design** workbench

Drawing the Sketch for Base Surface

- Invoke the **Sketcher** workbench by selecting the XY plane as the sketching plane.
- Draw a circle with center at the origin and diameter of 90.
- Place two points on the circle, as shown in Figure 9-46. These points will be used to create the guide curve and define the closing point for creating the loft surface later in this tutorial.

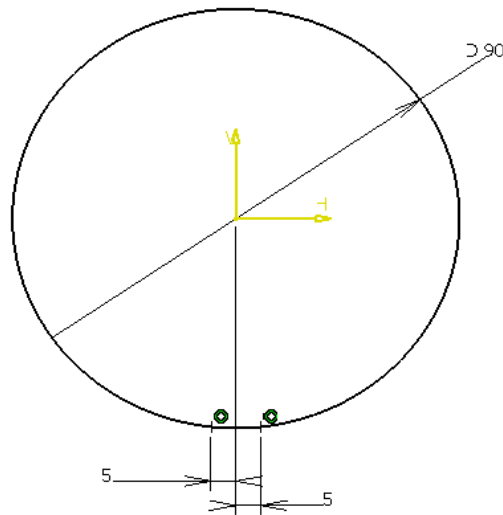


Figure 9-46 First section for the base surface

- Exit the **Sketcher** workbench.
- Create a plane at an offset of 20 from the XY plane.

6. Invoke the **Sketcher** workbench by selecting **Plane.1** as the sketching plane.
7. Draw the sketch, as shown in Figure 9-47, and exit the **Sketcher** workbench.

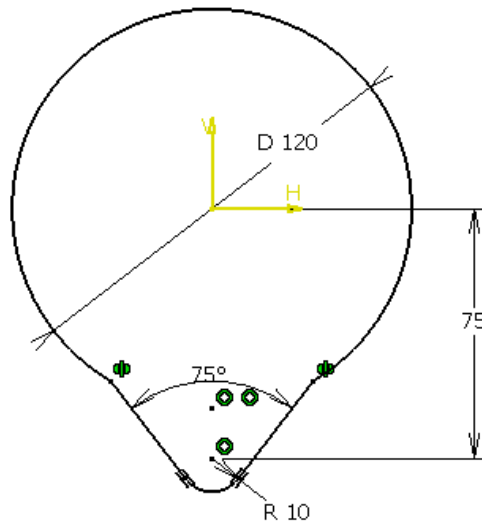


Figure 9-47 The second section for base surface

Next, you need to draw a line joining the two points in the two sections.

8. Choose the **Line** button from the **Wireframe** toolbar, the **Line Definition** dialog box is displayed. Create the line by selecting the two points, as shown in to Figure 9-48.

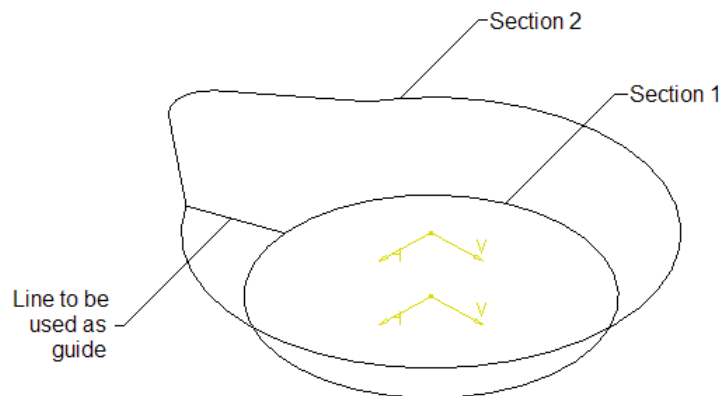



Figure 9-48 The line created between two points on the sections

Creating the Base Surface

The base surface in this case will be a loft surface.

1. Choose the **Multisections surface** button from the **Surfaces** toolbar. The **Multi-sections Surface Definition** dialog box is displayed and you are prompted to select a curve. 
2. Select section 1, as shown in Figure 9-48. The name of the selected entity is displayed in **Curve** selection area of the **Multi-sections Surface Definition** dialog box. By default, a closing point is created. Move the cursor on the text **Closing Point1** in the geometry area and right-click to invoke the contextual menu.
3. Choose the **Replace closing point** option from the contextual menu and select the point on section 1 from the geometry area, refer to Figure 9-49.

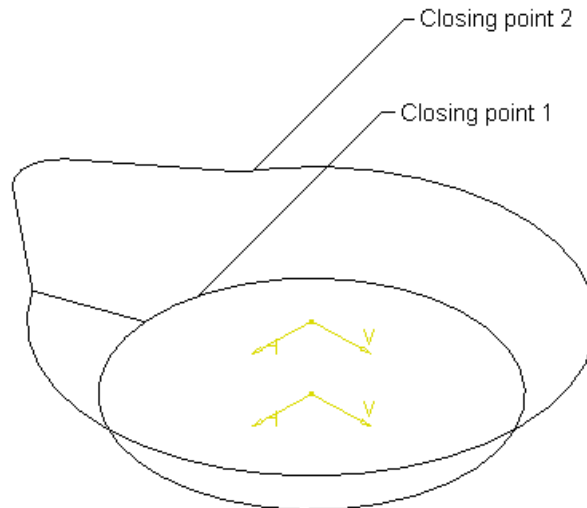


Figure 9-49 The sketch showing the position of closing points

4. Activate the **Curve** selection area by clicking on it and choose the **Add** button to select the next section. You are prompted to select a curve.
5. Select section 2 from the geometry area and replace the default closing point with the closing point given in Figure 9-49.
6. Next, click in the **Guide** selection area to invoke the selection tool; you are prompted to select a curve. Select the guide curve shown in Figure 9-48.
7. Choose the **OK** button from this dialog box to complete the multisection lofted surface. Figure 9-50 shows the multisection lofted surface.



Figure 9-50 The multisection lofted surface

Creating the Revolved Surface

Next, you need to create the revolved surface.

1. Invoke the **Sketcher** workbench by selecting YZ plane as the sketching plane.
2. Choose the **Axis** button from the **Profile** toolbar and draw a vertical axis passing through the origin.
3. Next, draw the sketch, as shown in Figure 9-51 and exit the **Sketcher** workbench

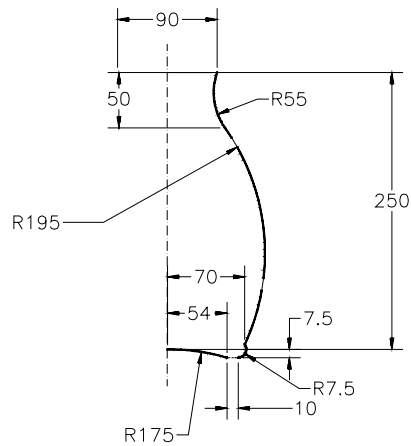


Figure 9-51 The axis and profile to be revolved

4. Choose the **Revolve** button from the **Surface** toolbar; the **Revolution Surface Definition** dialog box is displayed.



5. Select the profile to be revolved, if it is not already selected. The revolution axis drawn in the sketcher workbench is automatically selected and the preview of the revolved surface with default angle limits will be displayed in the geometry area.
6. Set the value of the **Angle 1** spinner to **360** and choose **OK** button from the **Revolution Surface Definition** dialog box. Figure 9-52 shows the model, after creating the revolved surface and hiding the sketch.



Figure 9-52 The model after creating the revolved surface

Creating the Swept surface

Next, you need to create a surface by sweeping a profile along a guide to create the handle of the jug.

1. Invoke the **Sketcher** by selecting YZ plane as the sketching plane.
2. Draw the profile of the guides, as shown in Figure 9-53.
3. Exit the **Sketcher** workbench. Click anywhere in the geometry area to exit the current selection set.
4. Create a plane normal to the guide at the upper endpoint.
5. Invoke the **Sketcher** workbench, using **Plane 2** as the sketching plane.
6. Draw an ellipse, as shown in Figure 9-54.
7. Choose the **Isometric View** button from the **View** toolbar.
8. Press and hold the CTRL key down, select the center point of the ellipse and the upper end point of the guide. Apply the **Coincident** Constraint between the two entities.
9. Exit the **Sketcher** workbench. Click anywhere in the geometry area to exit the current selection set.

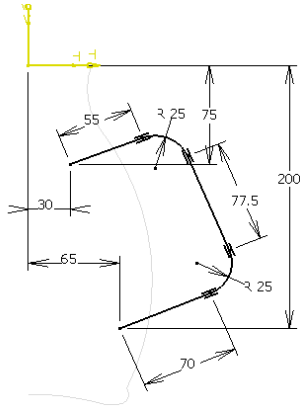


Figure 9-53 The guide for Swept surface

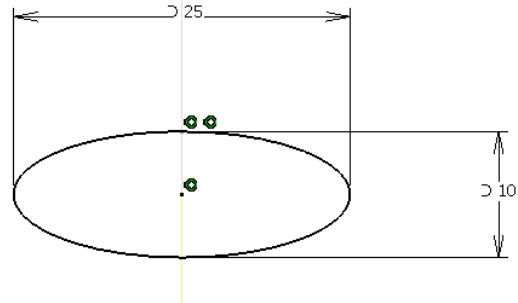




Figure 9-54 The profile for swept surface

10. Choose the **Sweep** button from the **Surface** toolbar. The **Swept Surface Definition** dialog box is displayed and you are prompted to select a profile. 
11. Select ellipse from the geometry area. You are prompted to select a guide curve.
12. Select the guide curve from the geometry area and choose the **OK** button from the dialog box to complete the swept surface.

Splitting the Swept surface

The swept surface is extended beyond the revolved surface. Therefore, you are required to remove the unwanted portion of the swept, surface which is inside the jug.

1. Choose the **Split** button from the **Operation** toolbar; the **Split Definition** dialog box is displayed and you are prompted to select the curve or surface to split. 
2. Select the swept surface from the geometry area.
3. Now, select the revolve surface as the cutting element.
4. Choose the **OK** button to complete the split operation. The model will look similar to the one shown in Figure 9-55.

On rotating the view of the model, you will note that the unwanted portion of the swept surface is removed.

Saving the File

1. Choose the **Save** button from the **Standard** toolbar; the **Save As** dialog box is displayed.
2. Enter *c09tut1* in the **File name** area. Choose the **Save** button from the **Save as** dialog box.



Figure 9-55 The isometric view of the model after splitting the swept surface

3. Close the part file by choosing **File > Close** from the menu bar.

Tutorial 2

In this tutorial, you will create the model, as shown in Figure 9-56. The drawing views and dimensions of the model are shown in Figure 9-57. **(Expected time: 45min)**

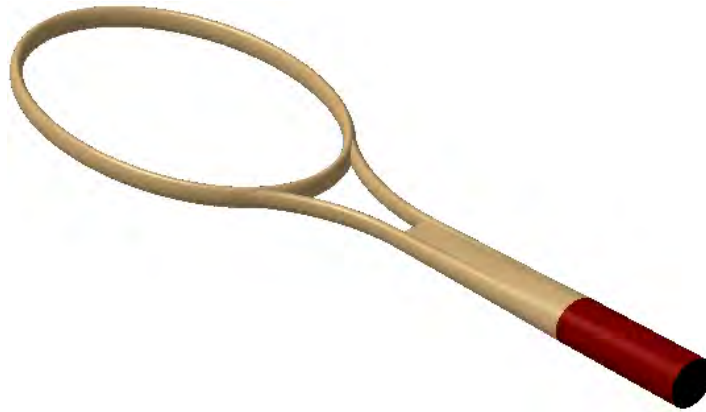


Figure 9-56 The isometric view of the model

The following steps are required to complete this tutorial:

- a. Start a new file in the **Wireframe and Surface Design** workbench.
- b. Create swept surface as the base feature, refer to Figures 9-58 through 9-60.
- c. Create the second swept surface, refer to Figures 9-61 through 9-63.

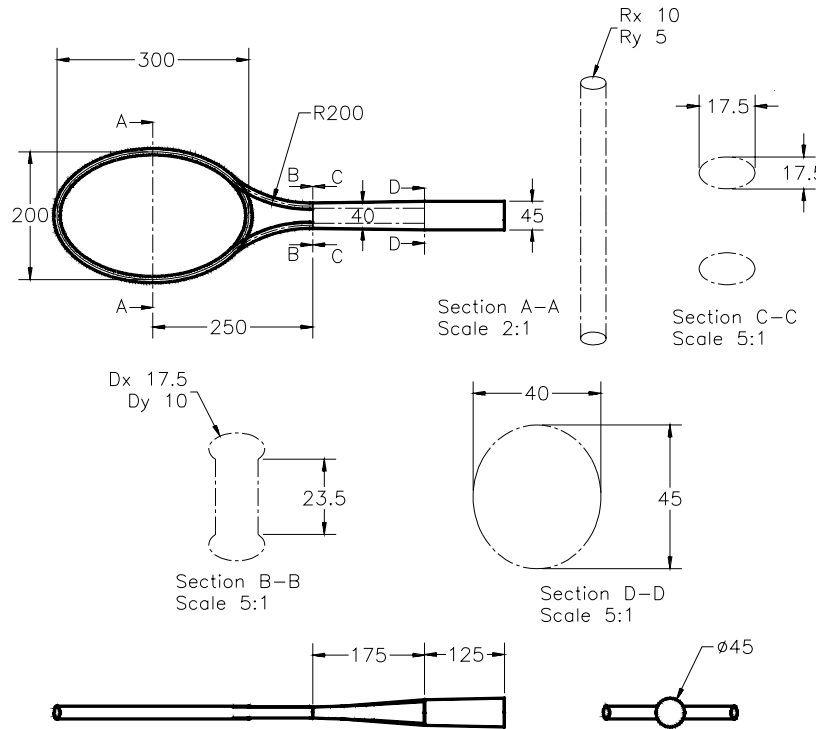


Figure 9-57 The views and dimensions of the model

- d. Create the symmetry of the second swept surface, refer to Figure 9-64.
- e. Create the multisection lofted surface, refer to Figures 9-65 through 9-68.
- f. Create the blend surface, refer to Figures 9-69 and 9-70.
- g. Create the fill surfaces, refer to Figure 9-71.

Drawing the Sketch for Base Surface

First you need to draw the profile and the guide curve for creating the swept surface.

1. Choose **Start** > **mechanical Design** > **Wireframe and Surface Design** from the menu bar.
2. Invoke the **sketcher** workbench, selecting the XY plane as the sketching plane.
3. Draw an ellipse, as shown in Figure 9-58, and exit the **Sketcher** workbench.
4. Invoke the **Sketcher** workbench, selecting the ZX plane as the sketching plane.
5. Draw an ellipse, as shown in Figure 9-59, and exit the **Sketcher** workbench.

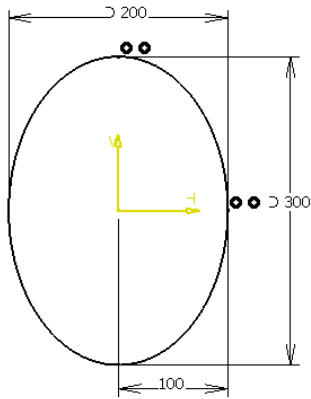


Figure 9-58 The guide curve

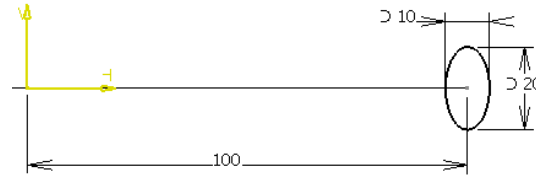



Figure 9-59 The sweep profile

Creating the Base Surface

After drawing the profile and guide curve, you need to create a swept surface by sweeping the profile along the guide curve.

1. Choose the **Sweep** button from the **Surfaces** toolbar; the **Swept Surface Definition** dialog box is displayed. 
2. The **Explicit** button is chosen in the **Profile type** area by default. If not, choose it, you are prompted to select a profile.
3. Select the sweep profile from the geometry area; you are prompted to select a guide curve.
4. Select the guide curve from the geometry area and choose the **OK** button. The resulting swept surface, after hiding the sketches is displayed, as shown in Figure 9-60.

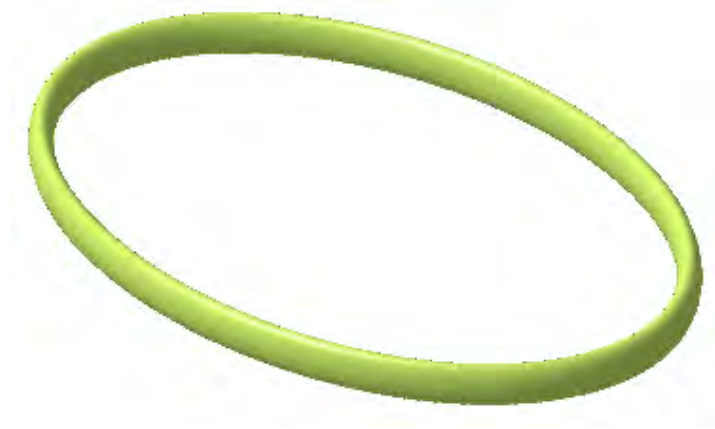


Figure 9-60 The resulting swept surface

Drawing Sketches for the Second Sweep Feature

The second feature is also a swept surface hence a profile and a guide curve is required to create this surface .

1. Invoke the **sketcher** workbench by selecting the XY plane as the sketching plane.
2. Draw the guide curve, as shown in Figure 9-61.
3. Exit the **Sketcher** workbench and click anywhere in the geometry area to clear the current selection set.
4. Create a plane normal to the guide curve at the upper endpoint of the line.
5. Invoke the **Sketcher** workbench by selecting **Plane.1** as the sketching plane.
6. Draw the sketch of sweep profile, as shown in Figure 9-62.

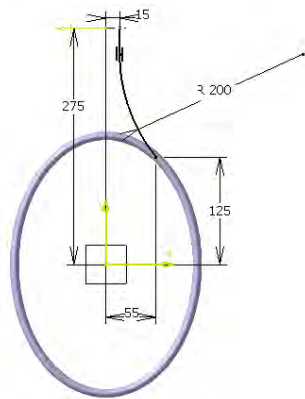


Figure 9-61 The sketch of the guide curve for creating the second sweep feature

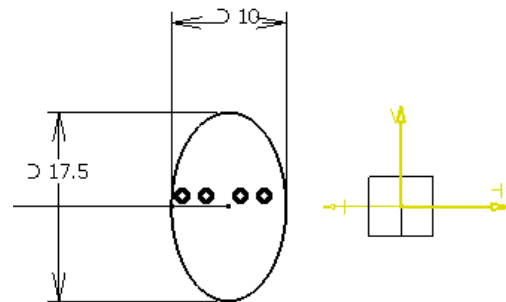


Figure 9-62 The sketch of the sweep profile for creating the second sweep feature

7. Apply the **Coincident** constraint between the center of ellipse and the end point of the guide curve.
8. Exit the **Sketcher** workbench and click anywhere in the geometry area to remove the current selection set.

Creating the Second Sweep Feature

After drawing the sketches for the second sweep feature, you need to create the swept surface.

1. Choose the **Sweep** button from the **Surfaces** toolbar. The **Swept Surface Definition** dialog box is displayed.



2. The **Explicit** button is chosen in the **Profile type** area by default. If not choose it; you are prompted to select a profile.
3. Select the sweep profile from the geometry area; you are prompted to select a guide curve.
4. Select the guide curve from the geometry area and choose the **OK** button from the **Swept Surface Definition** dialog box. The model, after creating the second swept surface, will look similar to the one shown in Figure 9-63.
5. Next, you will create a mirrored copy of second sweep feature. Select **Sweep.2** from the **Specification Tree** and then invoke the **Symmetry** tool by choosing **Insert > Operation > Symmetry** from the menu bar. The **Symmetry Definition** dialog box is displayed and you are prompted to select the reference point, line or plane.
6. Select the YZ plane from the **Specification Tree**. The preview of the Symmetry surface is displayed in the geometry area.
7. Choose the **OK** button from the **Symmetry Definition** dialog box to complete the symmetric feature. Figure 9-64 shows the model with the symmetry feature.

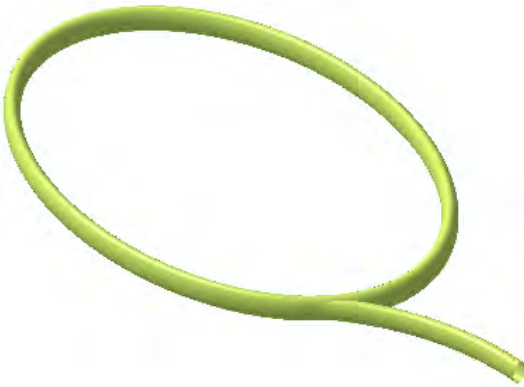


Figure 9-63 The model after creating the second sweep feature

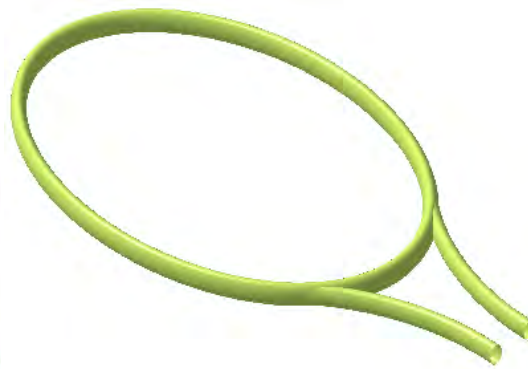


Figure 9-64 The model after creating the symmetry feature



Note

The **Symmetry** tool is discussed in detail in chapter 10.

Creating the Multisection Surface

The next feature you will create is the multisection surface. For creating this surface, you need to draw two sections, as discussed below:

1. Invoke the **Sketcher** workbench by selecting **Plane.1** as the sketching plane.

2. Choose the **Project 3D Elements** button from the **Operation** toolbar and select the elliptical sections of the second sweep and symmetry feature.
3. Complete the sketch, as shown in Figure 9-65, and exit the **Sketcher** workbench.
4. Create a plane at an offset distance of 425 from the ZX plane.
5. Invoke the **Sketcher** workbench using the newly created plane.
6. Draw the sketch for second section, as shown in Figure 9-66. Note that the sketch consists of an ellipse and four points.

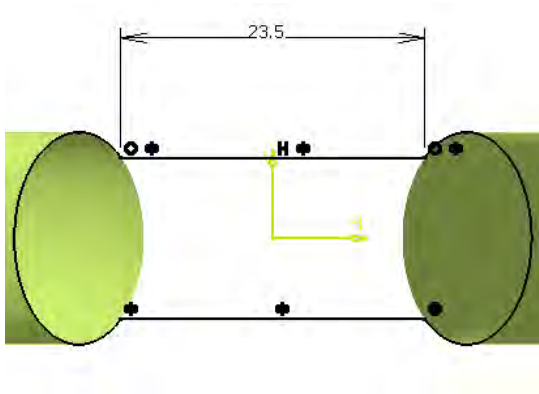


Figure 9-65 The first section for creating the multisection surface

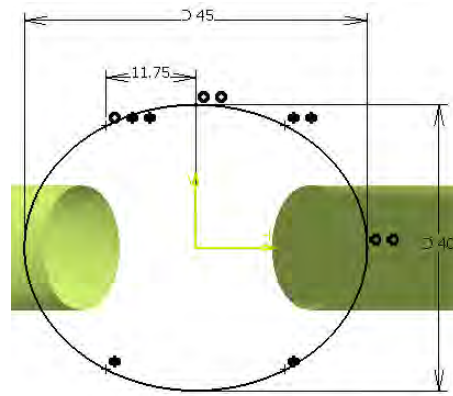


Figure 9-66 The second section for creating the multisection surface

7. Exit the **Sketcher** workbench and click anywhere in the geometry area to remove the current selection set.
8. Choose the **Multisections surface** button from the **Surfaces** toolbar. The **Multi-sections Surface Definition** dialog box is displayed.
9. Select the first section from the geometry area.
10. Next, select the second section from the geometry area; a closing point is created. You need to replace this closing point with another closing point.
11. Move the cursor on the text **Closing Point 2**. Invoke the contextual menu and choose the **Replace** option. Choose the closing point shown in Figure 9-67.
12. Next, choose the **Coupling** tab from the **Multi-sections Surface Definition** dialog box. You are prompted to add, remove or edit coupling, or select point to add coupling.
13. Choose the **Add** button; the **Coupling** dialog box is displayed, and you are prompted to select the coupling point.

14. Select the first coupling point on the first section, refer to Figure 9-66. The selected point is displayed in the **Coupling** dialog box.
15. Select the first coupling point on the second section, refer to Figure 9-66. The coupling created is displayed in the geometry area.
16. Activate the coupling selection area by clicking on it and then choose the **Add** button from the **Multi-sections Surface Definition** dialog box. The **Coupling** dialog box is displayed and you are prompted to select coupling point.
17. Create the second, third, and fourth couplings, refer to Figure 9-66.
18. Choose the **OK** button from the **Multi-sections Surface Definition** dialog box to complete the surface. Figure 9-68 shows the model, after creating the multisection lofted surface and hiding the sketches and the plane.

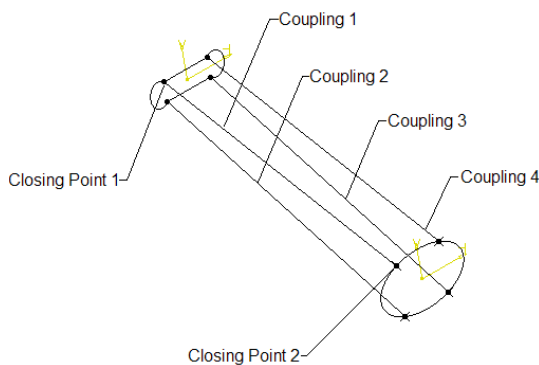


Figure 9-67 The couplings and closing points

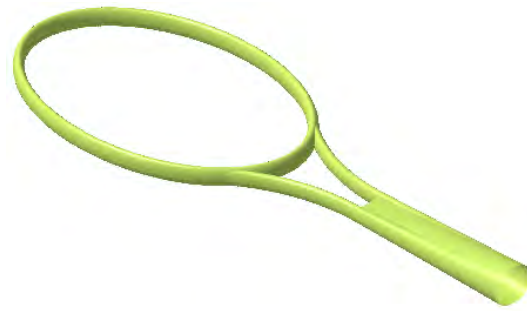



Figure 9-68 The model after creating the multisection surface

Creating the Blended Surface

Next, you need to create a blended surface.

1. Create a plane at an offset distance of 550 from the ZX plane and invoke the **Sketcher** workbench by selecting the newly created plane.
2. Draw the sketch of the blend section, as shown in Figure 9-69. In this figure, the display of previously created surfaces has been turned off.
3. Exit the **Sketcher** workbench and click in the geometry area to remove the current selection set.
4. Choose the **Blend** button from the **Surfaces** toolbar; the **Blend Definition** dialog box is displayed and you are prompted to the select first support or select second curve. 

5. Select the elliptical sketch drawn for creating the lofted surface.
6. Now, select the sketch of the blend section from the geometry area.
7. Choose the **OK** button from the **Blend Definition** dialog box. The model, with the blend surface will be displayed, as shown in Figure 9-70.

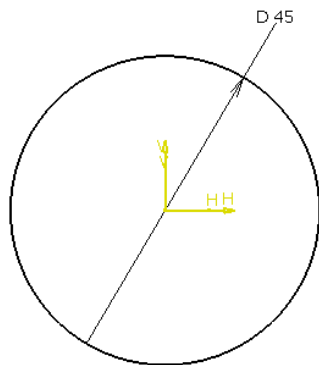



Figure 9-69 The sketch of the blend section



Figure 9-70 The model after creating the blend surface

Creating the Fill Feature

Next, you will create the fill surface to close the open end of the blended feature.

1. Choose the **Fill** button from the **Surfaces** toolbar; the **Fill Surface Definition** dialog box is displayed. 
2. Select **Sketch.7** from the **Specification Tree**.
3. Choose the **OK** button from the **Fill Surfaces Definition** dialog box.
4. Similarly, create another fill surface using **Sketch.5** to cover the open side of the **Multi-section 1 Surface**. You need to split the fill surface using the sweep and symmetry surface. You can apply different colors to different surfaces for better visualization by right-clicking on them and choosing the properties option from the contextual menu. The final model will be, as shown in Figure 9-71.

Saving the File

1. Once the model is complete, you need to save the file. Choose the **Save** button from the **Standard** toolbar. The **Save As** dialog box is displayed.
2. Enter *c09tut2* in the **File name** area. Choose the **Save** button from the **Save as** dialog box.
3. Close the part file by choosing **File > Close** from the menu bar.

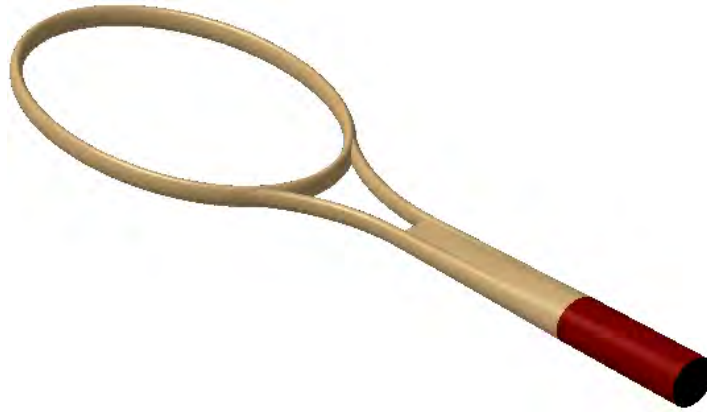


Figure 9-71 The final model

SELF-EVALUATION TEST

Answer the following questions and then compare your answer with those given at the end of this chapter.

1. In CATIA, the _____ tool is provided to extrude a close or open profile up to the defined limits.
2. The _____ tool is used to create a feature by revolving a profile about an axis.
3. The _____ tool is used to create a spherical surface by defining the angular limits.
4. The _____ tool is used to create a cylindrical surface by defining the center point and direction.
5. The _____ tool is used to create a surface by sweeping a profile along a guide curve.
6. The **Multisections surface** tool is used to create a surface using only two sections (T/F).
7. The **Blend** surface tool is used to create a surface using only two sections (T/F).
8. The **Join** tool is used to join two surfaces into one surface (T/F).
9. The **Split** tool is used to split a surface or a curve using a cutting element (T/F).
10. The **Trim** tool is used to trim only the surface element (T/F).

REVIEW QUESTIONS

Answer the following questions.

1. The _____ tool is used to create the wireframe element in the helix shape.
2. The _____ tool is use to create the fill surface.
3. The _____ tool is used to create a circular surface by defining the three guide curves.
4. The _____ is used to create a surface by offsetting the reference surface.
5. The _____ check box is selected to invoke the **Repeat object** dialog in the **Offset Surface Definition** dialog box.
6. In CATIA, which tool is used to create the offset surface?
 - (a) Extrude
 - (b) Revolve
 - (c) Offset
 - (c) Sweep
7. In the **Swept Surface Definition** dialog box, which of the **Profile type** is used to create a surface using two guide curves?
 - (a) Explicit
 - (b) Line
 - (c) Circle
 - (c) Conic
8. Which of the following tools is used to create a surface by joining two surfaces?
 - (a) Extrude
 - (b) Trim
 - (c) Join
 - (c) Split
9. Which of the following tools is used to create a blended surface between the two curves.
 - (a) Revolve
 - (b) Multi-section Surface
 - (c) Blend
 - (c) Split
10. Which of the following tools is used to trim two surfaces with respect to each other?
 - (a) Circle
 - (b) Trim
 - (c) Offset
 - (c) Split

EXERCISE

Exercise 1

In this exercise, you will create the surface model shown in Figure 9-72. The orthographic views of the surface model are shown in Figure 9-73. **(Expected time: 30 min)**

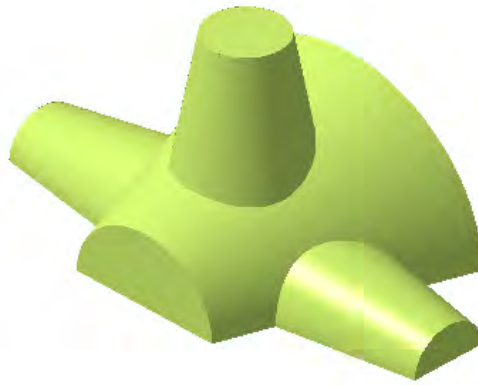


Figure 9-72 The isometric view of the model

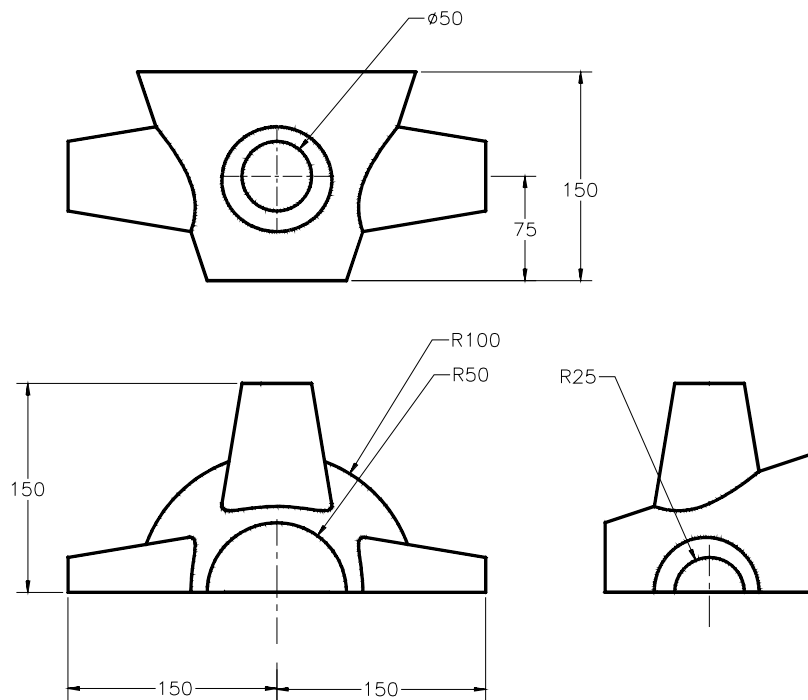


Figure 9-73 The view of the model

Exercise 2

In this exercise, you will create the surface model shown in Figure 9-74. The orthographic views of the surface model are shown in Figure 9-75. (Expected time: 45 min)



Figure 9-74 The isometric view of the model

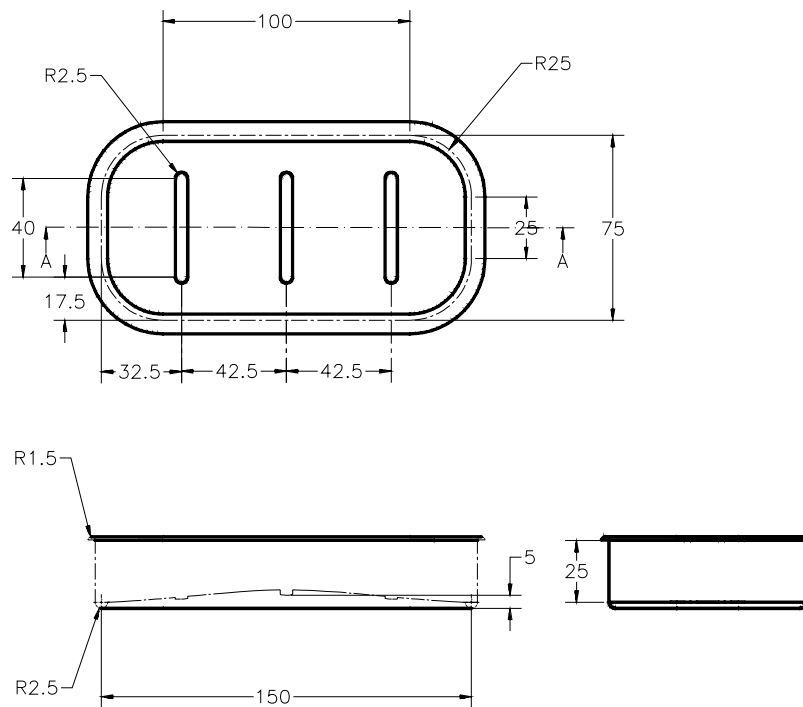


Figure 9-75 The view of the model

Answers to Self-Evaluation Test

1. Extrude, 2. Revolve, 3. Sphere, 4. Cylinder, 5. Sweep, 6. F, 7. T, 8. T, 9. T, 10. F