



# ***Chapter 10***

---

## ***Working with the 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 multisections surfaces.*
- *Create blended surfaces.*
- *Split surfaces.*
- *Trim surfaces.*
- *Join surfaces.*

## NEED FOR SURFACE MODELING

The product and industrial designers give special importance to product styling and providing a unique shape to components. Generally, this is done to make the product look attractive and presentable. Most of the times, the product's shape is managed using the surface modeling techniques. Surface models are three-dimensional models with no thickness and 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 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 an existing 3D mechanical part design with wireframe and basic surface features.

### Starting the Wireframe and Surface Design Workbench

Start a new session of CATIA V5 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 the sketcher workbench can also be used to create surfaces. The tools for constructing the wireframe geometries are discussed next.

### Creating Circles

<b>Menu:</b>	Insert > Wireframe > Circle
<b>Toolbar:</b>	Circle-Corner > Circle



In the **Wireframe and Surface Design** workbench, the **Circle** tool is used to create arcs and circles. Choose the **Circle** button from the **Wireframe** toolbar; the **Circle Definition** dialog box is displayed, as shown in Figure 10-1. The **Center and radius** option is selected by default in the **Circle type** drop-down list. Therefore, you are prompted to select the center point. You can select a predefined point or create a new one by choosing any one of the options from the contextual menu, which is available when you right-click in 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 and then specify the required radius value in the **Radius** spinner. You can set the angular limits of the arc from the **Circle Limitation** area. Finally, choose the **OK** button to complete the arc.

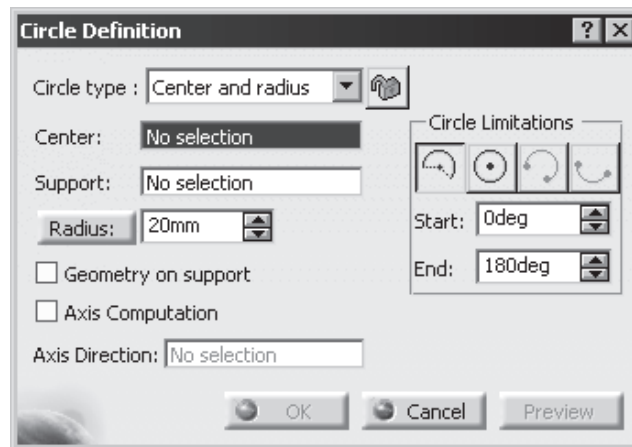


Figure 10-1 The Circle Definition dialog box

The other tools such as **Corner** and **Connect Curve** were discussed in 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 10-2, and then choose the spline button.

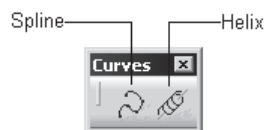


Figure 10-2 The Curves toolbar

The **Spline Definition** dialog box, shown in Figure 10-3, is displayed and you are prompted to select a point. You can select a predefined point or create a new one using the options from the contextual menu, which will be displayed when you right-click in the **Points** selection area of the dialog box. On selecting a point, you are further prompted to select a point, direction (line or plane), or curve. You can choose a number of points to draw the spline.

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

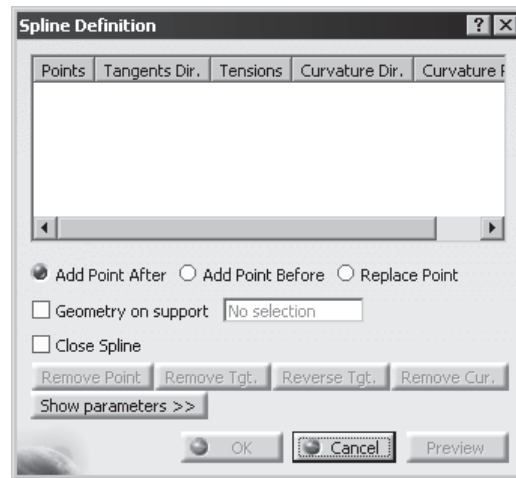


Figure 10-3 The *Spline Definition* dialog box

## Creating a 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 10-4. You are prompted to select the helix starting point. Select a predefined point, or create a new one using the options from the contextual menu, which will be displayed when you right click in the **Starting point** selection area of the **Helix Curve Definition** dialog box.

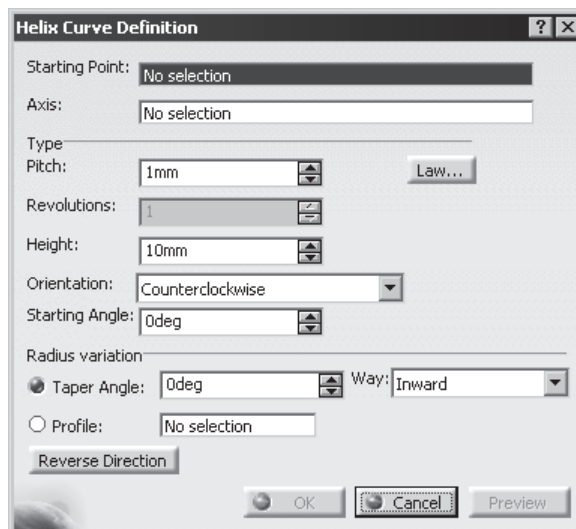
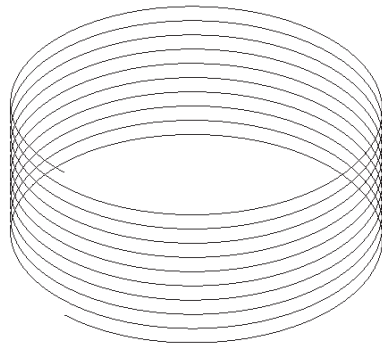
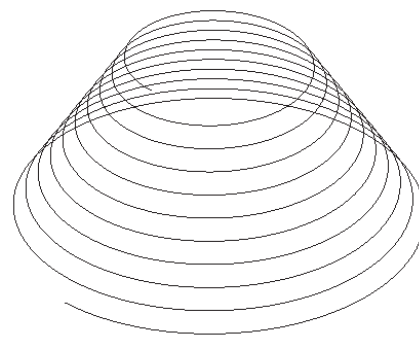


Figure 10-4 The *Helix Curve Definition* dialog box

Next, you are prompted to select a line as the helix axis. Select a predefined line or draw a new one 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 in the **Radius variation** area of the dialog box. Figure 10-5 shows a helix without a taper angle. Figure 10-6 shows a helix with a taper angle.



**Figure 10-5** The helix without specifying the taper angle



**Figure 10-6** The helix with a specified taper angle

## CREATING SURFACES

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

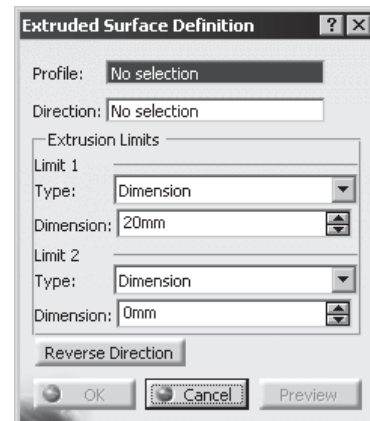
### Creating Extruded Surfaces

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



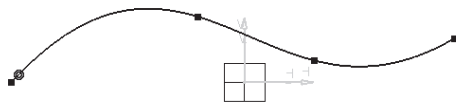
The extruded surfaces are created by extruding a profile and specifying the extrusion depth and direction vector. The basic parameters that are required to create an extruded surface are profile, direction for extrusion, and extrusion limits. To create an extruded surface, you first need to draw the profile to be extruded using the **Sketcher** workbench or the tools 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 10-7.

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



**Figure 10-7** The *Extruded Surface Definition* dialog box

be extruded. Select the profile to see the preview. On drawing the profile using the tools from the **Wireframe** toolbar, you are prompted to specify the direction for extrusion. Specify the 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 **Dimension** spinners. From this release onward, you can also select the **Up-to element** option from the **Type** drop-down list and select a limiting element to specify the termination. Figure 10-8 shows the profile to be extruded and Figure 10-9 shows the resulting extruded surfaces.



**Figure 10-8** The profile to be extruded



**Figure 10-9** 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.



#### Note

While creating a surface, if the result is multiple surface bodies, the **Multi-Result Management** dialog box will be displayed when you exit the creation dialog box. You can select the surface bodies that you need to retain using the options in this dialog box.

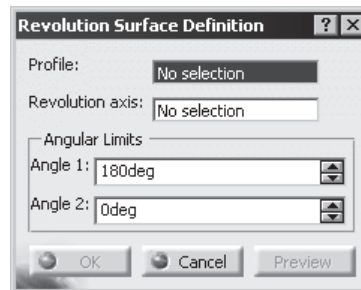
## Creating Revolved Surfaces

**Menu:** Insert > Surfaces > 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 the 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 10-10.

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 10-11 shows a profile and an axis of revolution to create the revolved surface. The resulting surface, revolved through an angle of 180-degree, is shown in Figure 10-12.



**Figure 10-10** The *Revolution Surface Definition* dialog box



**Figure 10-11** The profile and revolution axis



**Figure 10-12** Surface revolved through an angle of 180-degree

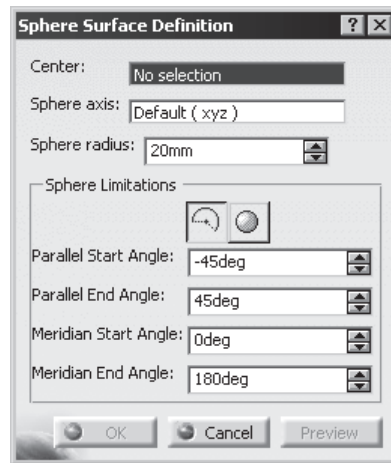
## Creating Spherical Surfaces

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



This tool is used to create spherical surfaces. When you invoke this tool, the **Sphere Surface Definition** dialog box is displayed, as shown in Figure 10-13.

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 vary the angle values using the options in the **Sphere Limitations** area or by directly dragging the limiting arrows in the geometry area. Figure 10-14 shows the spherical surface created by defining the origin as the center. This surface has the default axis system and sphere limitation values.



**Figure 10-13** The *Sphere Surface Definition* dialog box



**Figure 10-14** A spherical surface



**Tip.** You can create a 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 or create a new one 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 10-15, and you are prompted to select a reference surface.

Select the reference surface from the geometry area and specify the offset value in the **Offset** spinner. Choose the **Reverse Direction** button 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 10-16.

In this dialog box, specify the required number of instance(s). Choose the **OK** button to create the offset surfaces. Figure 10-17 shows a reference surface and an offset surface.

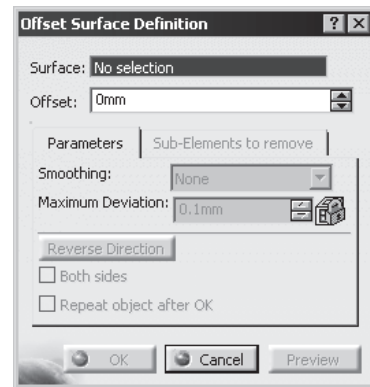


Figure 10-15 The *Offset Surface Definition* dialog box

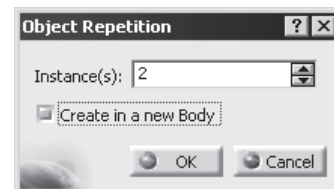


Figure 10-16 The *Object Repetition* dialog box

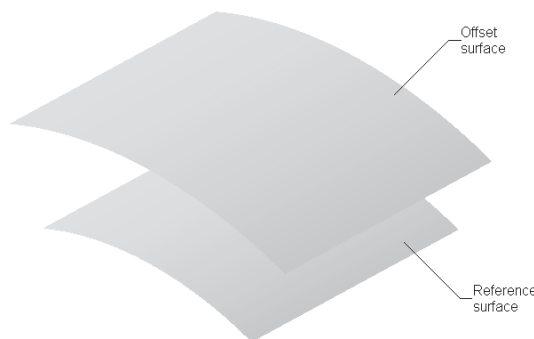


Figure 10-17 An *offset surface*

With this release also, the option of smoothening the complex offset surfaces is provided in the drop-down list. You will notice that by default, the **None** radio option is selected in the **Smoothing** drop-down list. This ensures that uniform smoothening is applied throughout the offset surface. If you select the **Automatic** option, additional smoothening is applied to the areas of the surface that are unable to be offset because of geometric conditions. Therefore, the offset that fails with the **None** option can be completed using the **Automatic** option. While creating an offset surface using the **Automatic** option, sometimes a **Warning** message window is displayed. This window warns that some faces cannot be offset accurately, so a local smoothening is applied. Choose the **Close** button from this dialog box.



#### Note

*Sometimes, 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.*

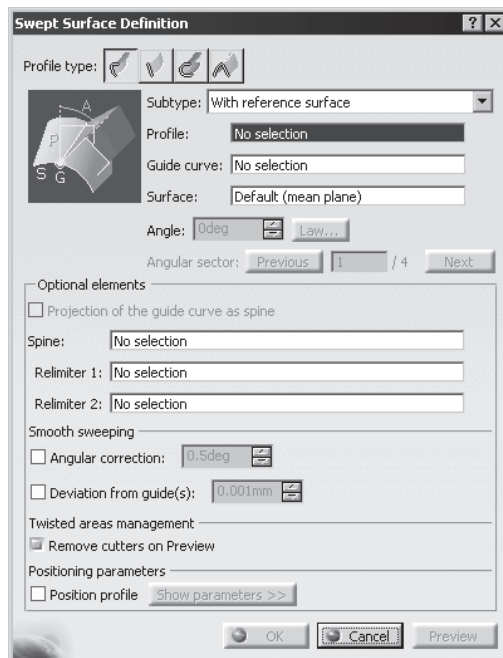
## Creating Swept Surfaces

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



The **Sweep** tool creates 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 10-18, 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 10-19 shows a profile and a guide curve. Figure 10-20 shows the resulting swept surface.

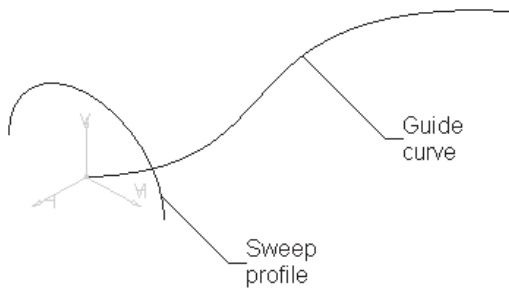
Various other tools to create swept surfaces are discussed in the following section.



**Figure 10-18** The **Swept Surface Definition** dialog box



**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.



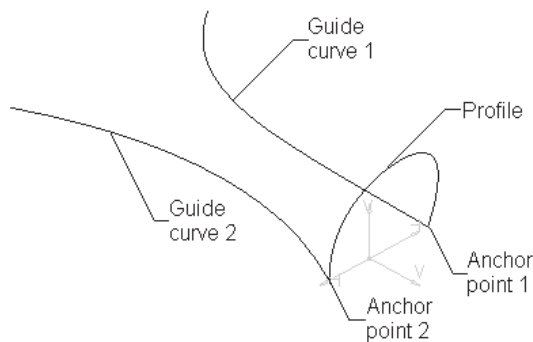
**Figure 10-19** The sweep profile and guide curve



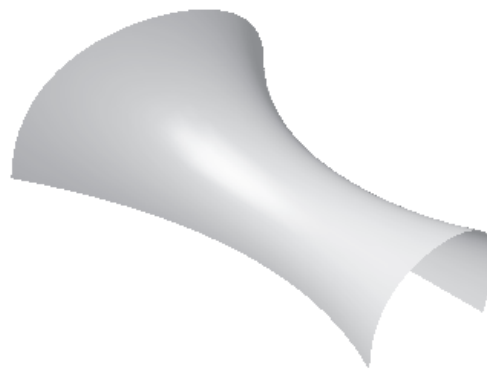
**Figure 10-20** The resulting swept surface

### Swept Surface with Two Guide Curves

You can also create a swept surface using more than one guide curves. To do so, draw the profile and the two guide curves as separate sketches. Now, invoke the **Swept Surface Definition** dialog box. Select the **With two guide curves** option from the **Subtype** drop-down list; you will be prompted to select a profile. After you select the profile, you will be 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 10-21 shows a profile and guide curves. The swept surface created using the two guide curves is shown in Figure 10-22.



**Figure 10-21** The sweep profile and guide curves



**Figure 10-22** 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 a swept surface with two limits, you need to draw two limit curves. Next, invoke the **Swept Surface Definition** dialog box. Select the **Line** button from the **Profile type** area in the dialog box. The parameters in the **Swept Surface Definition** dialog box change and they appear as shown in Figure 10-23. Also, you are prompted to select the first guide curve. On selecting the guide curve, you will be prompted

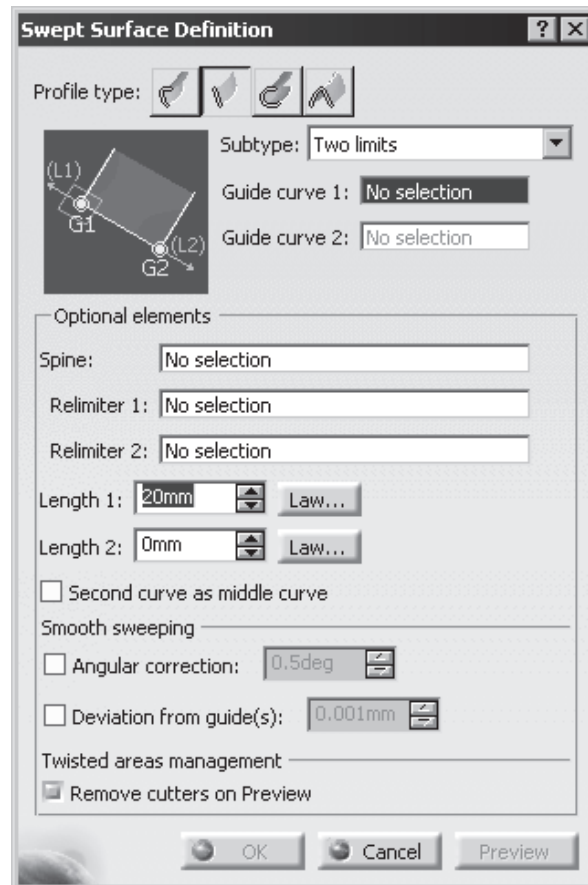


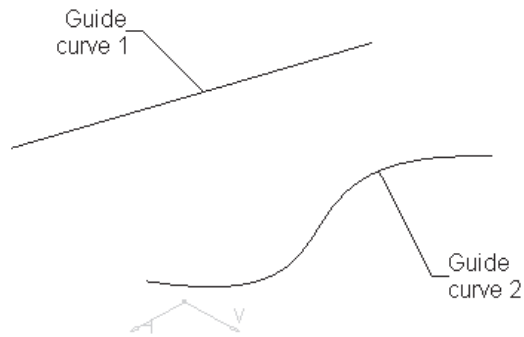
Figure 10-23 The Swept Surface Definition dialog box

to select the 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 defined as the spine. Choose the **OK** button to create the swept surface. Figure 10-24 shows the limit curves and Figure 10-25 shows the resulting surface.

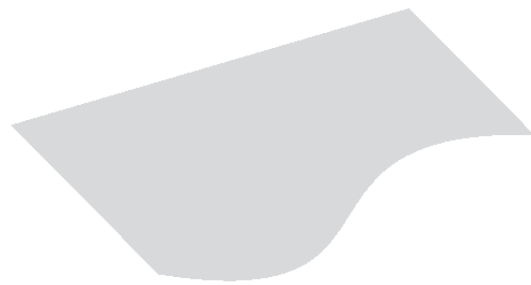
### Swept Surface with Three Curves

You can also create a circular swept surface using three guide curves. To create this type of surface, 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 and choose the **Circle** button from the **Profile type** area. The parameters in the **Swept Surface Definition** dialog box will change, as shown in Figure 10-26. You are prompted to select the first guide curve that will 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 the 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

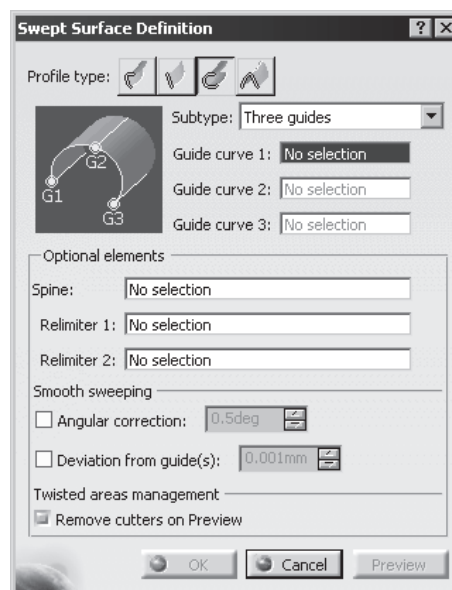
surface. Figure 10-27 shows the guide curves and Figure 10-28 shows the resulting swept surface.



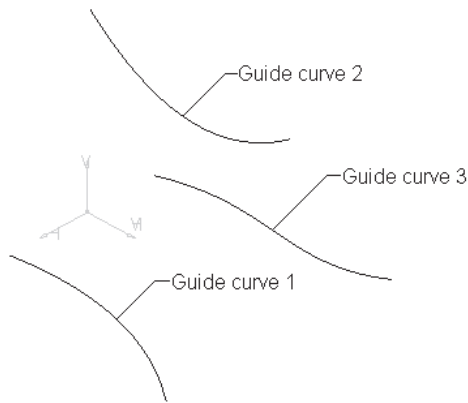
**Figure 10-24** The sweep profile and guide curves



**Figure 10-25** The resulting swept surface



**Figure 10-26** The Swept Surface Definition dialog box



**Figure 10-27** The guide curves



**Figure 10-28** The resulting swept surface

## Creating Fill Surfaces

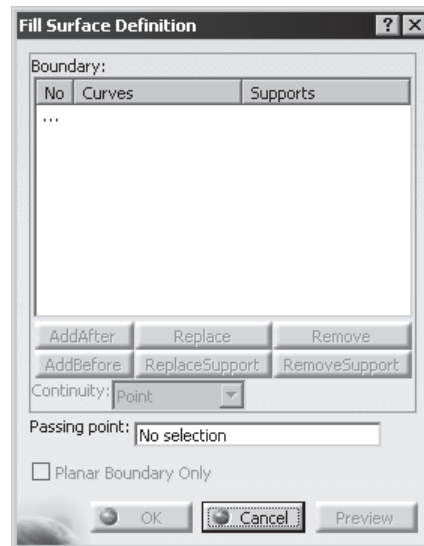
**Menu:** Insert > Surfaces > Fill  
**Toolbar:** Surfaces > Fill



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 the **Fill** button from the **Surface** toolbar. The **Fill Surface Definition** dialog box is displayed, as shown in Figure 10-29.

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 10-30 shows the curves drawn to create the fill surface and Figure 10-31 shows the resulting fill surface.

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



**Figure 10-29** The **Fill Surface Definition** dialog box

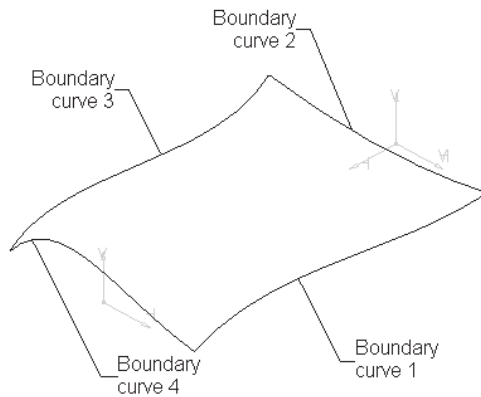


Figure 10-30 The boundary curves



Figure 10-31 The resulting fill surface

## Creating Multisections Surfaces

**Menu:** Insert > Surfaces > Multisections surface  
**Toolbar:** Surfaces > 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 **Multi-sections surface** button from the **Surface** toolbar; the **Multi-sections Surface Definition** dialog box is displayed, as shown in Figure 10-32.

You are prompted to select a curve. Select the first section curve; you are prompted to select a new curve or tangent surface. Select the second section curve from the geometry area and click in 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 multisection surface. Figure 10-33 shows sections and guide curves to create the multisection surface and Figure 10-34 shows the resulting surface.

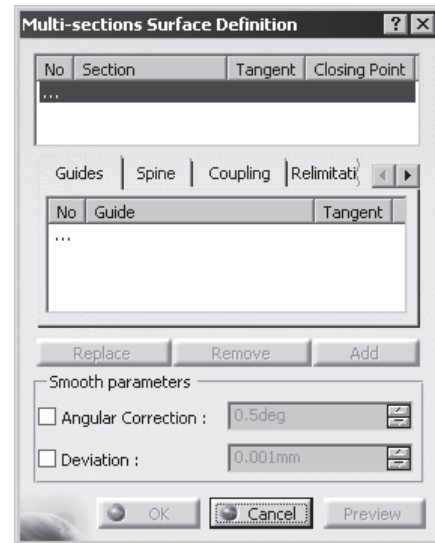


Figure 10-32 The Multi-sections Surface Definition dialog box



### Note

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

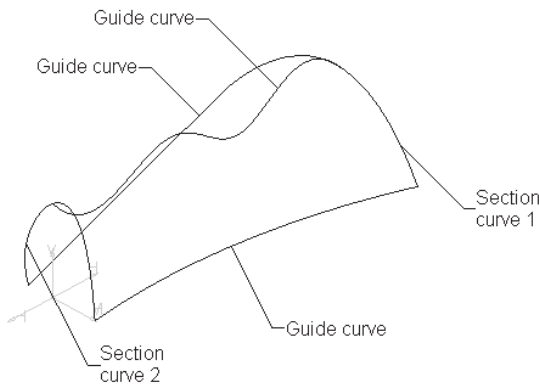


Figure 10-33 Sections and guide curves

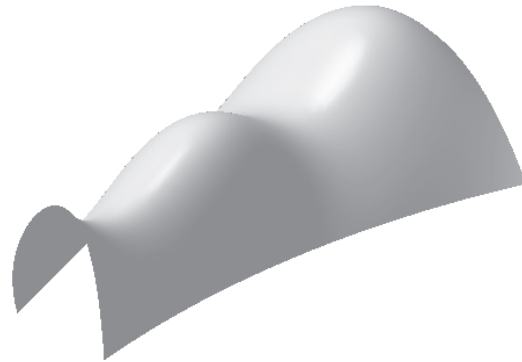


Figure 10-34 The resulting multisections surface

## 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 edges of existing surfaces. If you select support surfaces with curves, the resulting blended surface will be tangent to the support surfaces. To create a blended 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 10-35.

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 and choose the **OK** button from the **Blend Definition** dialog box. Figure 10-36 shows the curves and support surfaces to create the blended surface and Figure 10-37 shows the resulting blended surface.

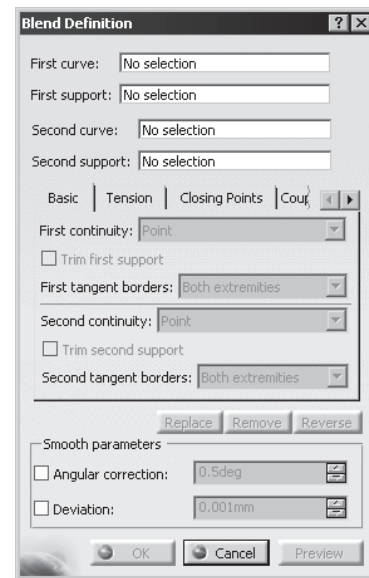
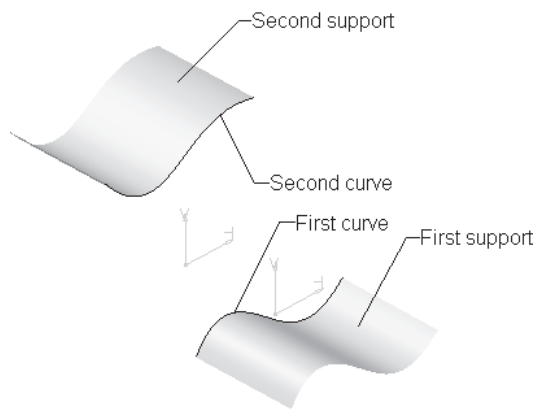


Figure 10-35 The Blend Definition dialog box

## OPERATION ON THE 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 V5 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.





**Figure 10-36** The curves and support surfaces



**Figure 10-37** The resulting blended surface

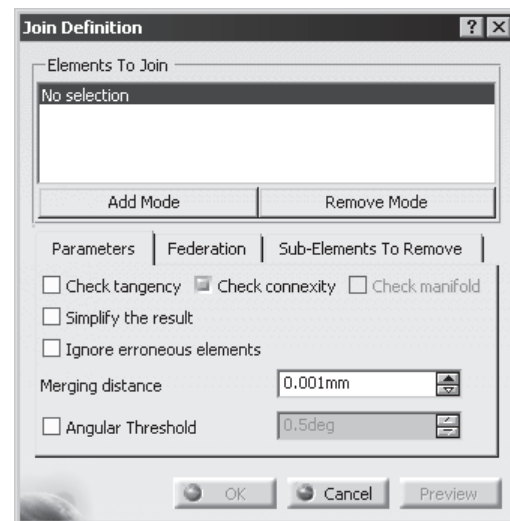
## Joining Surfaces

**Menu:** Insert > Operations > Join  
**Toolbar:** Operations > Join-Healing > Join

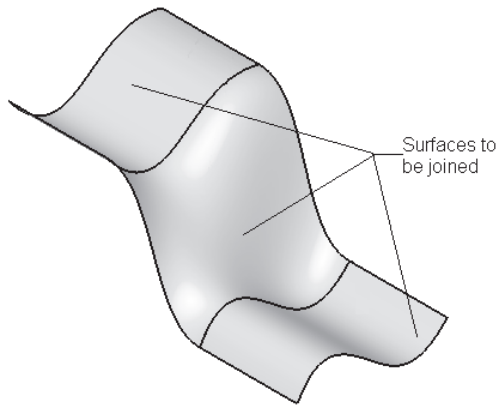


Generally, most of the surface models comprise of various individual surfaces that are connected to each other. To utilize the surface model for creating a solid model, you first need to join all individual surfaces to form a single surface. You can also join individual curves that are connected to each other to form a single curve using this tool. You will learn more about curves in the later chapters. When you invoke this tool, the **Join Definition** dialog box is displayed, as shown in Figure 10-38.

Select the surfaces to be joined from the geometry area; their names will be displayed in the **Elements To Join** selection area. Keep the other default settings as it is and choose the **OK** button. The resulting surface will be formed by joining all the selected surfaces. Figure 10-39 shows the surfaces to be joined. Figure 10-40 shows the resulting single surface, after joining.



**Figure 10-38** The **Join Definition** dialog box



*Figure 10-39 Surfaces to be joined*



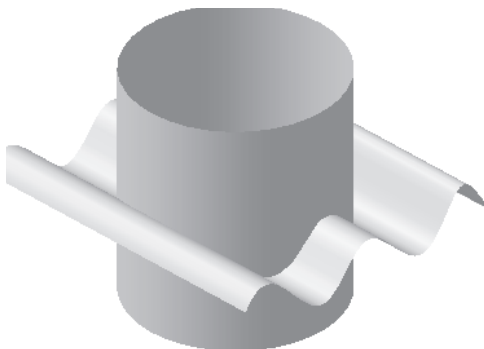
*Figure 10-40 Resulting joint surface*

## Splitting Surfaces

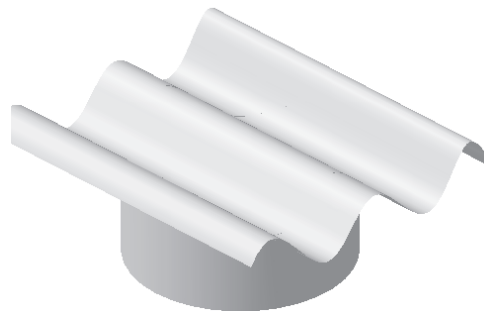
**Menu:** Insert > Operations > Split  
**Toolbar:** Operations > Split



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 wireframe element. To understand the concept of this tool, consider a case in which two intersecting surfaces are created, as shown in Figure 10-41. 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 be 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 a transparent side. You can choose the **Other side** button 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 10-42 shows the split surface.



*Figure 10-41 The split surface and the cutting surfaces*



*Figure 10-42 The resulting split surface*

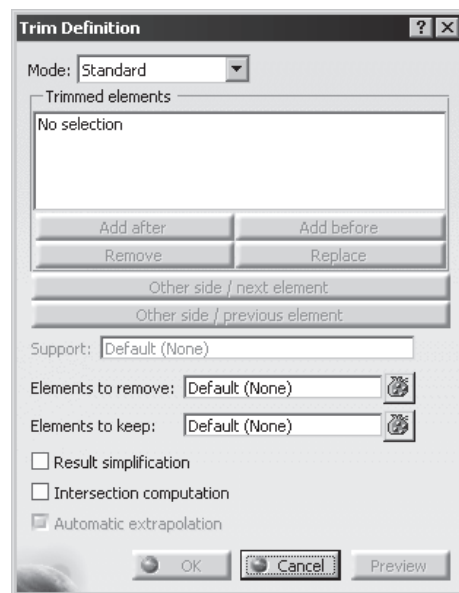
With this release, this tool is enhanced and now you can select more than one surface or wireframe elements to be split. For a multiple selection, you need to choose the **Filter Selection** button provided on the right of the **Element to cut** selection area. On doing so, the **Elements to cut** dialog box is displayed. Select multiple elements that you need to split. After selecting multiple entities, click once in the **Cutting elements** selection area and select the cutting elements. Now, choose the **OK** button from the **Split Definition** dialog box.

## Trimming Surfaces

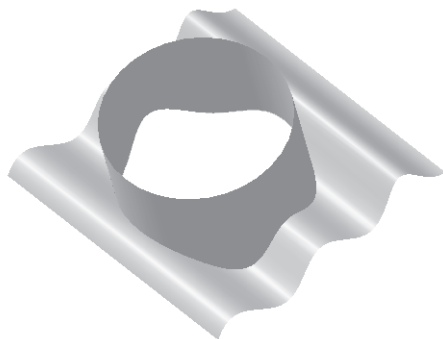
**Menu:** Insert > Operations > Trim  
**Toolbar:** Operations > Split-Trim > Trim



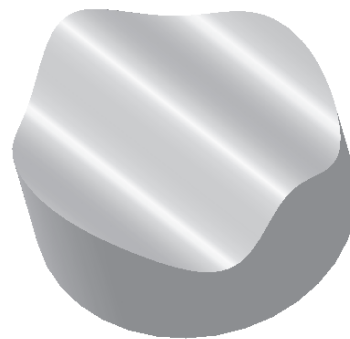
The **Trim** tool allows you to trim two intersecting surfaces or curves with respect to each other. When you invoke this tool, the **Trim Definition** dialog box will be displayed, as shown in Figure 10-43. Select a surface as the first element and the other surface as the second element. One side of each surface will be displayed as transparent. You can choose the **Other side / next element** and **Other side / previous element** buttons to reverse the sides of the surfaces to be trimmed. Figure 10-44 shows surfaces trimmed using the **Trim** tool. Figure 10-45 shows the other sides trimmed.



**Figure 10-43** The **Trim Definition** dialog box



**Figure 10-44** The trimmed surfaces



**Figure 10-45** The trimmed surface

## TUTORIALS

### Tutorial 1

In this tutorial, you will create the model shown in Figure 10-46. Its views and dimensions are shown in Figure 10-47. **(Expected time: 45 min)**

The following steps are required to complete this tutorial:

- a. Start CATIA V5 and start a new file in the **Wireframe and Surface Design** workbench.
- b. Draw the sketches for the multisections surface, refer to Figures 10-48 through 10-50.
- c. Create the multisections surface, refer to Figures 10-51 and 10-52.
- d. Draw the sketch to create the revolve surface, refer to Figure 10-53.
- e. Create the revolved surface, refer to Figure 10-54.
- f. Draw the sketch to create the sweep profile, refer to Figures 10-55 and 10-56.
- g. Create the swept surface, refer to Figure 10-57.
- h. Split the swept surface with the revolved surface.



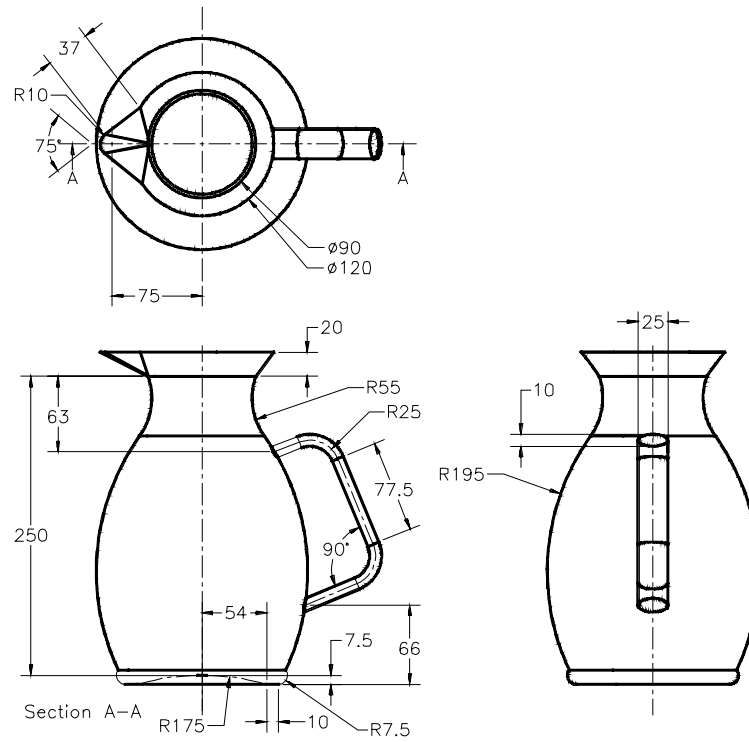
*Figure 10-46 The isometric view of the model*

### Starting a New Part File

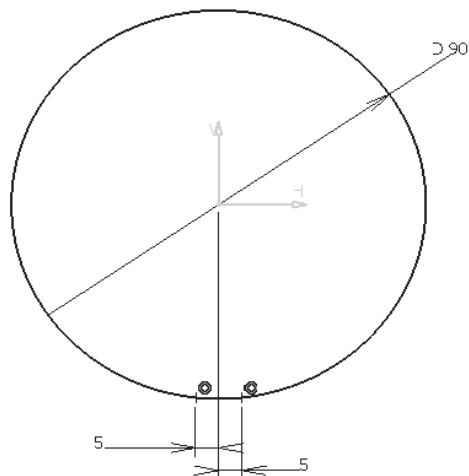
1. Start CATIA V5 and choose **Close** from the **File** menu. The start screen of CATIA V5 is displayed. Choose **Start > Mechanical Design > Wireframe and Surface Design** from the menu bar. Choose **OK** from the **New Part** dialog box to start a new file in **Wireframe and Surface Design** workbench

### Drawing the Sketch for the Base Surface

1. Invoke the **Sketcher** workbench by selecting the xy plane as the sketching plane and draw a circle with the center at the origin and diameter 90.
2. Place two points on the circle, as shown in Figure 10-48. These points will be used to create the guide curve and define the closing point for creating the loft surface later in this tutorial.

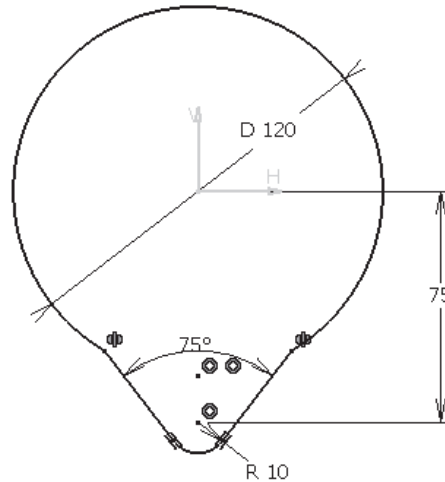


**Figure 10-47** Views and dimensions of the model



**Figure 10-48** First section for the base surface

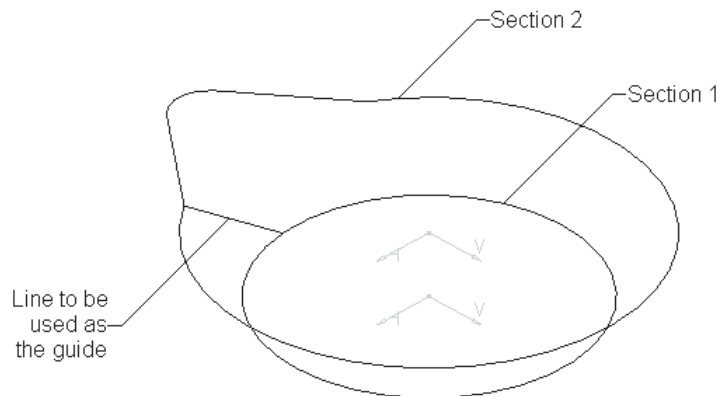
3. Exit the **Sketcher** workbench. Choose **Insert > Wireframe > Plane** to create a plane at an offset of 20 from the xy plane.
4. Invoke the **Sketcher** workbench by selecting **Plane.1** as the sketching plane.
5. Draw the sketch, as shown in Figure 10-49, and exit the **Sketcher** workbench.



**Figure 10-49** The second section for the base surface

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


6. 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 Figure 10-50.

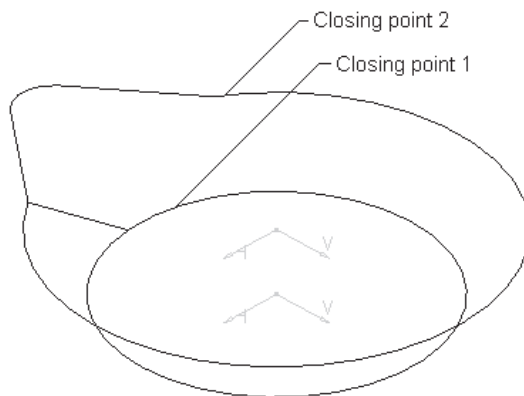


**Figure 10-50** The line created between two points on the sections

## 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 10-50. The name of the selected entity is displayed in the selection area of the **Multi-sections Surface Definition** dialog box. By default, a closing point is created. Move the cursor on the text in the **Closing Point** column 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 10-51.
4. 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 10-51.
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 10-50.
7. Choose the **OK** button from this dialog box to complete the multisection lofted surface. Figure 10-52 shows the multisection lofted surface.




**Figure 10-51** The sketch showing the position of the closing points

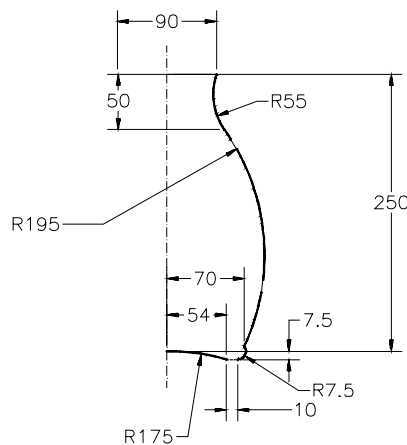


**Figure 10-52** The multisection lofted surface

## Creating the Revolved Surface

Next, you need to create the revolved surface.

1. Invoke the **Sketcher** workbench by selecting the 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 10-53 and exit the **Sketcher** workbench
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 the default angle limits, will be displayed in the geometry area.
6. Set the value of the **Angle 1** spinner to **360** and choose the **OK** button from the **Revolution Surface Definition** dialog box. Figure 10-54 shows the model, after creating the revolved surface and hiding the sketch.



**Figure 10-53** The axis and profile to be revolved



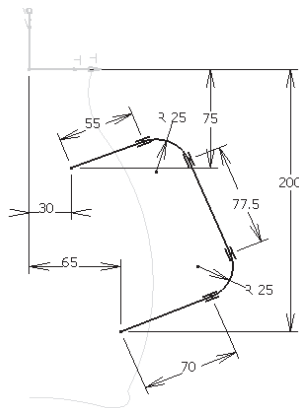


*Figure 10-54 The model after creating the revolved surface*

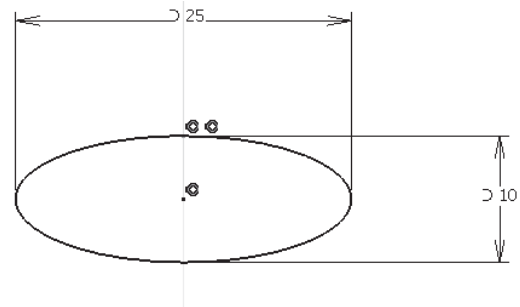
### Creating the Swept Surface

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 the yz plane as the sketching plane.
2. Draw the profile of the guides, as shown in Figure 10-55.
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 10-56.



*Figure 10-55 The guide for the swept surface*




*Figure 10-56 The profile for the swept surface*

7. Choose the **Isometric View** button from the **View** toolbar.
8. Press and hold the CTRL key, 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.
10. Choose the **Sweep** button from the **Surface** toolbar; the **Swept Surface Definition** dialog box is displayed. Choose the **Explicit** button from the **Profile type** area; you are prompted to select a profile. 
11. Select the 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 be split. 
2. Select the swept surface from the geometry area.
3. Now, select the revolved 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 10-57.

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

### Saving the File

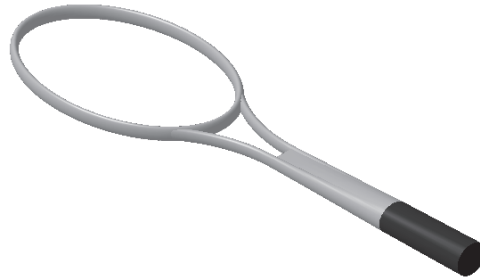
1. Choose the **Save** button from the **Standard** toolbar; the **Save As** dialog box is displayed.
2. Enter *c10tut1* in the **File name** area. Choose the **Save** button from the **Save as** dialog box.
3. Close the part file by choosing **File > Close** from the menu bar.



*Figure 10-57 The isometric view of the model after splitting the swept surface*

## Tutorial 2

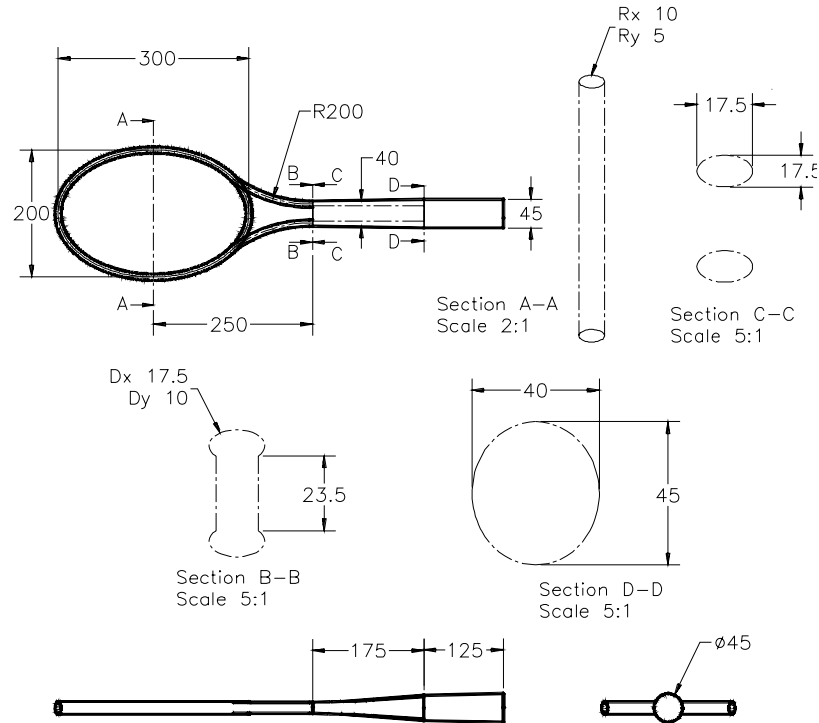
In this tutorial, you will create the model, as shown in Figure 10-58. Its drawing views and dimensions are shown in Figure 10-59. **(Expected time: 45 min)**



*Figure 10-58 The isometric view of the model*

The following steps are required to complete this tutorial:

- Start a new file in the **Wireframe and Surface Design** workbench.
- Create a swept surface as the base feature, refer to Figures 10-60 through 10-62.
- Create the second swept surface, refer to Figures 10-63 through 10-65.
- Create the symmetry of the second swept surface, refer to Figure 10-66.
- Create the multisections surface, refer to Figures 10-67 through 10-70.
- Create the blended surface, refer to Figures 10-71 and 10-72.
- Create the fill surfaces, refer to Figure 10-73.



**Figure 10-59** The views and dimensions of the model

## Drawing the Sketch for the 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 after selecting the xy plane as the sketching plane.
3. Draw an ellipse, as shown in Figure 10-60 and exit the **Sketcher** workbench.
4. Invoke the **Sketcher** workbench after selecting the zx plane as the sketching plane.
5. Draw an ellipse, as shown in Figure 10-61, and exit the **Sketcher** workbench.

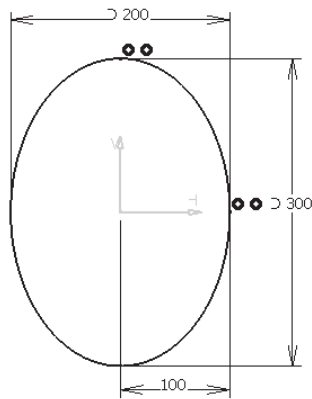


Figure 10-60 The guide curve

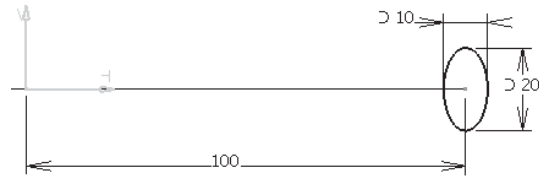


Figure 10-61 The sweep profile

## 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 10-62.

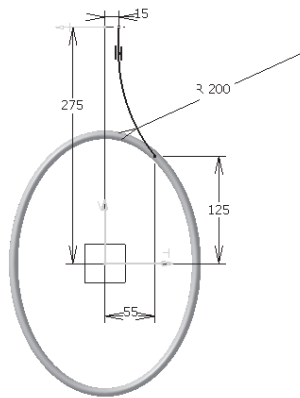


Figure 10-62 The resulting swept surface

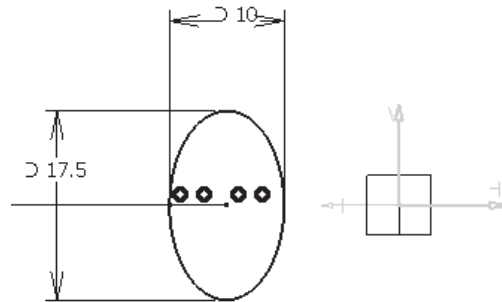
## Drawing Sketches for the Second Sweep Feature

The second feature is also a swept surface. Therefore, a profile and 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 10-63.
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 10-64.



**Figure 10-63** The sketch of the guide curve for creating the second sweep feature



**Figure 10-64** The sketch of the sweep profile for creating the second sweep feature

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

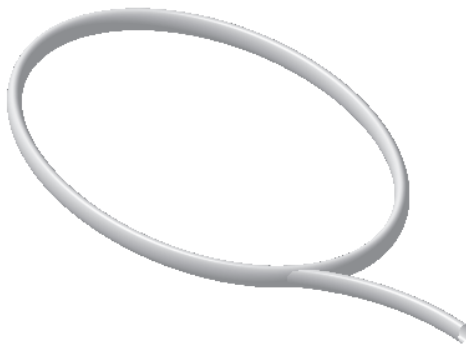
## Creating the Second Sweep Feature

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

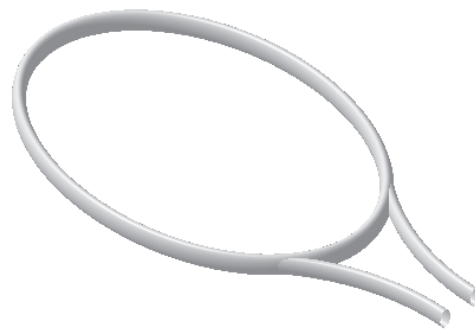
1. Choose the **Sweep** button from the **Surfaces** toolbar; the **Swept Surface Definition** dialog box is displayed.



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 10-65.
5. Next, you will create a mirrored copy of the 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 10-66 shows the model with the symmetry feature.



**Figure 10-65** The model after creating the second sweep feature



**Figure 10-66** The model after creating the symmetry feature



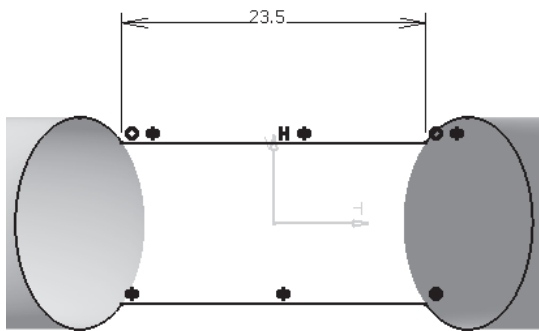
**Note**

The **Symmetry** tool has been discussed in detail in Chapter 9.

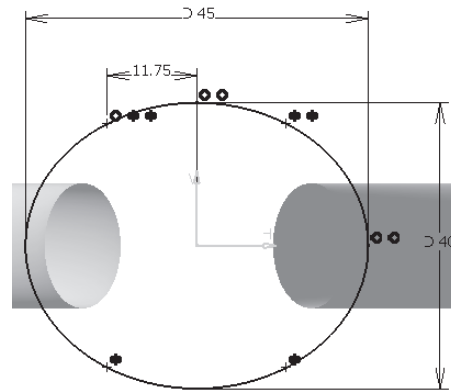
## Creating the Multisection Surface

The next feature to be created 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 10-67, 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 the second section, as shown in Figure 10-68. Note that the sketch consists of an ellipse and four points.



**Figure 10-67** The first section for creating the multisection surface

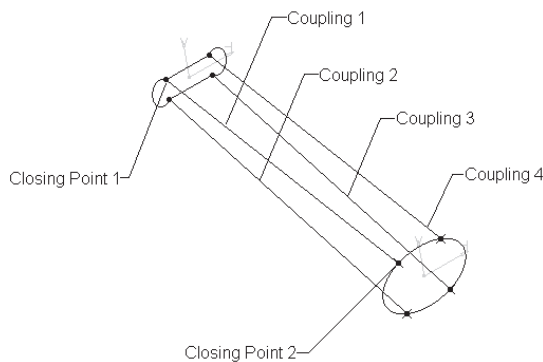


**Figure 10-68** The second section for creating the multisection surface

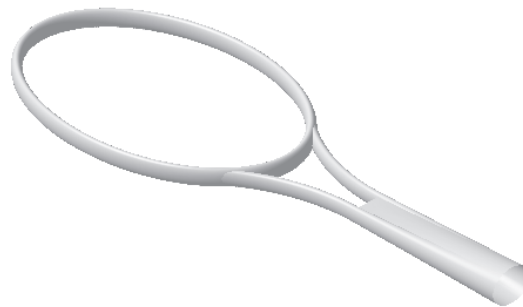
7. Exit the **Sketcher** workbench and click anywhere in the geometry area to remove the current selection set.
8. Choose the **Multisections surface** button from the **Surfaces** toolbar; the **Multi-sections Surface Definition** dialog box is displayed.
9. Select the first section from the geometry 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. Modify the closing points based on Figure 10-69.



12. Next, choose the **Coupling** tab from the **Multi-sections Surface Definition** dialog box. You are prompted to add, remove, or edit the coupling, or select a point to add the 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 10-69. The selected point is displayed in the **Coupling** dialog box.
15. Select the second coupling point on the second section, refer to Figure 10-69. The coupling created is displayed in the geometry area.
16. Activate the coupling selection area by clicking in 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 a coupling point.
17. Create the second, third, and fourth couplings, refer to Figure 10-69.
18. Choose the **OK** button from the **Multi-sections Surface Definition** dialog box to complete the surface. Figure 10-70 shows the model, after creating the **Multi-sections Surface** and hiding the sketches and the plane.



**Figure 10-69** The couplings and closing points




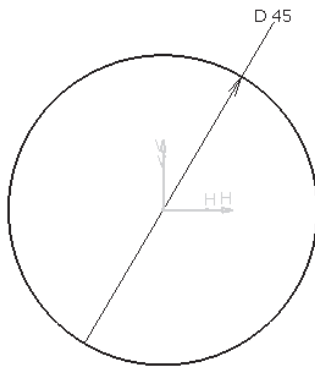
**Figure 10-70** The model after creating the multisection surface

## Creating the Blended Surface

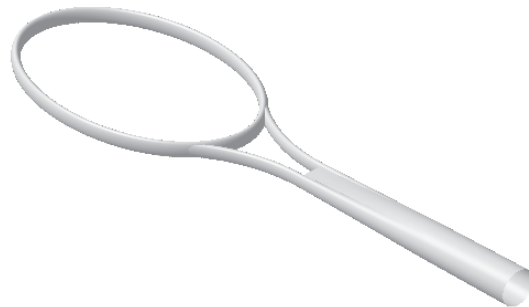
Next, you need to create a blended surface.

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 blended section, as shown in Figure 10-71. In this figure, the display of the 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 select the first support or the second curve. 
5. Select the elliptical sketch drawn for creating the lofted surface.
6. Now, select the sketch of the blended section from the geometry area.
7. Choose the **OK** button from the **Blend Definition** dialog box. The model, with the blended surface will be displayed, as shown in Figure 10-72.




**Figure 10-71** The sketch of the blended section

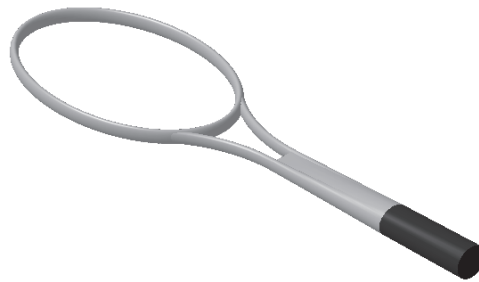


**Figure 10-72** The model after creating the blended surface

## 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 a better visualization by right-clicking on them and choosing the properties option from the contextual menu. The final model will be created, as shown in Figure 10-73.



*Figure 10-73 The final model*

### **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 *c10tut2* 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.

### **SELF-EVALUATION TEST**

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

1. In CATIA V5, the \_\_\_\_\_ tool is provided to extrude a closed or an 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** 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

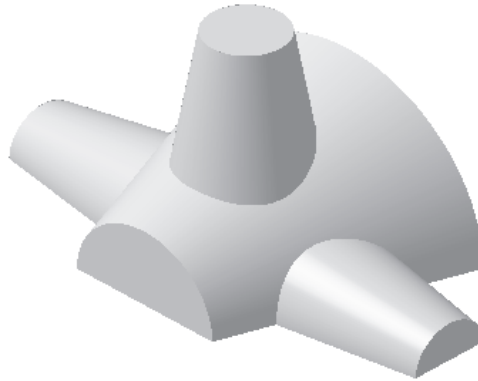
Answer the following questions:

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 \_\_\_\_\_ tool 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 V5, which tool is used to create the offset surface?
  - (a) **Extrude**
  - (b) **Revolve**
  - (c) **Offset**
  - (d) **Sweep**
7. In the **Swept Surface Definition** dialog box, which of the **Profile** types is used to create a surface using two guide curves?
  - (a) **Explicit**
  - (b) **Line**
  - (c) **Circle**
  - (d) **Conic**
8. Which of the following tools is used to create a surface by joining two surfaces?
  - (a) **Extrude**
  - (b) **Trim**
  - (c) **Join**
  - (d) **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**
  - (d) **Split**
10. Which of the following tools is used to trim two surfaces with respect to each other?
  - (a) **Circle**
  - (b) **Trim**
  - (c) **Offset**
  - (d) **Split**

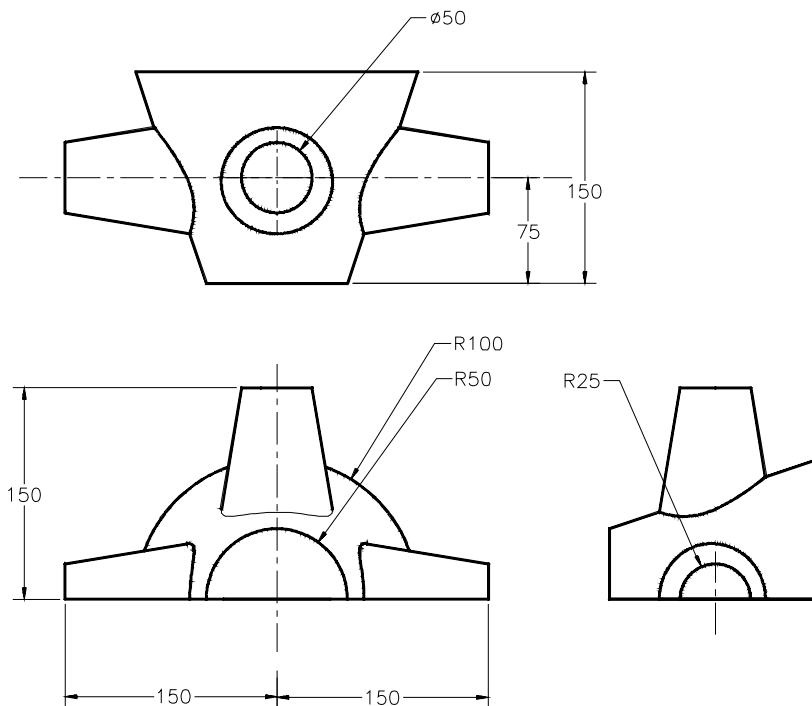
## EXERCISES

### Exercise 1

In this exercise, you will create the surface model shown in Figure 10-74. Its orthographic views are shown in Figure 10-75. (Expected time: 30 min)



*Figure 10-74 The isometric view of the model*



*Figure 10-75 Views and dimensions of the model*

## Exercise 2

In this exercise, you will create the surface model shown in Figure 10-76. Its orthographic views are shown in Figure 10-77. **(Expected time: 45 min)**



Figure 10-76 The isometric view of the model

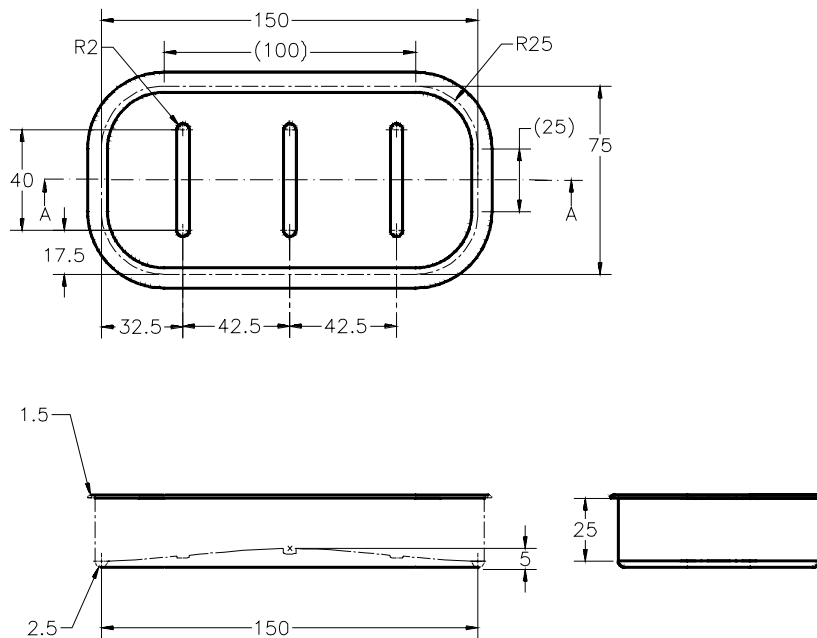


Figure 10-77 Views and dimensions of the model

### 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