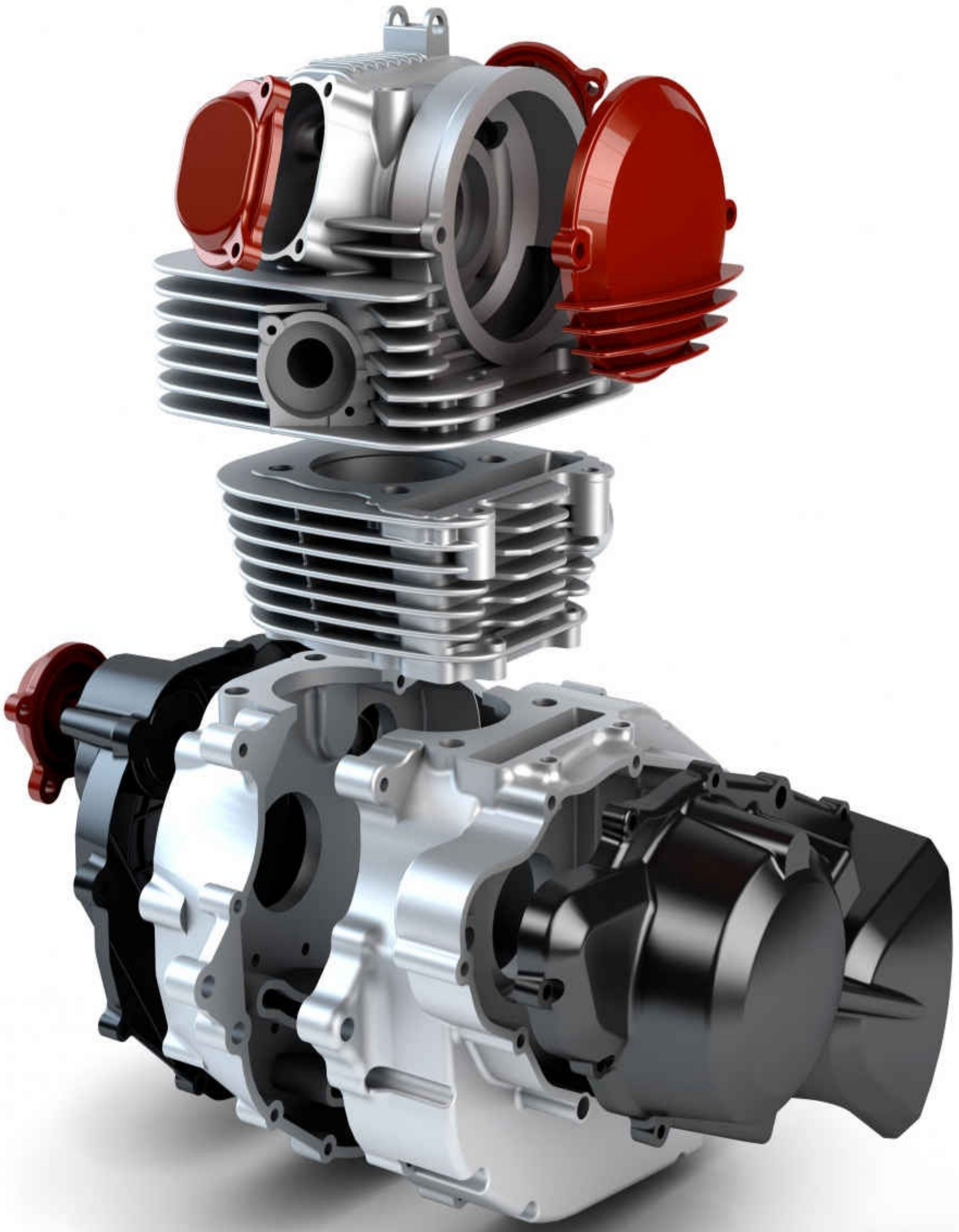
CATIA V5-6R2015 Basics - Part II

Part Modeling



Tutorial Books

CATIA V5-6R2015
Basics – Part II

Tutorial Books

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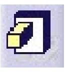
Contents

Chapter 3: Basic Sketch Based Features

Sketch-Based features are used to create basic and simple parts. Most of the times, they form the base for complex parts as well. These features are easy to create and require a single sketch. Now, you will learn the commands to create these features.

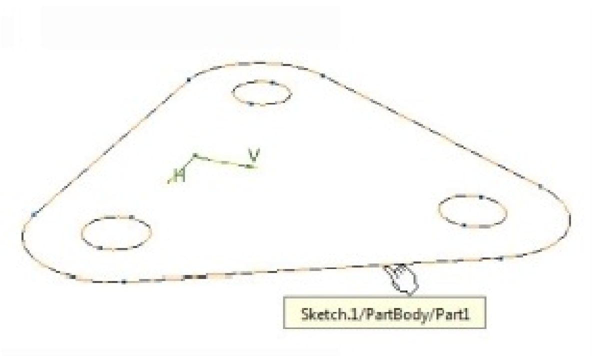
The topics covered in this chapter are:

- *Pad Features*
- *Shaft Feature*
- *Pocket Features*
- *Reference Planes*
- *More Options in the Pad and Shaft commands*
- *View commands*

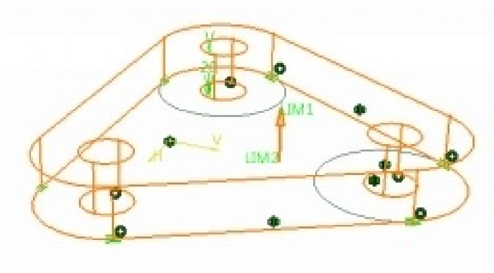
 Pad

Pad is the process of taking a two-dimensional profile and converting it into 3D feature by giving it some thickness. A simple example of this would be taking a circle and converting it into a cylinder.

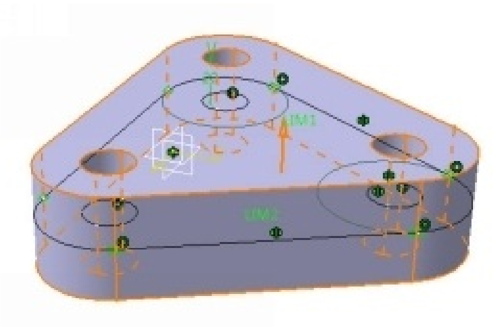
1. Once you have created a sketch profile or profiles you want to *Pad*, activate the **Pad** command (On the **Sketch-Based Features** toolbar, click **Pads** drop-down > **Pad** (or) click **Insert > Sketch-Based Features > Pad** on the Menu).
2. Click on the sketch profile to add thickness to it.



3. On the **Pad Definition** dialog, type-in a value in the **Length** box.

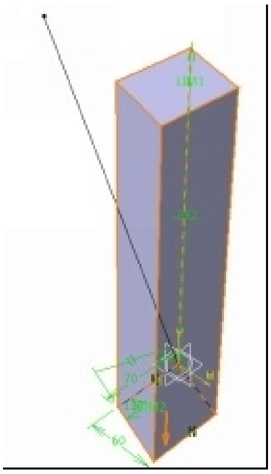


4. If you want to add equal thickness on both sides of the sketch, then check the **Mirror extent** option.
5. Click the **Preview** button to view how the model would look when completed.

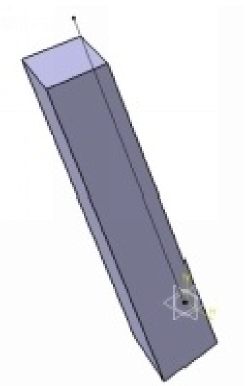


6. Click **OK** to complete the *Pad* feature.

While creating a *Pad* feature, CATIA adds material in the direction normal to the sketch.

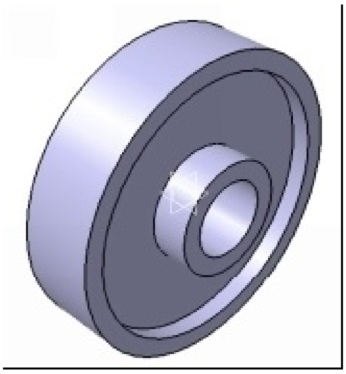


If you want to manually define the direction in which the material will be added, then click the **More** button on the **Pad Definition** dialog. Click in the **Reference** selection box and select a line.

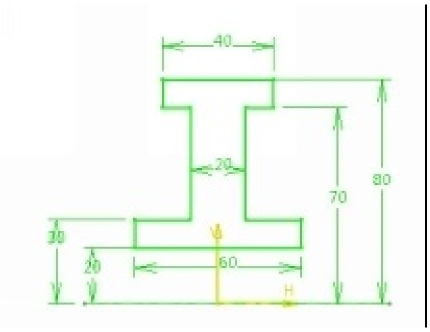


 Shaft

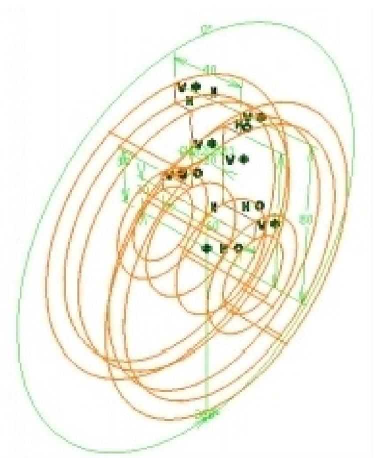
Revolving is the process of taking a two-dimensional profile and revolving it about a centerline to create a 3D geometry (shapes that are axially symmetric). While creating a sketch for the *Shaft* feature, it is important to think about the cross-sectional shape that will define the 3D geometry once it is revolved about an axis. For instance, the following geometry has a hole in the center.



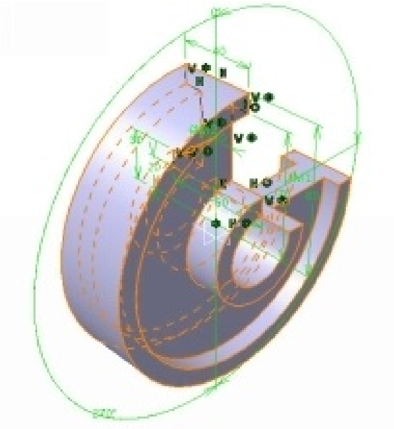
This could be created with a separate *Pocket* or *Hole* feature. But in order to make that hole part of the *Shaft* feature, you need to sketch the axis of revolution so that it leaves a space between the profile and the axis.



1. After completing the sketch, activate the **Shaft** command (On the **Profile** toolbar, click the Shaft icon (or) click **Insert > Sketch-Based Features > Shaft** on the Menu).
2. The sketch will be revolved by full 360 degrees.



3. If you want to enter an angle of revolution, type-in a value in the **First Angle** box.

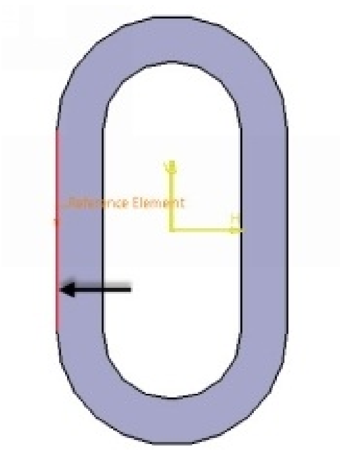


4. On the dialog, click **OK** to complete the *Shaft* feature.

 Project 3D Elements

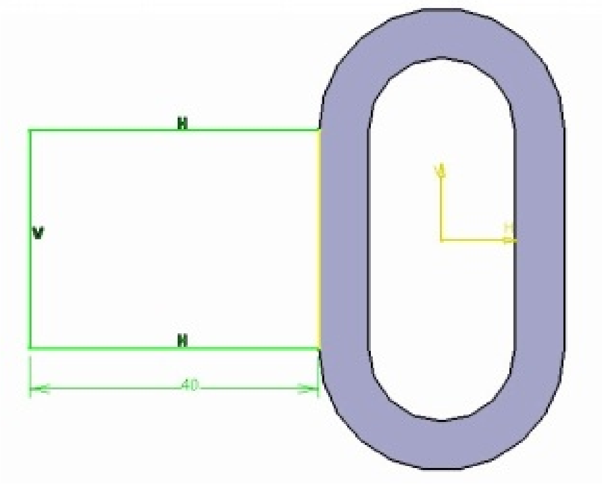
This command projects the edges of a 3D geometry onto a sketch plane.

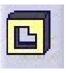
1. Activate the Sketcher Workbench by selecting a plane or model face.
2. On the **Operations** toolbar, click **3D Geometry** drop-down **> Project 3D Elements** (or) click I**nsert > Operations > 3D Geometry > Project 3D Elements** on the Menu.
3. Click on the edges of the model geometry to project them on to the sketch plane.
4. Click **OK** on the **Projection** dialog.



The projected element will be yellow in color and fully constrained. If you want to convert it into a normal sketch element, then right click on it and select **Mark.object > Isolate**.

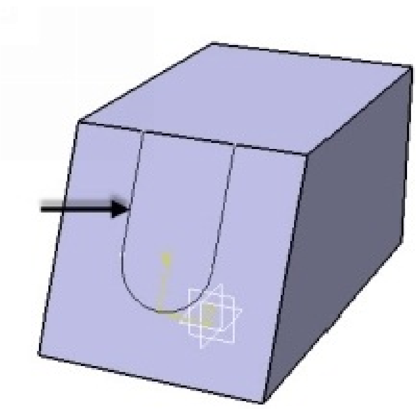
5. Complete the sketch and exit the workbench.



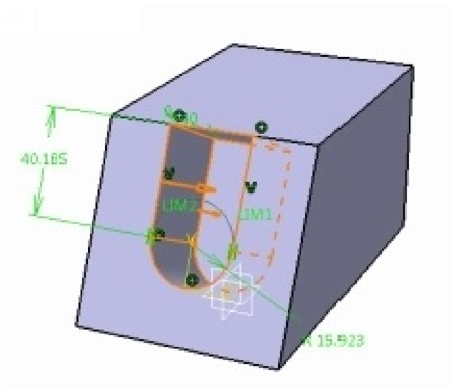
 The Pocket command

This command removes material from the geometry by extruding a sketch. It functions on the same lines of the **Pad** command.

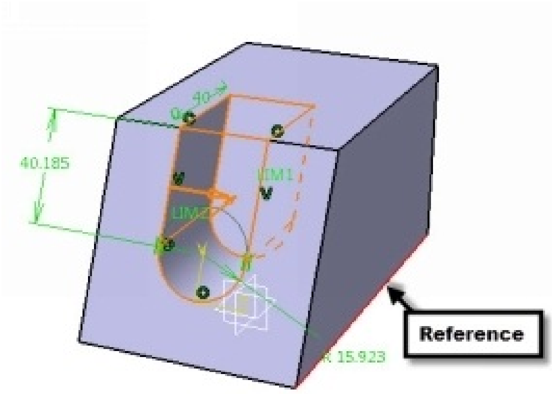
1. Draw a sketch on a plane or a model face.
2. On the **Sketch-Based Features** toolbar, click the **Pocket** icon (or) click **Insert > Sketch-Based Features > Pocket** on the Menu.
3. Select the sketch.



4. On the **Pocket Definition** dialog, type-in a value in the **Depth** box and click **Preview**.



5. If you want to define the direction of material removal, then the **More** button to expand the dialog.
6. Click in the **Reference** selection box and select an edge or line to define the direction.

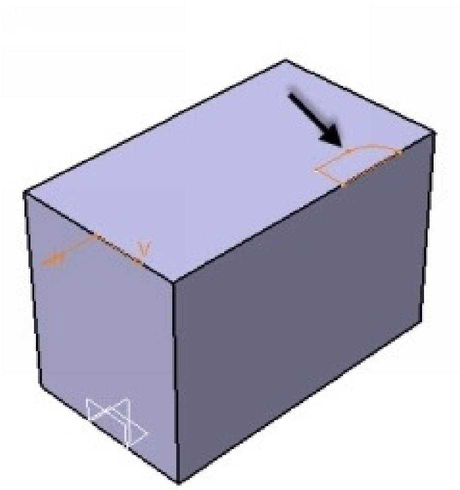


7. Click **OK** to complete the pocket feature.

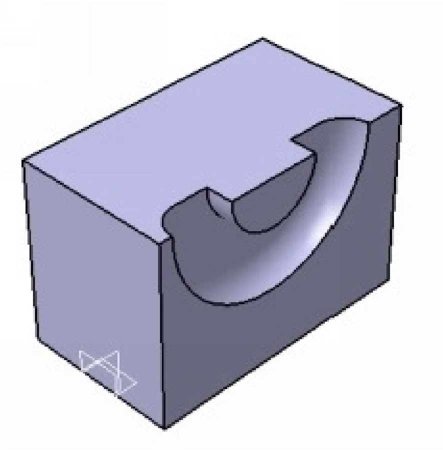
 The Groove command

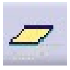
This command removes material from the geometry by revolving a sketch about an axis. It functions in a way similar to the **Shaft** command.

1. Draw a sketch on a plane or a model face. Also, draw a centerline using the **Centerline** command.
2. On the **Sketch-Based Features** toolbar, click the **Groove** icon (or) click **Insert > Sketch-Based Features > Groove** on the Menu.
3. Select the sketch. If you have created the centerline, then groove will be created automatically.



4. On the **Groove Definition** dialog, type-in values in the **First angle** and **Second angle** boxes.
5. Click **OK** to complete the groove feature.



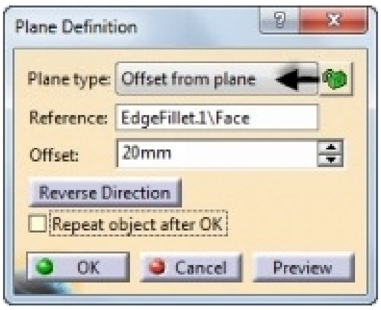
 The Plane command

Each time you start a new part file, CATIA V5 automatically creates default Reference planes. Planes are a specific type of elements in CATIA V5, known as Reference Elements. These features act as supports to your 3D geometry. In addition to the default Reference features, you can create your own additional planes. Until now, you have learned to create sketches on any of the default reference planes (XY, YZ, and XZ planes). If you want to create sketches and geometry at locations other than default reference planes, you can create new reference planes manually. You can do this by using the **Plane** command.

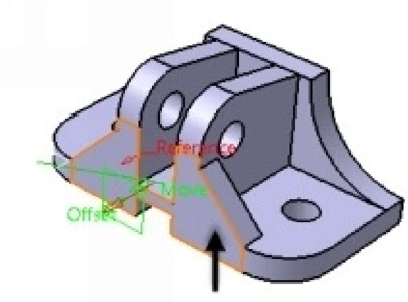
Offset from plane

This method creates a reference plane, which will be parallel to a face or another plane.

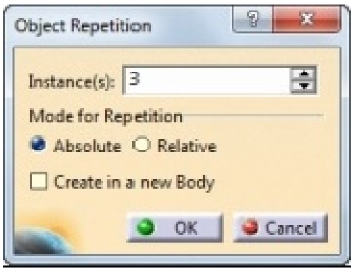
1. Activate the **Plane** command (On the **Reference Elements** toolbar, click the **Plane**).
2. On the **Plane Definition** dialog, select **Plane type > Offset from plane**.



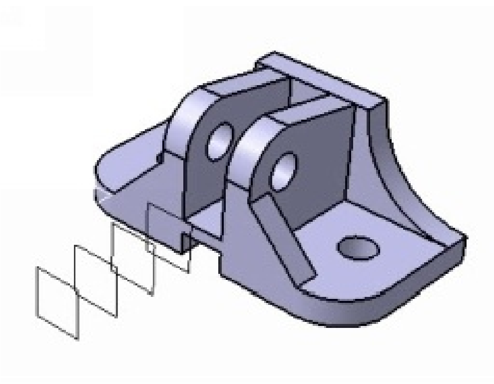
3. Select flat face.



4. Drag the Offset arrow that appears on the plane (or) type-in a value in the **Offset** box to define the offset distance.
5. On the dialog, you can click the **Reverse Direction** button to flip the plane to other side of the model face.
6. If you want to create more than one offset plane, then check the **Repeat object after OK** option and click **OK**.
7. On the **Object Repetion** dialog, type-in a value in the **Instance (s)** box.
8. Check the **Create a new body** option, if you want to create the repeated planes as a separate body.



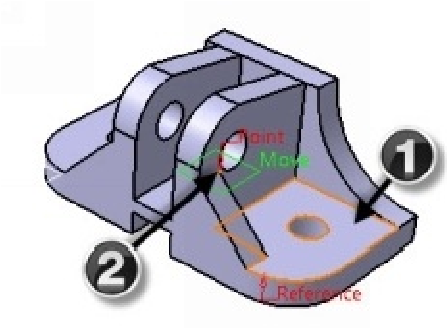
9. Click **OK** to create



Parallel through Point

This method creates a plane parallel to a flat face at a selected point.

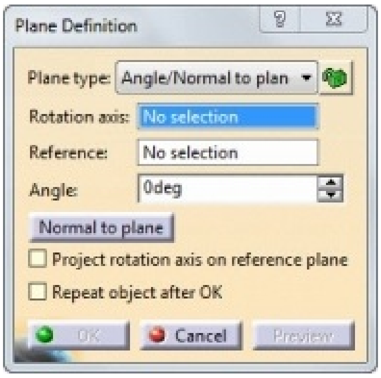
1. Activate the **Plane** command.
2. On the **Plane Definition** dialog, select **Plane type** > **Parallel through point,** and then select a flat face.
3. Select a point to define the parallel plane location.



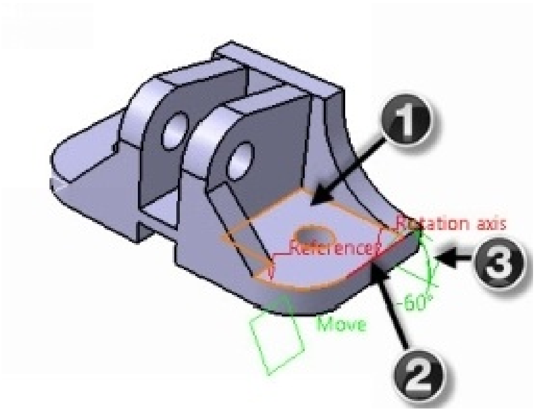
Angle/Normal to plane

This method creates a plane, which will be positioned at an angle to a face or plane.

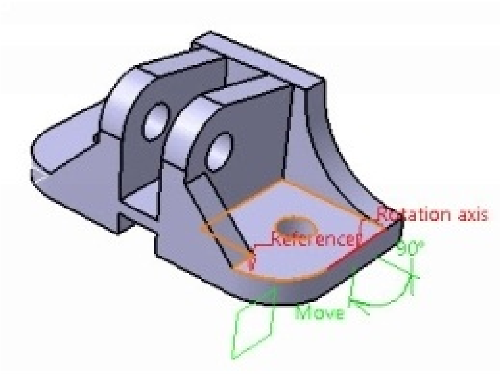
1. Activate the **Plane** command.



2. Select a flat face or plane to define the reference.
3. Select **Plane type** > **Angle/Normal to plane** on the **Plane Definition** dialog.
4. Click on an edge of the part geometry to define the rotation axis.
5. Type-in a value in the **Angle** box and press Enter.



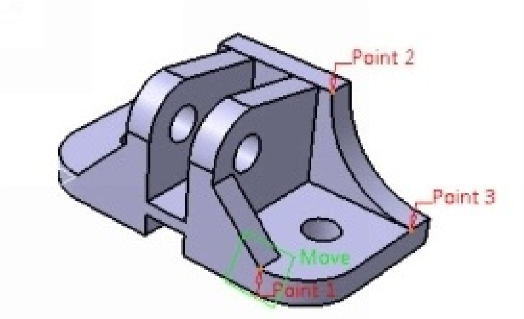
6. On the **Plane Definition** dialog, click the **Normal to plane** button to create a plane normal to the reference.
7. Click **OK**.



Through three points

This method creates a plane by selecting three points.

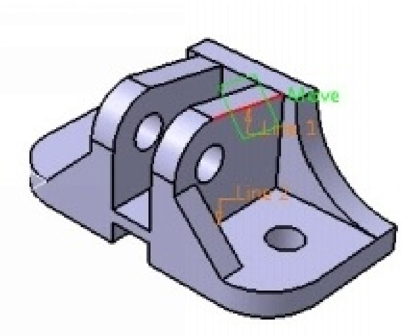
1. Activate the **Plane** command.
2. Select three points from the model geometry.
3. Click **OK** to create a plane passing through the points.



Through two lines

This method creates a plane by selecting two lines.

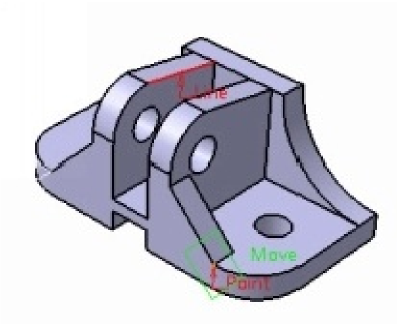
1. Activate the **Plane** command.
2. Select two lines from the model geometry.
3. Click **OK**.



Through point and line

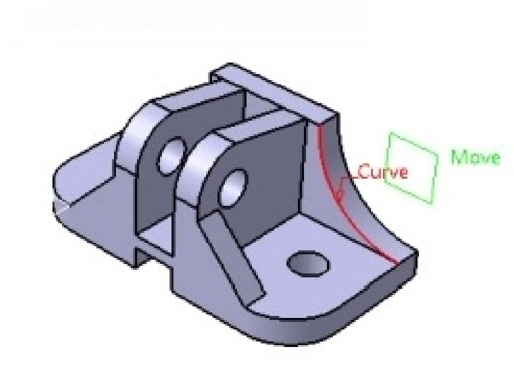
This method creates a plane by selecting a point and line.

1. Activate the **Plane** command.
2. Select a point and line.
3. Click **OK**.



Through planar curve

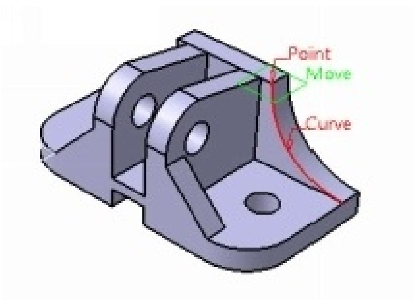
This method creates a plane by selecting a non-linear planar curve.



Normal to curve

This method creates a reference plane, which will be normal (perpendicular) to a line, curve, or edge.

1. On the **Plane Definition** dialog, click **Plane type > Normal to Curve** and select an edge, line, curve, arc, or circle.
2. Click on a point to define the location of the plane.

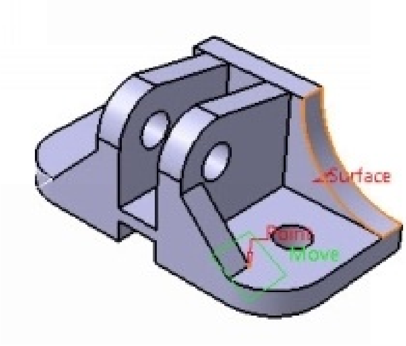


Tangent to surface

This method creates a plane tangent to a curved face.

1. On the **Plane Definition** dialog, click **Plane type > Tangent to surface** and select a curved face.

2. Click on a point. A plane tangent to the selected face appears.



Equation

This method creates a plane using the equation Ax+By+Cz=D. You have to type-in A, B, C, and D values.

1. On the **Plane Definition** dialog, click **Plane type > Equation**.
2. Type-in values in the **A**, **B**, **C**, and **D** boxes on the **Plane Definition** box. The x, y, z coordinates will be calculated automatically and a plane will be created at the origin passing though the x, y, z coordinates.

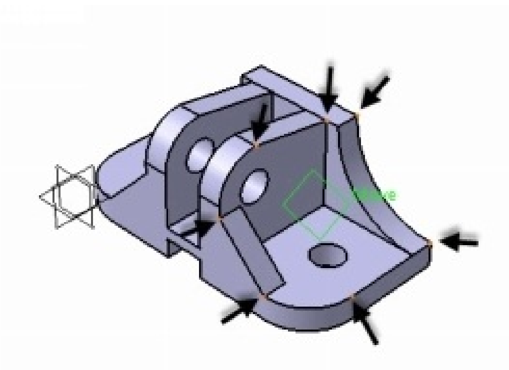
If you want to create a plane normal to the compass, then click the **Normal to compass** button and type-in values in the **C** and **D** boxes. A plane normal to the compass will be created at distance **D/C** from the origin.

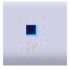
Click the **Parallel to Screen** button to create a plane parallel to screen.

Mean through points

This method creates a plane passing through points.

1. On the **Plane Definition** dialog, click **Plane type > Mean through points**.
2. Select multiple points to create a plane.





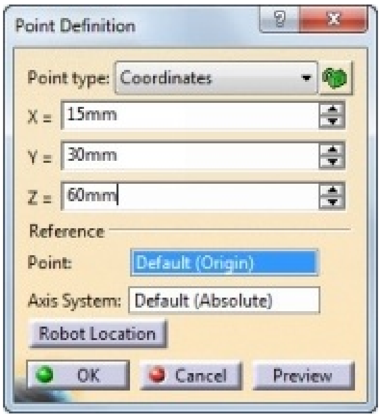
Point

The **Point** command creates points in the 3D space using seven different methods. Following sections explain you to create points using these methods.

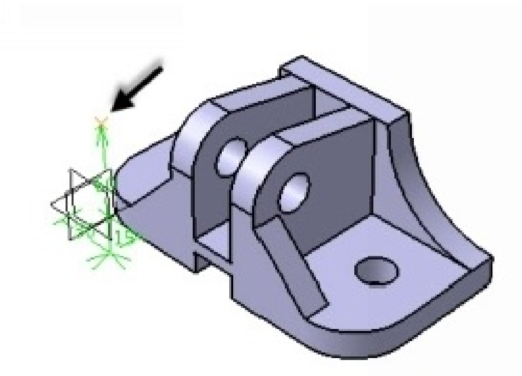
Coordinates

This method creates a point in the 3D space by using the X, Y, Z coordinates that you specify.

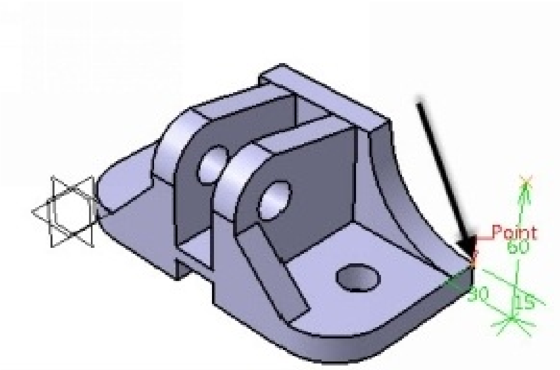
1. Activate the **Point** command (on the **Reference Elements** toolbar, click the **Point** icon).
2. On the **Point Definition** dialog, click **Point type > Coordinates**.
3. On the **Point Definition** dialog, type-in the X, Y, Z coordinates.



A point will be created with reference to origin (0, 0, 0).



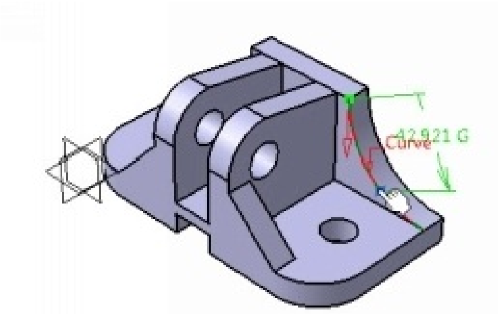
If you want to create a point with reference to a point other than the origin, then select a point from the graphics window.



On curve

This method creates a point on a curve.

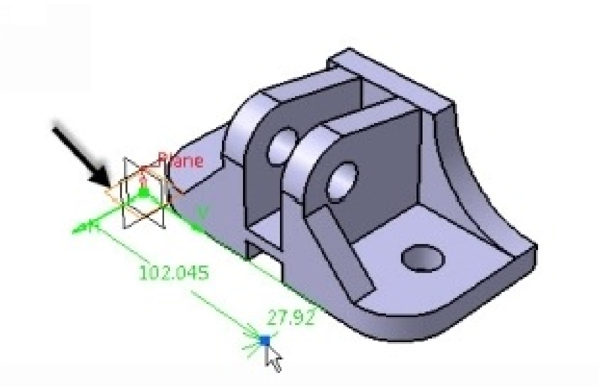
1. Activate the **Point** command and click on a curve or edge.
2. Move the pointer and click to define the location of the point on the curve.



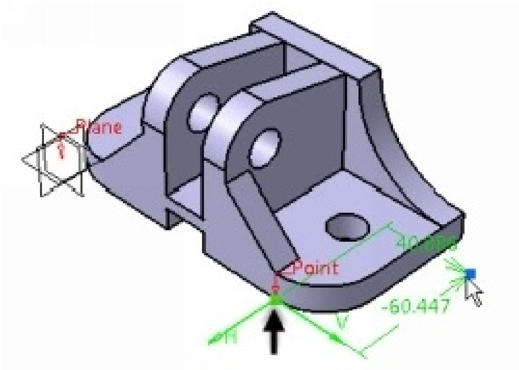
On Plane

This method creates point on a planar face.

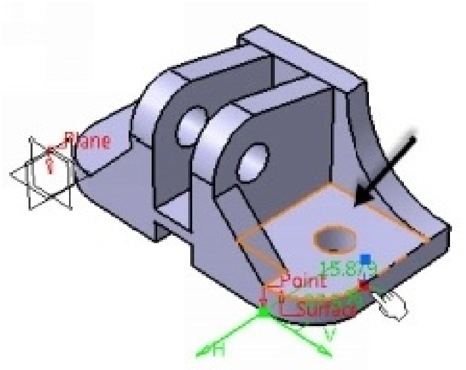
1. Activate the **Point** command and click on a planar face or plane.



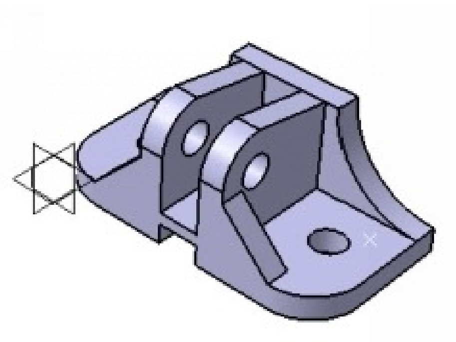
If you want to create the point with reference to a point other than the origin, then click in the **Point** selection box on the **Point Definition** dialog and click to define a reference point.



If you want to project the new point on to a surface, then click in the **Surface** selection box and select a surface.



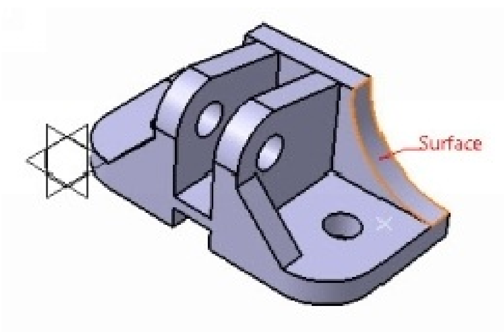
2. Move the pointer and click to define the location of the point (or) type-in values in the H and V boxes on the **Point Definition** dialog.



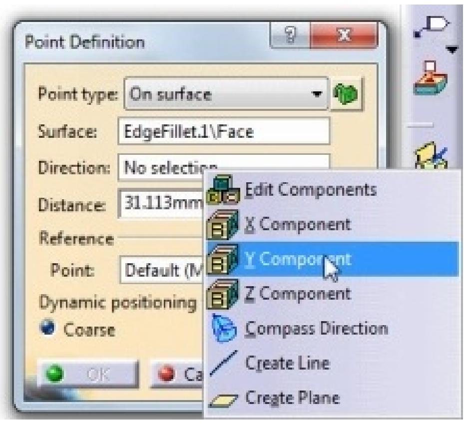
On Surface

This method creates point on a surface.

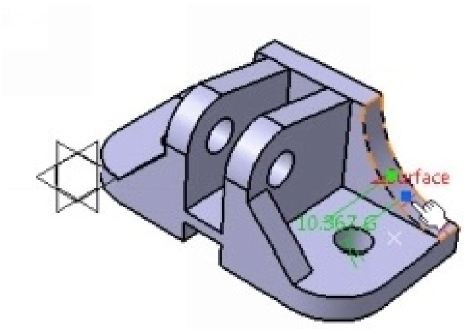
1. Activate the **Point** command and click on a surface.



2. On the **Point Definition** dialog, click in the **Point** selection box and define a reference point on the selected surface.
3. On the **Point Definition** dialog, right click in the **Direction** selection box and select an option to define the direction of the point. For example, select **Y Component** to create the point in the Y-direction.



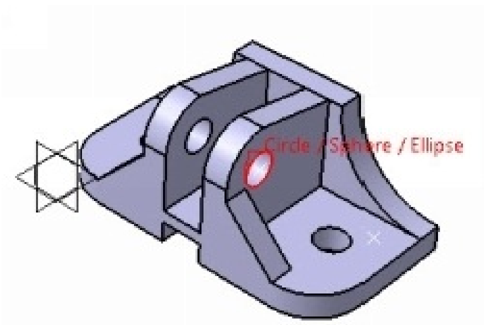
4. Select the **Dynamic Positioning** option.
5. Move the pointer and click to define the location of the point.



Circle/Sphere/Ellipse center

This method creates a point at the center of an arc, circle, sphere or ellipse.

1. On the **Point Definition** dialog, click **Point type > Circle/Sphere/Ellipse center**.
2. Click on an arc, circle, sphere, or elliptical entity.

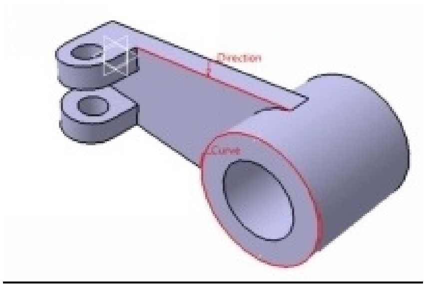


Tangent on curve

This method creates a point tangent to a curve.

1. On the **Point Definition** dialog, click **Point type > Tangent on curve**.

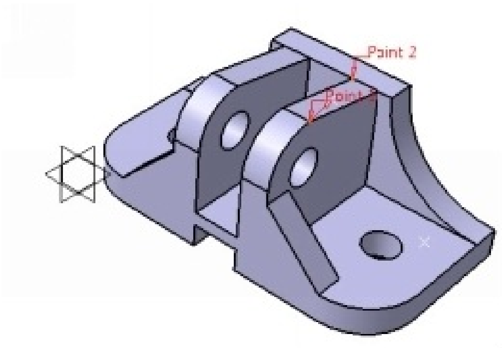
2. Click on an edge or curve.
3. Click on an edge or right click and select an option to define the tangential direction.
4. Click **OK** to create the point.



Between

This method creates a point between two points.

1. On the **Point Definition** dialog, click **Point type > Between**.
2. Select two points from the graphics window or model geometry.

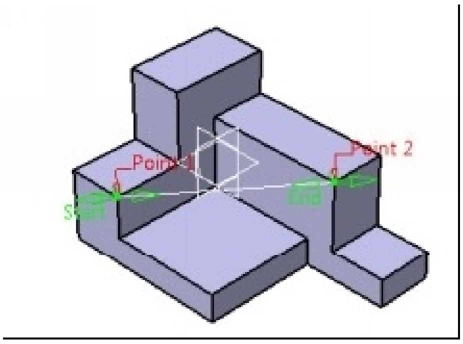


3. Type-in a value in the **Ratio** box. For example, if you type-in 0.5, the point will be created at the midpoint.
4. Click on the face, curve, or edge to define the supporting element.
5. Click **OK** to create the point.

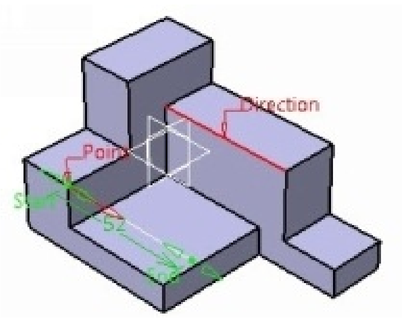
Line

The **Line** command (on the **Reference Elements** toolbar, click the **Line** icon) creates a line in the 3D space. The methods to create lines using this command are explained next.

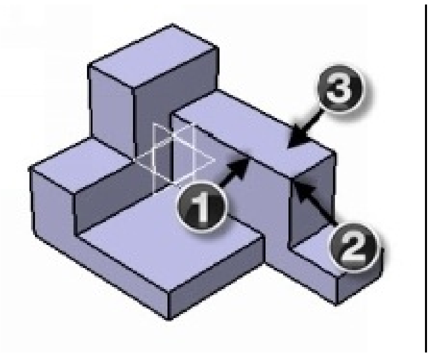
Point-Point

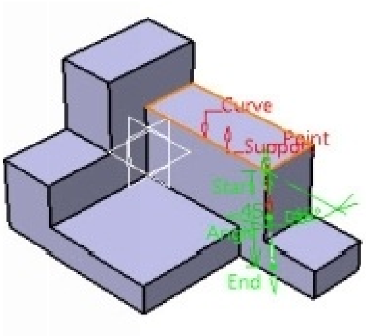


Point-Direction

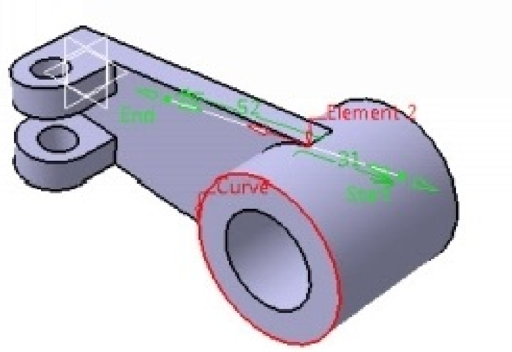


Angle/Normal to curve

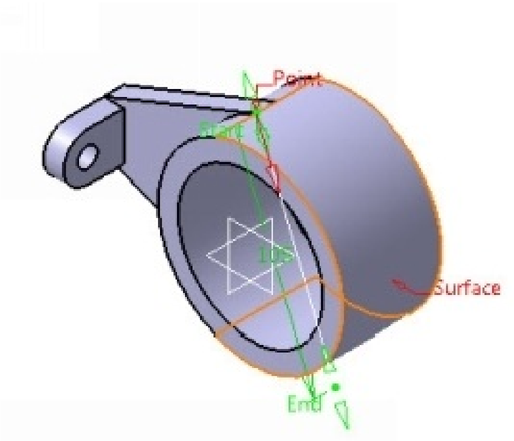




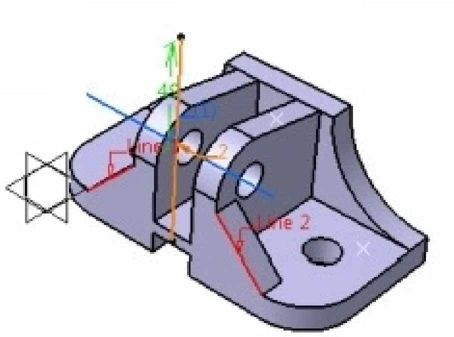
Tangent to curve



Normal to surface



Bisecting



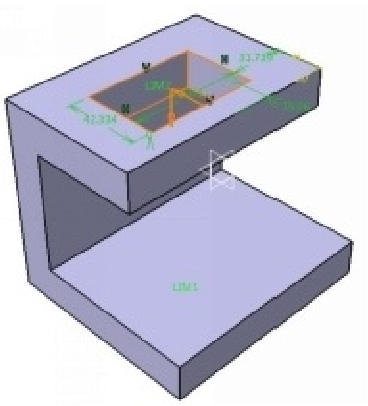
Additional options of the Pad and Pocket commands

The **Pad** and **Pocket** commands have some additional options to create a 3D geometry, complex features, and so on.

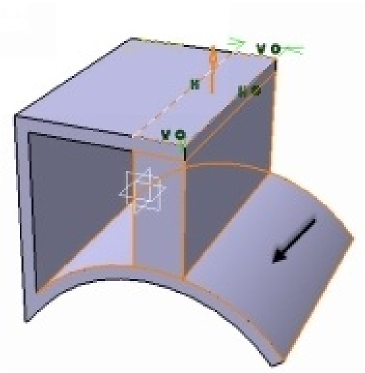
Limits

On the **Pad Definition** or **Pocket Definition** dialog, the **First Limit** and **Second Limit** sections have various options to define the start and end limits of the *Pad* or *Pocket* feature. These options are **Dimension**, **Up to next, Up to last, Up to plane,** and **Up to surface**.

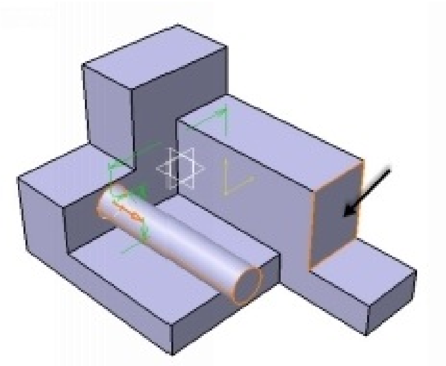
The **Up to next** option extrudes the sketch through the face next to the sketch plane.



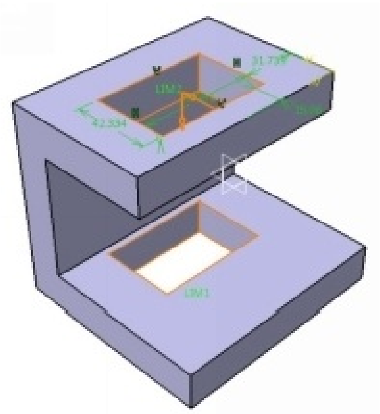
The **Up to surface** option extrudes the sketch up to a selected surface.



The **Up to plane** option extrudes the sketch from the sketch plane up to a selected planar face.

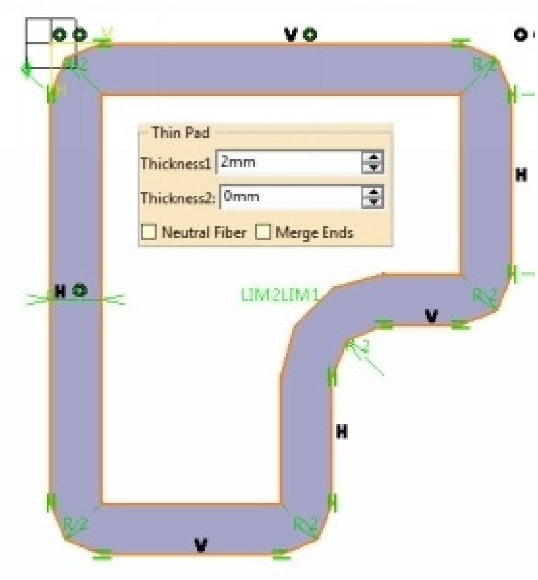


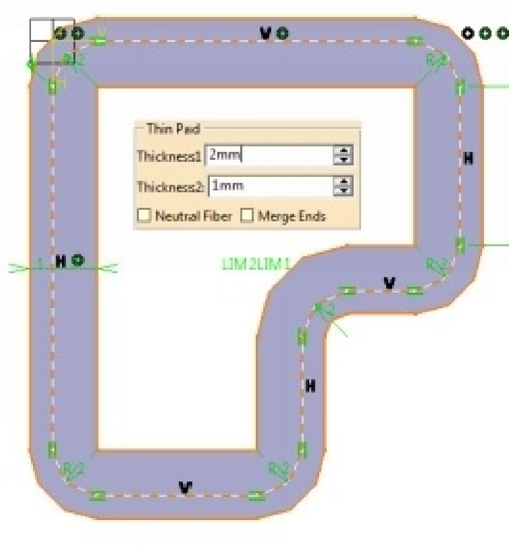
The **Up to last** option extrudes the sketch throughout the 3D geometry.

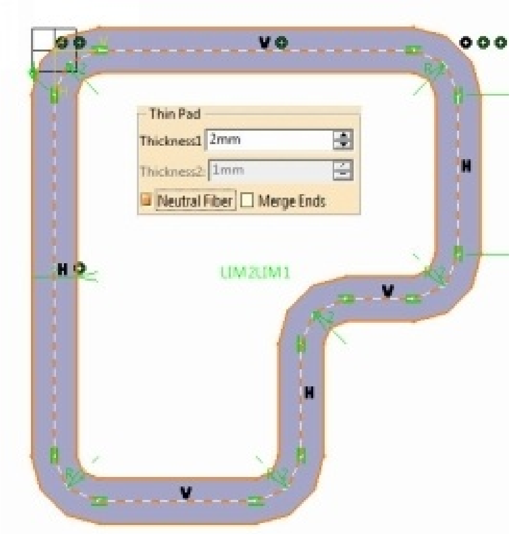


Thick

The **Thick** option will help you to add thickness to the selected sketch. Check this option on the **Pad Definition** or **Revolve Definition** dialog to add thickness to the sketch. Click the **More** button to view the **Thickness 1** and **Thickness 2** boxes. Type-in thickness values in these boxes. Check the **Neutral Fiber** option, if you want to add thickness symmetrically on both sides of the sketch.



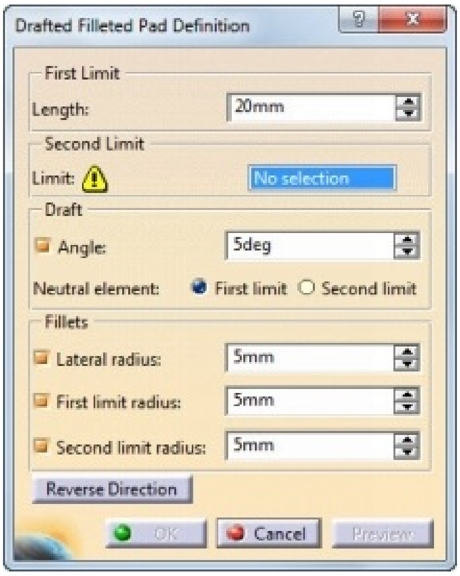




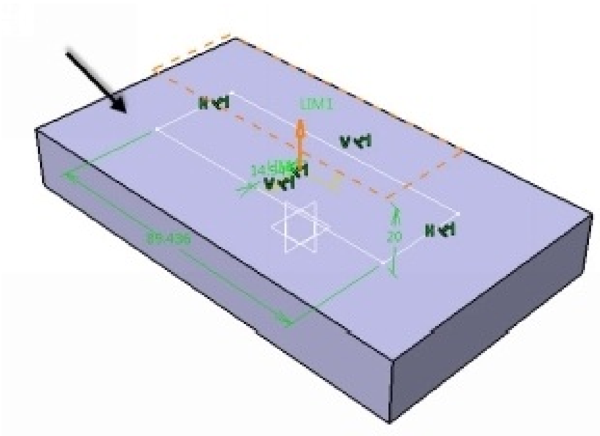
 The Drafted Filleted Pad command

This command creates a drafted pad feature with fillets.

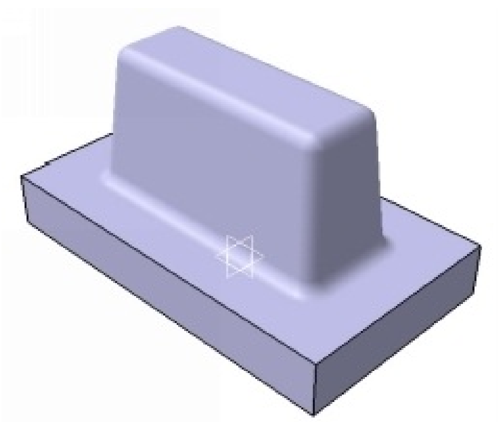
1. Draw the sketch and exit the workbench.
2. On the **Sketch-Based Features** toolbar, click **Pads** drop-down **> Drafted Filleted Pad** (or) click **Insert > Sketch-Based Features > Drafted Filleted Pad** on the Menu.
3. On the **Drafted Filleted Pa1d Definition** dialog, define the First Limit, Draft, and Fillets.



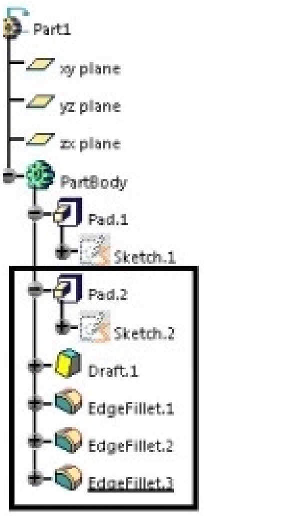
4. Click on a face or plane parallel to the sketch plane.



5. Click **OK**.



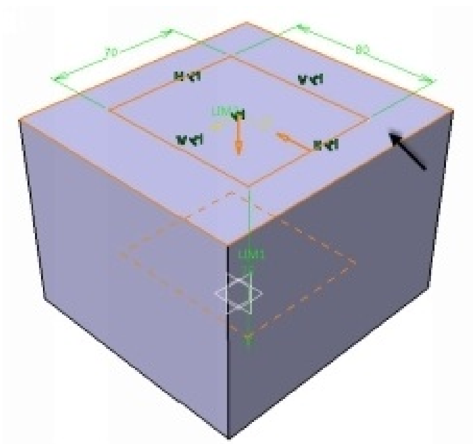
You will notice that the pad, draft, and fillets are created as separate features.



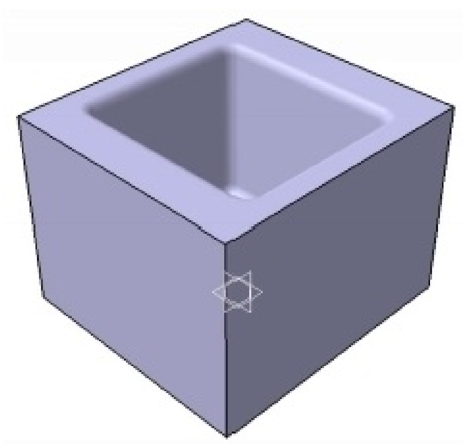
The Drafted Filleted Pocket command

This command creates a drafted pocket with fillets.

1. Draw a sketch and exit the workbench.
2. On the **Sketch-Based Features** toolbar, click **Pokets** drop-down **> Drafted Filleted Pocket** (or) click **Insert > Sketch-Based Features > Drafted Filleted Pocket** on the Menu.
3. Define the parameters on the dialog and select the Second limit.



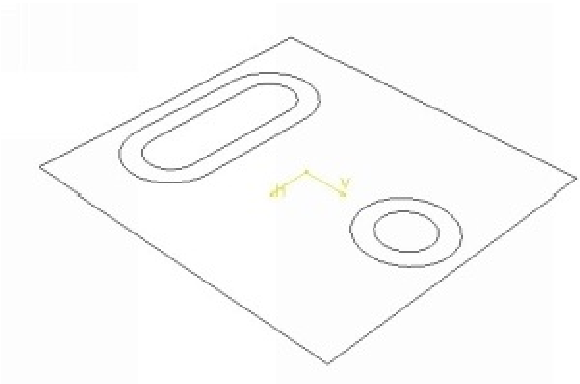
4. Click **OK**.



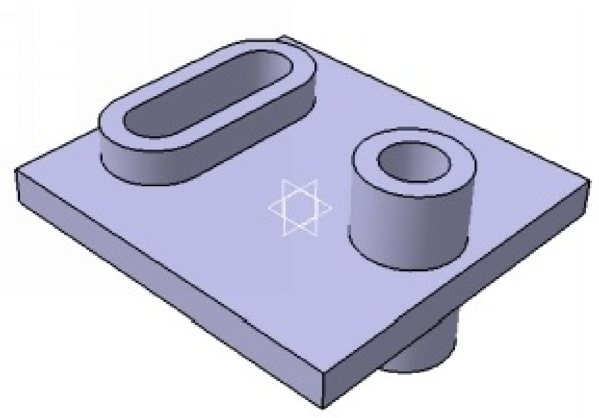
The Multi-Pad command

This command takes a sketch with internal loops and adds multiple thicknesses to it.

1. Draw a sketch, which contains internal loops. Exit the workbench.



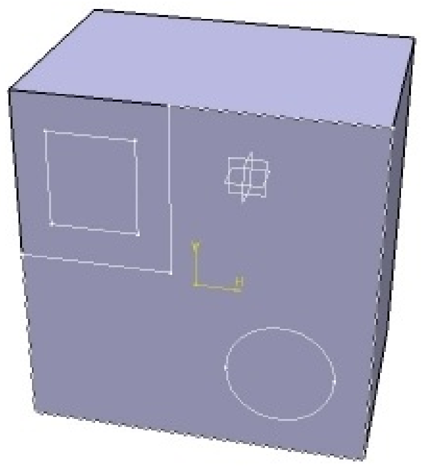
2. On the **Sketch-Based Features** toolbar, click **Pads** drop-down > **Multi-Pad** (or) click **Insert > Sketch-Based Features > Multi-Pad** on the Menu.
3. On the **Multi-Pad Definition** dialog, click on the Extrusion Domains one-by-one and type-in values in the **Length** box.
4. If you want to extrude the loops in both the directions, then click the **More** button on the dialog and define parameters in the **Second Limit** section.
5. You can also define the direction of the extrusion using the **Direction** section.
6. Click **OK**.



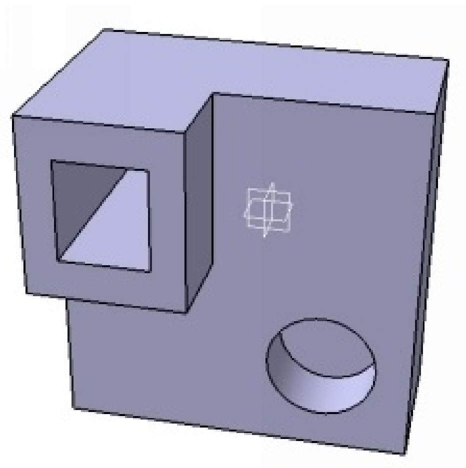
The Multi-Pocket command

This command uses a multi-loop sketch to remove material with multiple depths.

1. Draw a sketch, which contains internal loops. Exit the workbench.



2. On the **Sketch-Based Features** toolbar, click **Pokets** drop-down **> Multi-Pocket** (or) click **Insert > Sketch-Based Features > Multi-Pocket** on the Menu.
3. On the **Multi-Pocket Definition** dialog, select the extrusion domains and define the pocket depth.
4. Click **OK**.



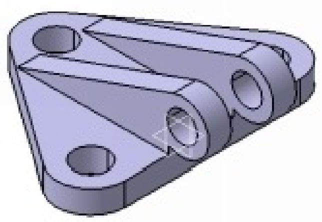
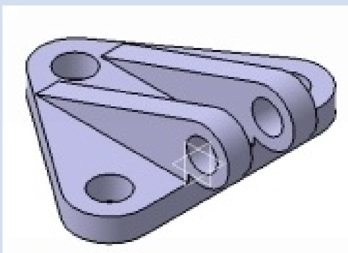
The Apply Material command

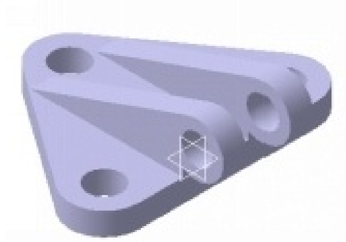
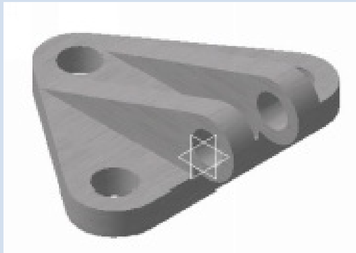
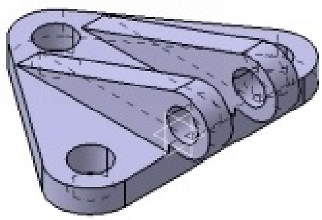
This command applies material to the solid geometry.

1. Activate this command by clicking the **Apply Material** button on the **Apply Material** toolbar.
2. Click on a tab on the **Library** dialog.
3. Select the material from the dialog, and then click on the part geometry.
4. Click **Apply Material** to apply material to the geometry.
5. Click **OK** to close the dialog.

View commands

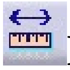
The model display in the graphics window can be determined using various view commands. Most of these commands are located on **View** toolbar or on the **View** menu. The following are some of the main view commands:

	**Fit All In**	The model will be fitted in the current size of the graphics window so that it will be visible, completely.	
	**Pan**	Activate this command and press the left mouse button. Drag the pointer to move the model view on plane parallel to screen.	
	**Rotate**	Activate this command and press the left mouse button. Drag the pointer to rotate the model view.	
	**Zoom In**	Click this button to zoom in to the geometry.	
	**Zoom Out**	Click this button to zoom out of the geometry.	
	**Normal View**	Activate this command and click on a plane or face. The plane will become normal to the screen.	
	**Create Multi-View**	This command creates a multi viewpoint in the graphics window.	
	**Shading with Edges**	This represents the model with shades along with visible edges.	
	**Shading with Edges without Smooth Edges**	This represents the model with shades along with visible edges. The smooth edges of the curved faces will be hidden.	

	**Shading**	This represents the model with shades without visible edges.	
	**Shading with Material**	This represents the model with shades of the applied material.	
	**Wireframe (NHR)**	This represents the model in wireframe.	
	**Customize view Parameters**	This command is used to create your own render style by customizing the view parameters.	
	**Shading with Edges and Hidden Edges**	This represents the model in shades along with hidden edges	
	**Quick View Drop-down**	Use this drop-down to change the model view orientation.	

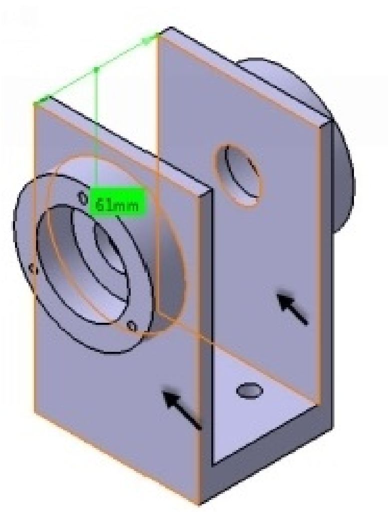
Measure Commands

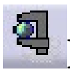
The measure commands help you to measure the physical properties of geometry. These commands are explained next.

Measure Between

This command measures the distance or angle between two elements.

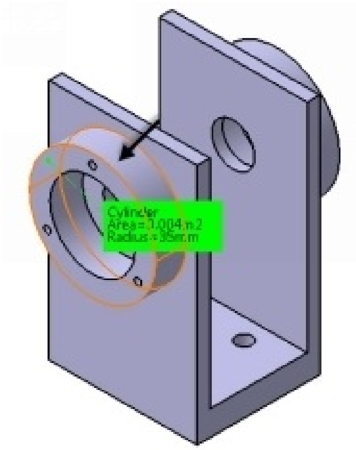
1. On the **Measure** toolbar, click the **Measure Between** button.
2. Click on the two elements. The distance between the two elements will appear.
3. Click **OK**.



Measure Item

This command displays the physical properties of the selected element based on the element type. For example, if you select a linear edge, it displays the length of the linear edge.

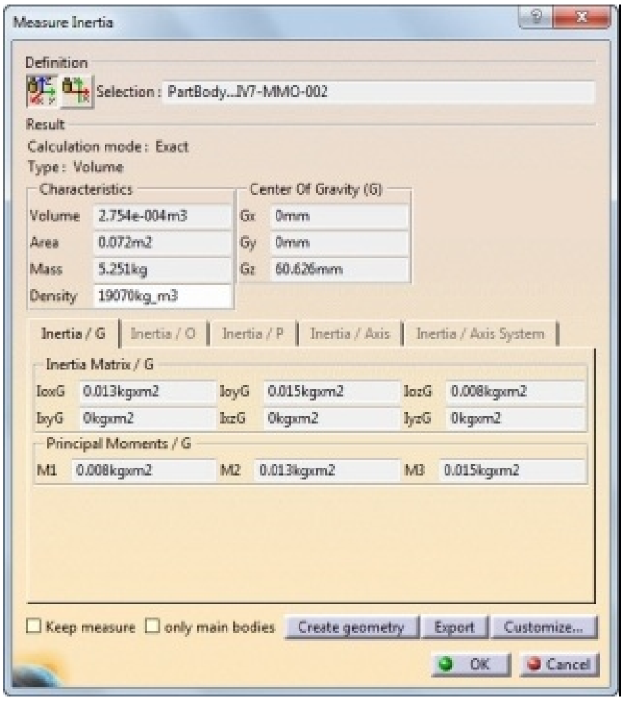
1. On the **Measure** toolbar, click the **Measure Item** button.
2. Click on an element to display the measurements of the element.
3. Click **OK**.



Measure Inertia

This command displays the physical properties of a part body such as volume, inertia, mass, center of gravity, and so on.

1. On the **Measure** toolbar, click the **Measure Inertia** button.
2. Click on the part body. The **Measure Inertia** dialog appears showing the properties of the body.

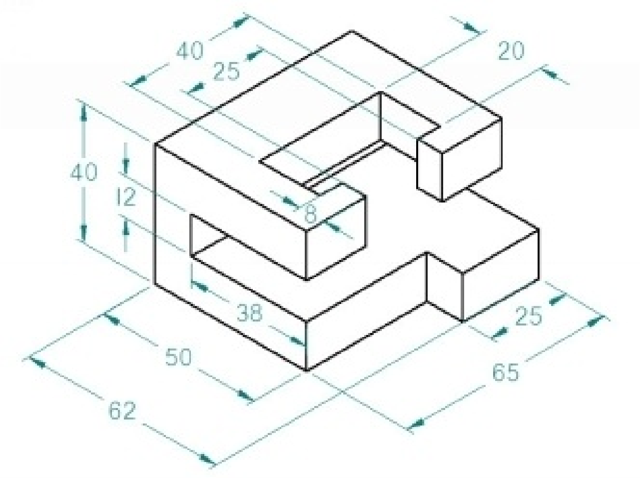


3. On this dialog, click the **Measure Inertia 2D** button, if you want to view the properties in a 2D plane.

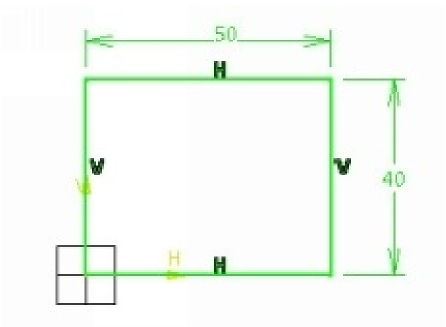
Examples

Example 1

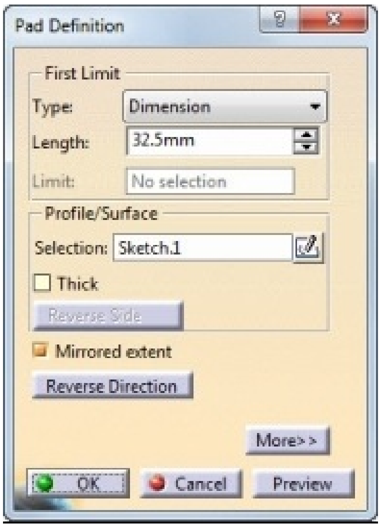
In this example, you will create the part shown below.

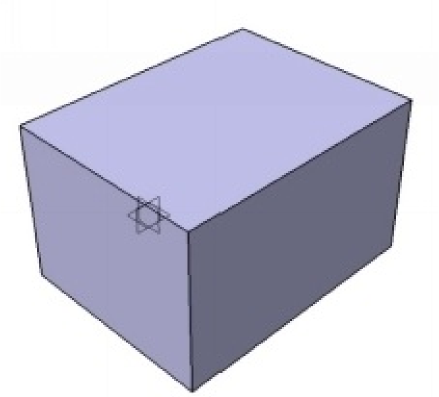


1. Start **CATIA V5-6R2015**.
2. On the Menu, click **File** > **New**.
3. On the **New** dialog, select **List of Types** > **Part**, and then click **OK**.
4. On the **New Part** dialog, type-in **C03-Example1**, and then click **OK**.
5. On the **Sketcher** toolbar, click the **Sketch** icon.
6. Click YZ plane to start the sketch.
7. On the **Profile** toolbar, click **Predefined Profile** > **Rectangle**.
8. Click the origin point to define the first corner of the rectangle.
9. Move the pointer toward top right and click to define the second corner.
10. Apply dimensions to the rectangle.

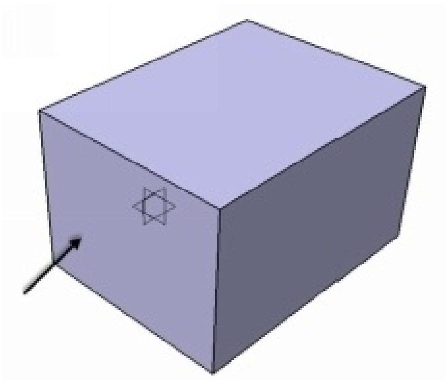


1. On the **Workbench** toolbar, click **Exit workbench**.
2. On the **Sketch-Based Features** toolbar, click **Pads** drop-down > **Pad**.
3. On the **Pad Definition** dialog, click **Type** > **Dimension**.
4. On the dialog, type-in **32.5** in the **Length** box.
5. Check the **Mirror extent** option and click **OK** to complete the *Pad* feature.

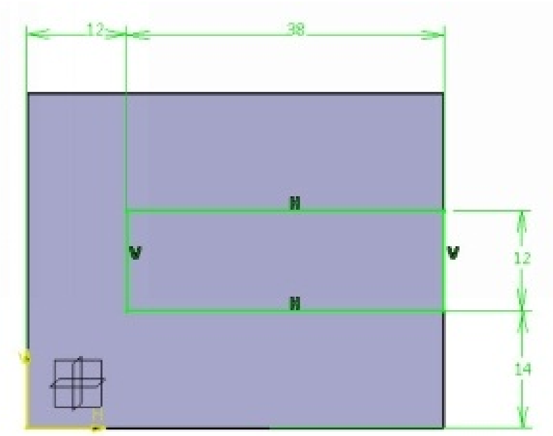




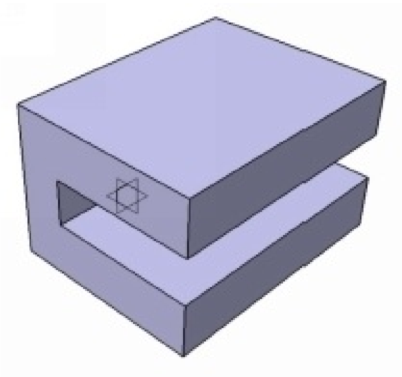
6. On the **Sketch-Based Features** toolbar, click **Pockets** drop-down > **Pocket**.
7. On the **Pocket Definition** dialog, under the **Profile/Surface** section, click the **Sketch** icon.
8. Click on the front face of the part geometry.



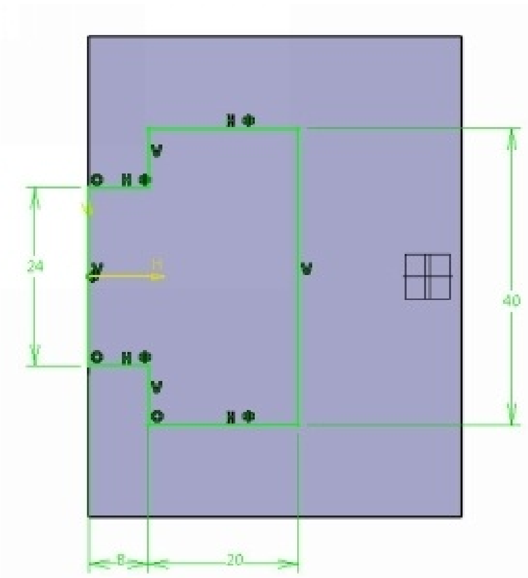
9. Activate the **Rectangle** command and draw a rectangle touching the right edge of the front face.
0. Apply constraints to it.



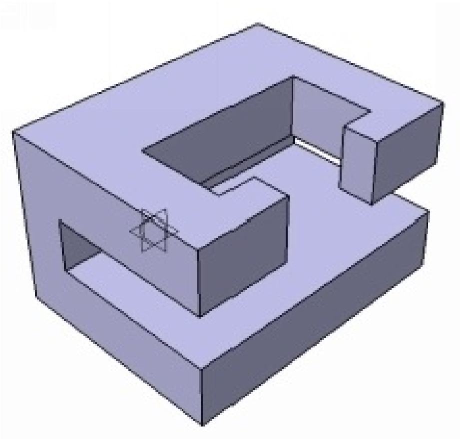
1. On the **Workbench** toolbar, click **Exit workbench**.
2. On the **Pocket Definition** dialog, select **Type > Up to last**.
3. Click **OK** to create the cut throughout the part geometry.



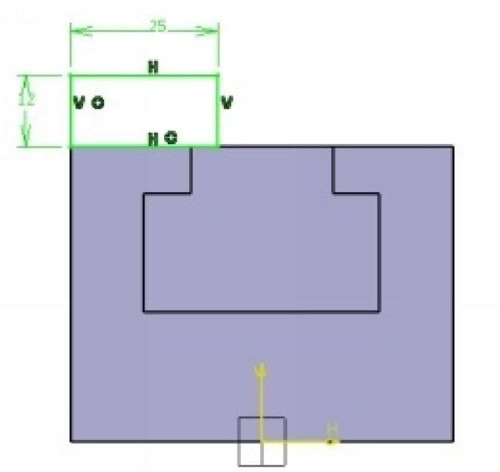
4. Activate the **Pocket** command and click the **Sketch** icon on the **Pocket Definition** dialog.
5. Click on the top face of the part geometry.
6. Draw a closed sketch on the top face.



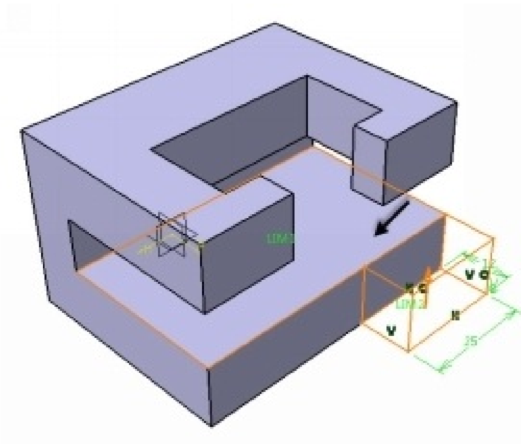
7. On the **Workbench** toolbar, click **Exit workbench**.
8. On the **Pocket Definition** dialog, click **Type** > **Up to n ext**.
9. Click **OK** to create the *Pocket* feature until the surface next to the sketch plane.



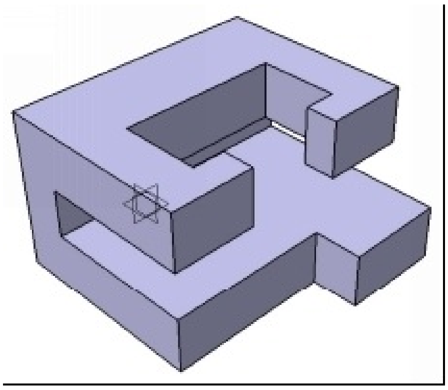
0. Activate the **Pad** command and click the **Sketch** icon on the **Pad Definition** dialog.
1. Click on the XY plane.
2. Draw a closed sketch. Apply dimensions and finish the sketch.



3. On the **Pad Definition** dialog, click **Type > Up to plane** and select the horizontal face of the part geometry, as shown in figure.



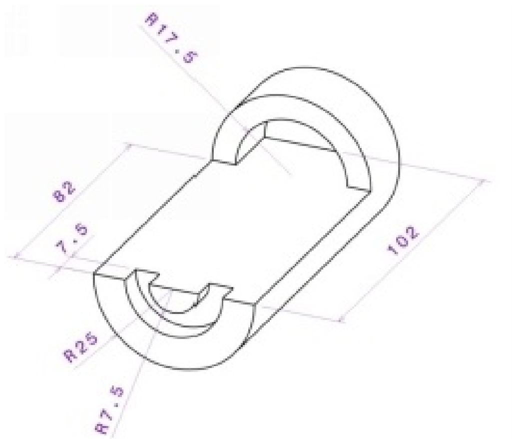
4. Click **OK** to complete the part.



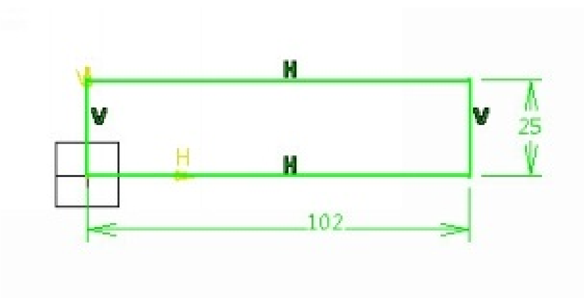
5. Save and close the file.

Example 2

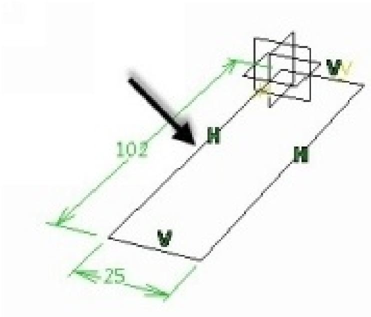
In this example, you will create the part shown below.



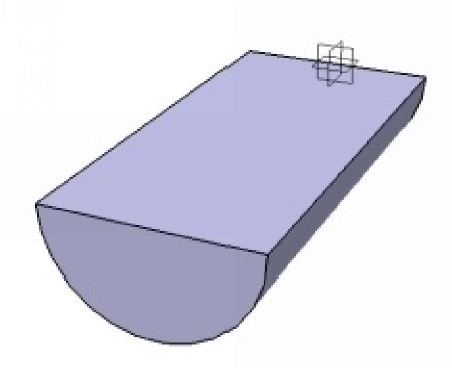
1. Start **CATIA V5-6R2015**.
2. On the Menu, click **File > New**.
3. On the **New** dialog, select **List of Types > Part**, and then click **OK**.
4. On the **New Part** dialog, type-in **C03-Example2**, and then click **OK**.
5. Draw a sketch on the XY plane, as shown below.



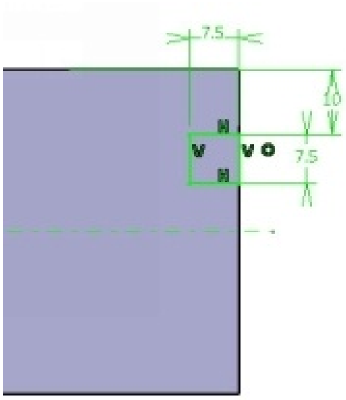
6. Exit the **Sketcher** Workbench.
7. On the **Sketch-Based Features** toolbar, click the **Shaft** icon.
8. Select the sketch and click on the line passing through the origin.



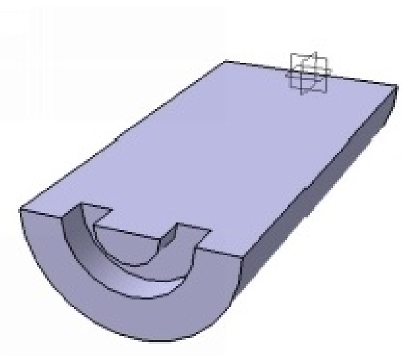
). On the **Shaft Definition** dialog, type-in 0 and 180 in the **First angle** and **Second angle** boxes, respectively.
0. Click **OK** to create the *Shaft* feature.



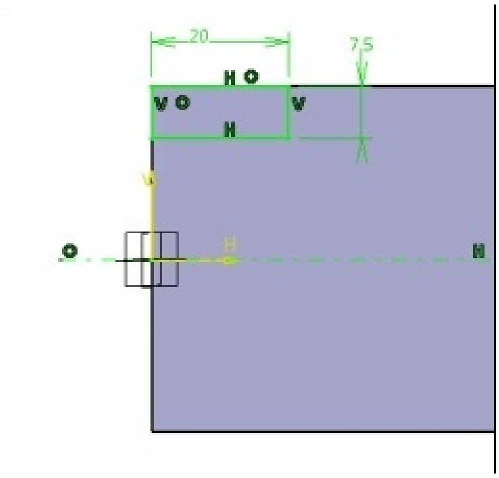
1. On the **Sketch-Based Features** toolbar, click the **Groove** icon (or) click **Insert > Sketch-Based Features > Groove** on the Menu.
2. On the **Groove Definition** dialog, click the **Sketch** icon and select the top face of the part geometry.
3. Draw the sketch on top face and apply dimensions. Exit the sketch.



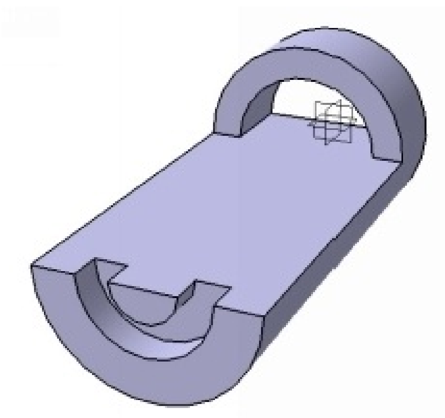
4. Type-in **0** and **180** in the **First angle** and **Second angle** boxes.
5. Click **OK** to create the revolved groove.



6. Activate the **Shaft** command and click the **Sketch** icon on the **Shaft Definition** dialog.
7. Click on the top face of the part geometry.
8. Draw a sketch and click **Exit workbench**.



9. Type-in the **180** and **0** in the **First angle** and **Second angle** boxes, respectively.
0. Click **OK** to add the *Shaft* feature to the geometry.



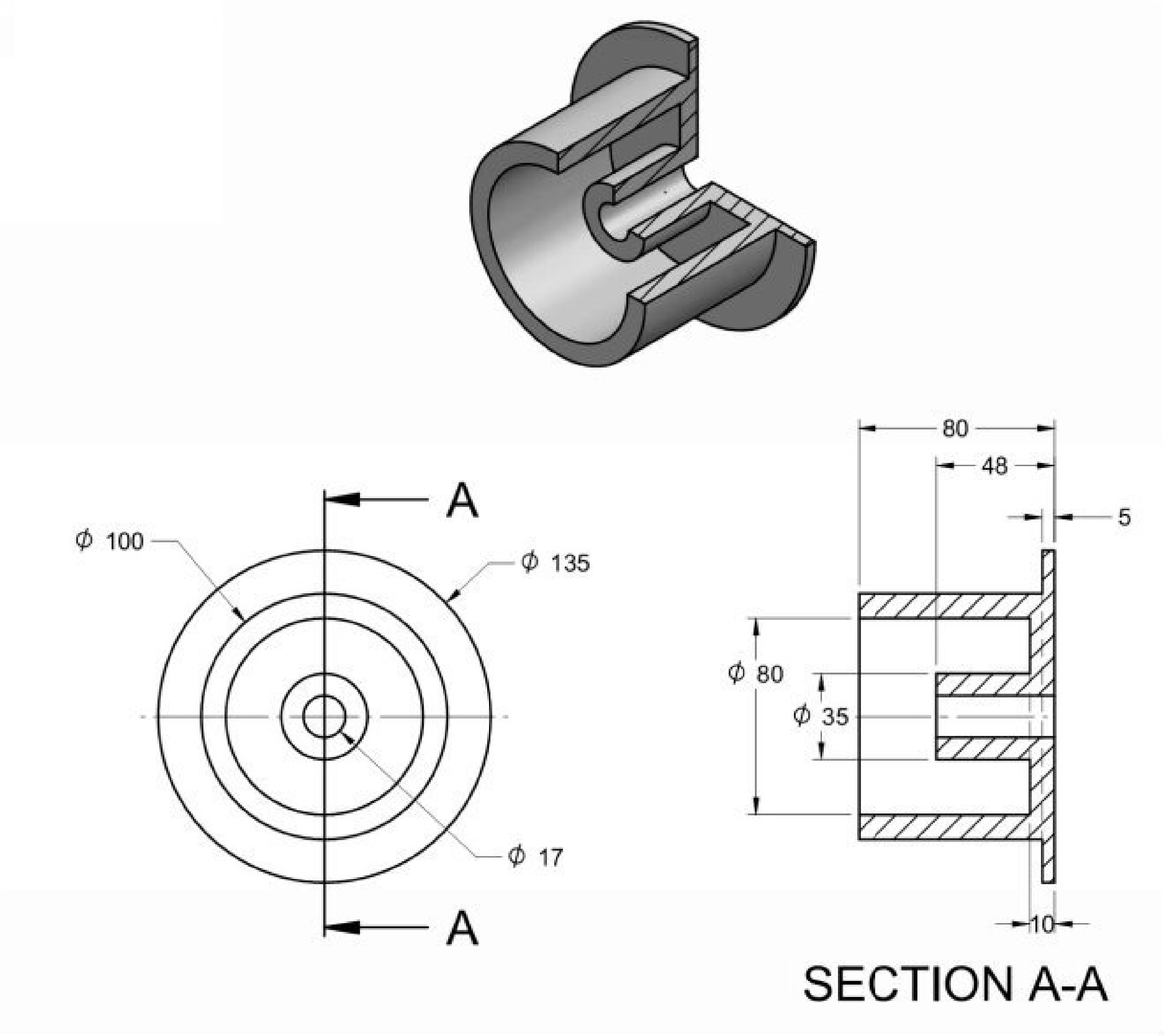
1. Save and close the file.

Questions

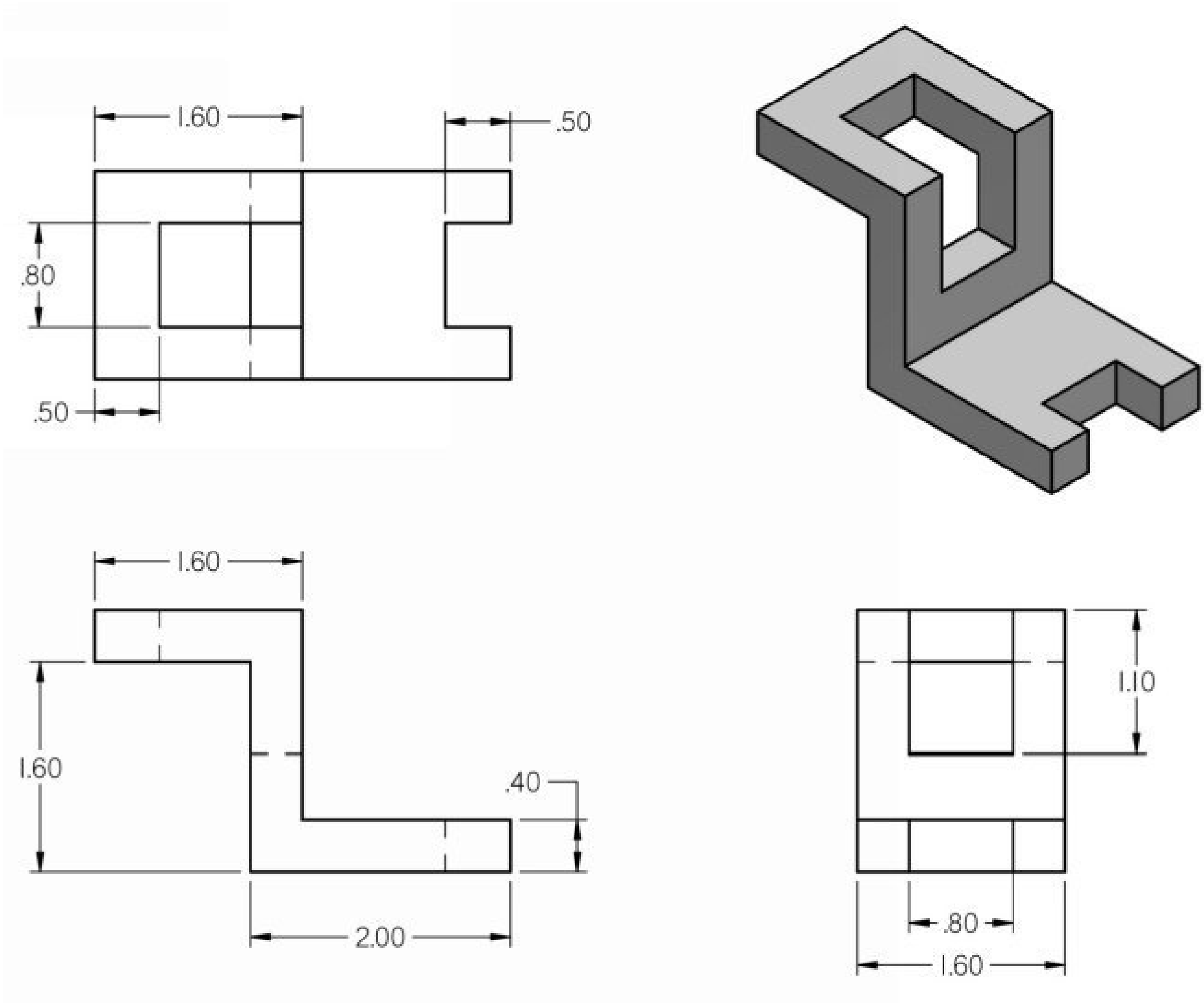
1. How do you create parallel planes in CATIA V5?
2. List any two-limit types available on the **Pad Definition** dialog.
3. List the commands to create pad features.
4. How do you create angled planes in CATIA V5?

Exercises

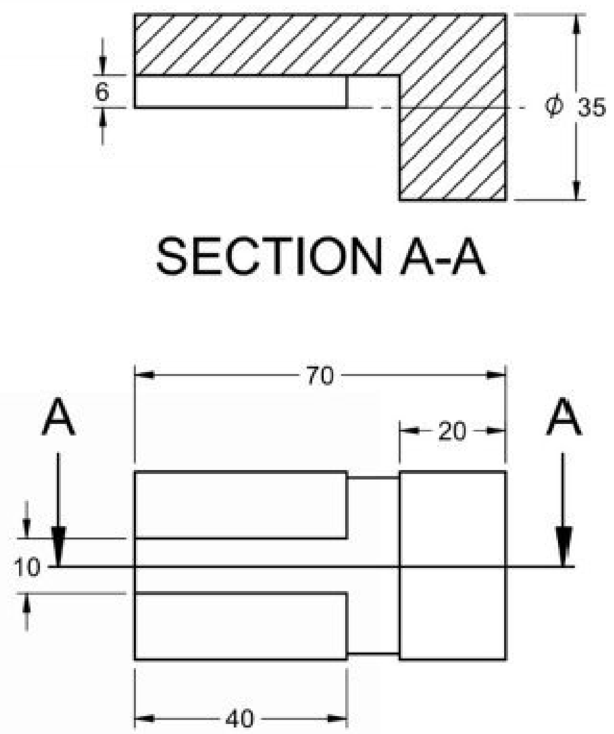
Exercise 1

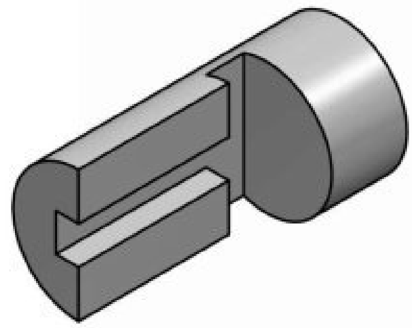


Exercise 2



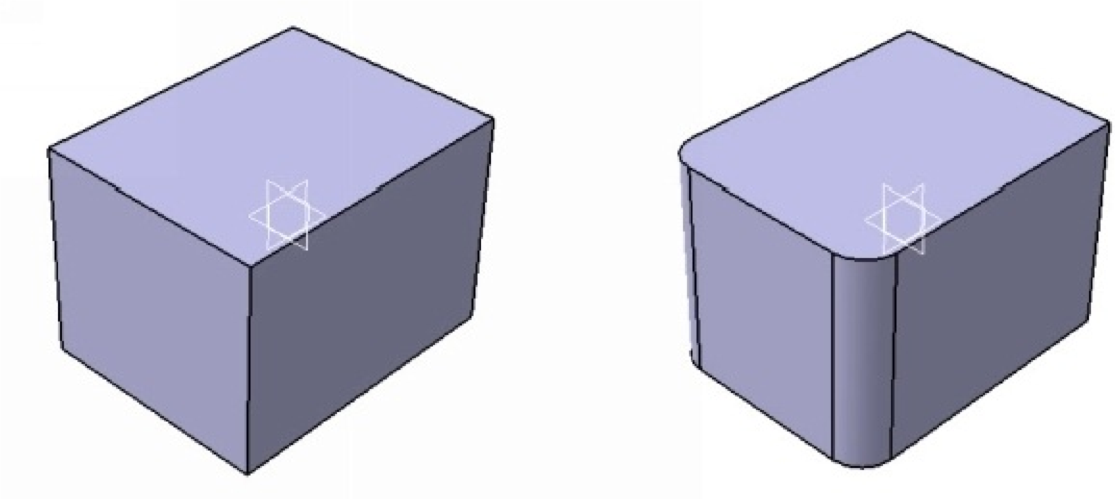
Exercise 3





Chapter 4: Holes and Dress-Up Features

So for, all of the features that were covered in the previous chapter were based on two-dimensional sketches. However, there are certain features in CATIA V5 that do not require a sketch at all. Features that do not require a sketch are called Dress-Up features. You can simply place them on your models. However, you must have some existing geometry to add these features. Unlike a sketch-based feature, you cannot use a Dress-Up feature for a first feature of a model. For example, to create a *Fillet* feature, you must have an already existing edge. In this chapter, you will learn how to add Holes and Dress-Up features to your design.



The topics covered in this chapter are:

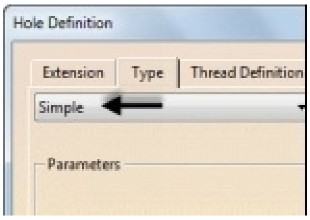
- *Holes*
- *Threads*
- *Fillets*
- *Chamfers*
- *Drafts*
- *Shells*

 Hole

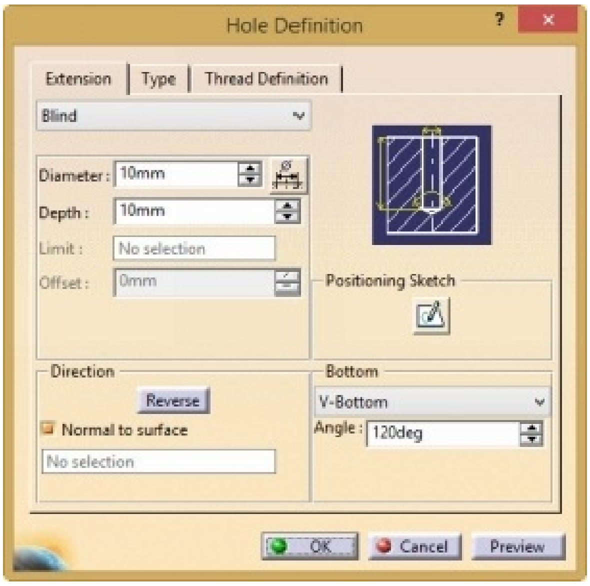
As you know, it is possible to use the *Pocket* command to create cuts and remove material. But, if you want to drill holes that are of standard sizes, the **Hole** command is a better way to do this. The reason for this is it has many hole types already predefined for you. All you have to do is choose the correct hole type and size. The other benefit is when you are going to create a 2D drawing, CATIA V5 can automatically place the correct hole annotation. Activate this command (On the **Sketch Based Features** toolbar, click the **Hole** icon) and click on a face to a add hole. You will notice that a dialog pops up. There are options in this dialog that make it easy to create different types of holes.

Simple Hole

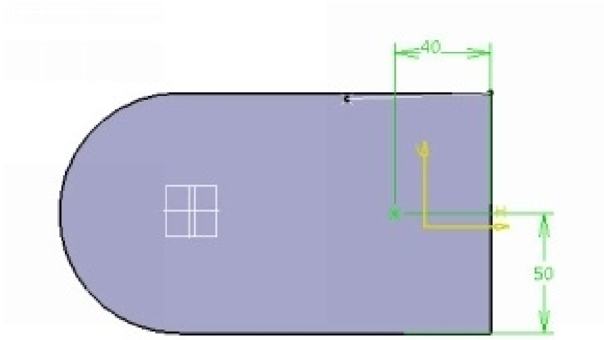
1. To create a simple hole feature, select **Type > Simple** on the **Hole Definition** dialog.



2. Under the **Extension** tab, type-in a value in the **Diameter** box and select the extension type.
3. If you want a through hole, click **Extension** drop-down > **Up To Last**. If you want a blind hole, then select **Blind** from the **Extension** drop-down. Next, type-in a value in the **Depth** box.
4. If you want a V-bottom hole, then select **Bottom > V-Bottom** and type-in a value in the **Angle** box.

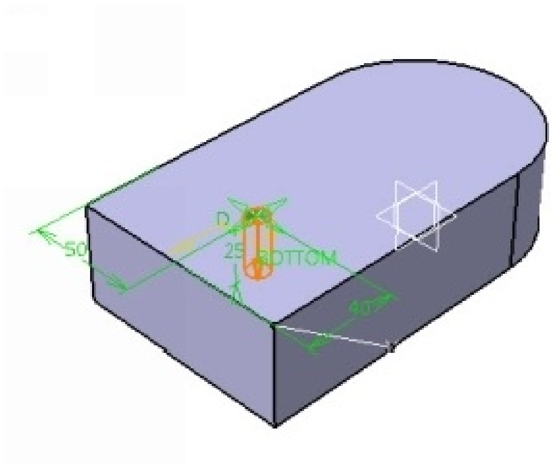


5. On the dialog, click **Positioned Sketch**, to activate the Sketcher Workbench.
6. Add dimensions to define the hole position.

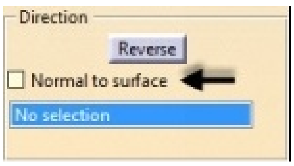


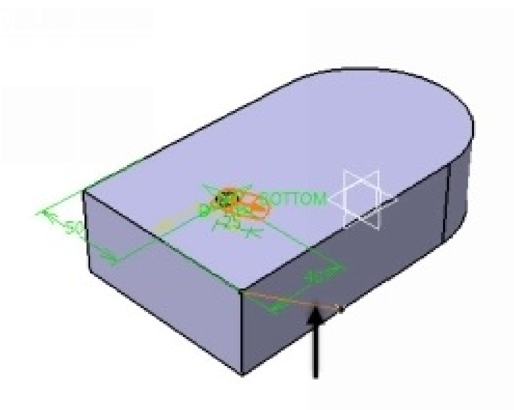
7. On the **Workbench** toolbar, click the **Exit Workbench** icon.

The hole will be created normal to the selected face.

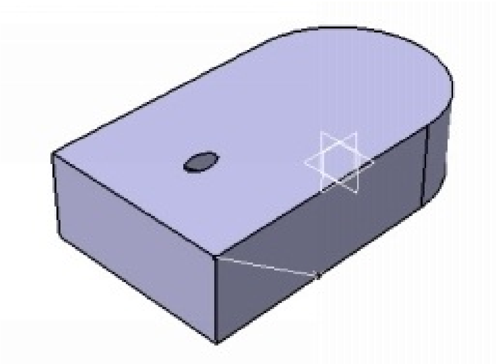


8. If you want to create holes at an angle or along a reference line, then uncheck the **Normal to Surface** option and select a reference line or edge. The hole will be create along the selected line or edge.





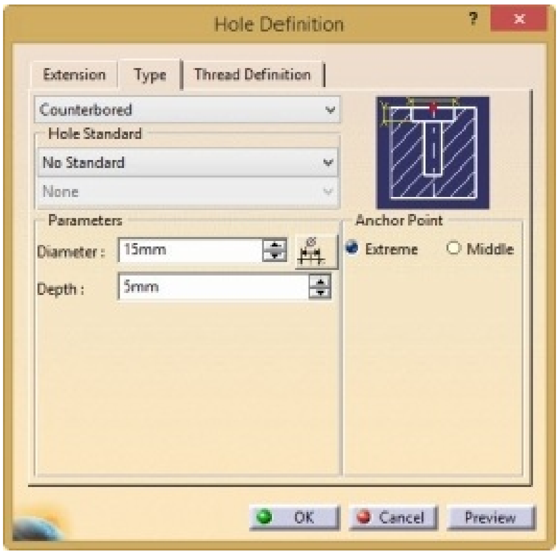
9. Click **OK** to create the holes.



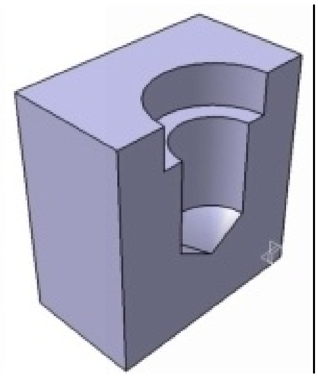
Counterbored Hole

A counterbore hole is a large diameter hole added at the opening of another hole. It is used to accommodate a fastener below the level of work piece surface.

1. To create a counterbore hole, select **Type**> **Counterbored** from the dialog.
2. Next, define the counterbore diameter and counterbore depth.



3. Click the **Extension** tab and type-in values in the **Diameter** and the **Depth** boxes.
4. Click the icon next to the **Diameter** box, if you want to specify the hole tolerances. On the **Limit of Size Definition** dialog, you can specify the tolerances using **General Tolerance**, **Numerical values**, **Tabulated values**, **Single Limit**, or **Information**. Click OK after specifying the tolerances.
5. If you want a V-bottomed counterbore hole, then select **Bottom** > **V-Bottom** and type-in a value in the **Angle** box.
6. Position the hole using the **Positioned sketch** icon.

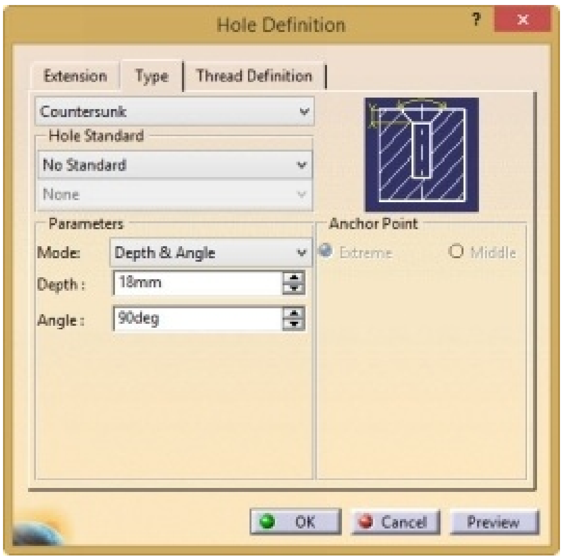


You can also create a standard counterbored hole using the **Hole Standard** drop-down. You can select **Metric_cap_screws** or **Socket_Head_Cap_Screws** hole standard. Next, select the hole size from the drop-down available below the **Hole Standard** drop-down.

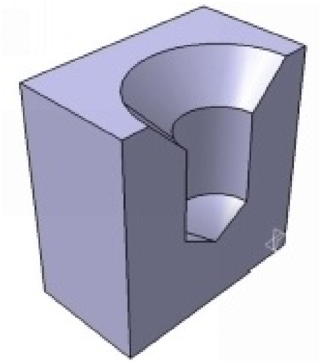
Countersunk Hole

A countersunk hole has an enlarged V-shaped opening to accommodate a fastener below the level of work piece surface.

1. To create a countersunk hole, select **Type > Countersunk**.
2. Under the **Parameters** section, select **Mode > Depth & Angle** or **Depth & Diameter** or **Angle & Diameter**. For example, if you select **Angle & Diameter**, you must specify the Countersink diameter and angle.
3. Type-in values in the **Parameters** section.



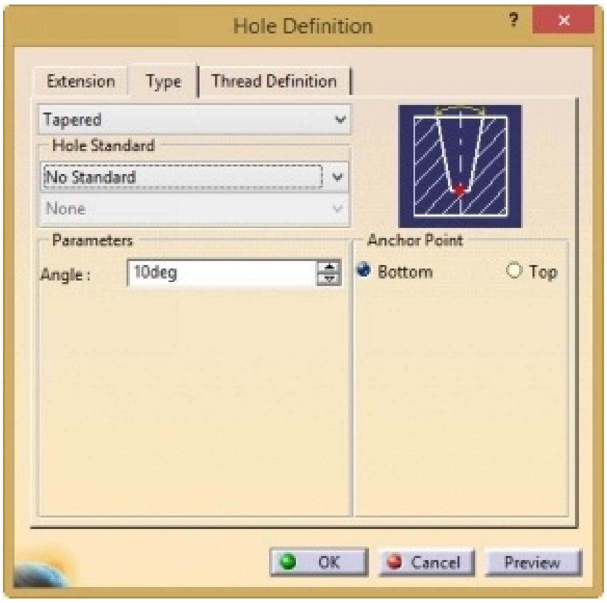
7. Click the **Extension** tab, specify the diameter, depth, and end condition of the hole.
8. Position the hole using the **Positioned sketch** icon.



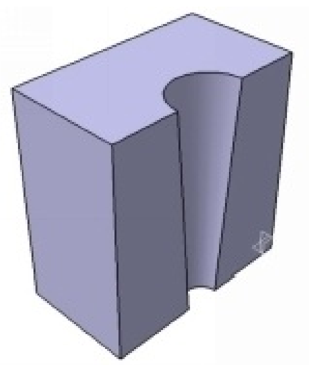
Tapered Hole

Tapering is the process of decreasing the hole diameter toward one end. A tapered hole has a smaller diameter at the bottom.

1. To create a tapered hole, select **Type > Tapered** on the **Hole Definition** dialog.
2. Type-in the taper angle value in the **Angle** box.
3. Select the **Anchor Point** to define the bottom or top diameter.

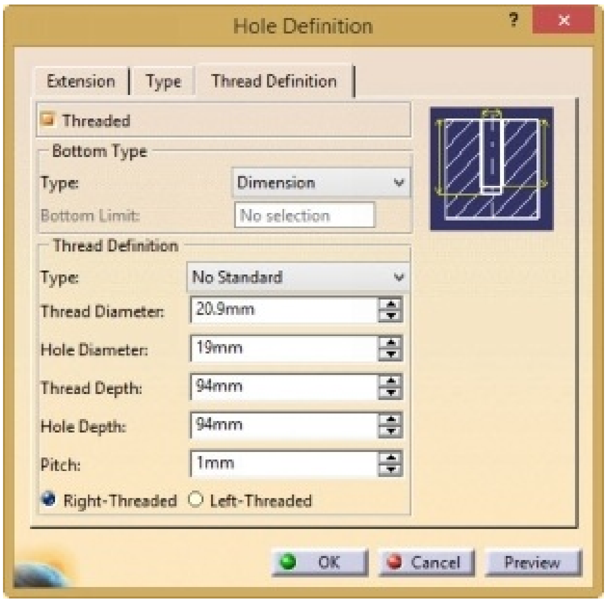


4. Specify the **Extension** parameters and hole position.



Threaded Hole

1. To create a threaded hole feature, click the **Thread Definition** tab on the **Hole Definition** dialog.



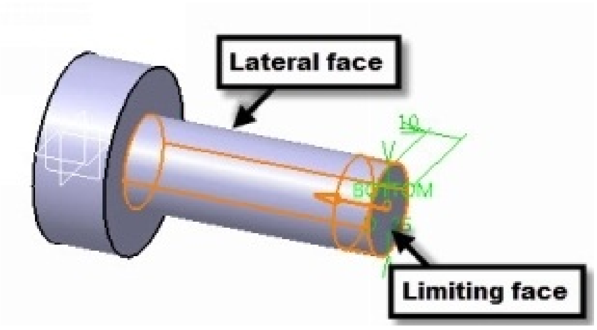
2. Check the **Threaded** option on the dialog.
3. Under the **Bottom Type** section, select the type of end condition. You can select **Dimension**, **Support Depth**, or **Up-To-Plane**. If you select **Dimension**, then you have to type-in the thread depth. The **Support Depth** option creates the thread throughout the hole. The **Up-To-Plane** option creates the thread up to a plane.
4. Under the **Thread Definition** section, select the type of the thread. You can select standard or non-standard threads. If you select standard threads, then you have to select the thread size from the **Thread Description** menu. All thread size and pitches will be calculated, automatically. If you select **No Standard** thread, then you have to type-in the thread diameter, hole diameter, and pitch.
5. Define the thread direction by select the **Right-Threaded** or **Left-Threaded** option.

The Thread/Tap command

This command adds a thread/tap feature to a cylindrical face. A thread is added to the outer cylindrical face, whereas a tap is added to the inner cylindrical face (holes). You add thread/tap features to a 3D geometry so that when you create a 2D drawing, CATIA V5 can automatically place the correct thread annotation.

1. On the **Dress-Up Features** toolbar, click the **Thread/Tap** button (or) click **Insert > Dress-Up Features > Thread/Tap** on the Menu bar. The **Thread/Tap Definition** dialog pops up on the screen.
2. To create a thread, select the **Thread** option and click on the outer cylindrical face of the part geometry.

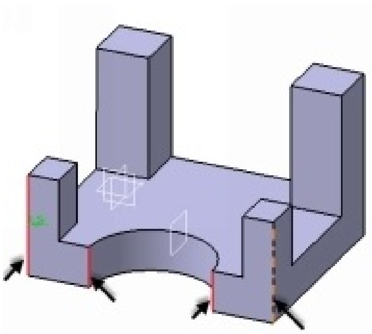
3. Click on the end face of the cylindrical feature to define the limiting face.
4. Under the **Numerical Definition** section, select the thread type from the **Type** menu. You can select a standard or non-standard thread type. In case of standard threads, the diameter of the cylinder should match any standard thread format.
5. For non-standard threads, type-in the thread diameter, thread depth, and pitch values.
6. Define the thread direction and click **OK**.



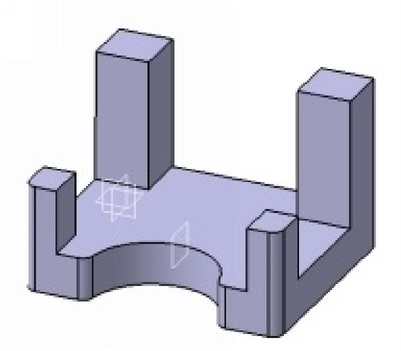
The Edge Fillet command

This command breaks the sharp edges of a model and blends them. You do not need a sketch to create a fillet. All you need to have is model edges.

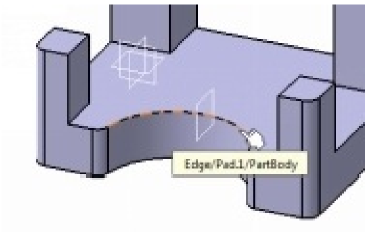
1. On the **Dress-Up Features** toolbar, click the **Edge Fillet** button (or) click **Insert > Dress-Up Features > Edge Fillet** on the Menu bar. The **Edge Fillet Definition** dialog pops up on the screen.
2. Select the edges to fillet. You can also select all the edges of a face by simply clicking on the face. By mistake, if you have selected a wrong edge you can deselect it by selecting the edge again.

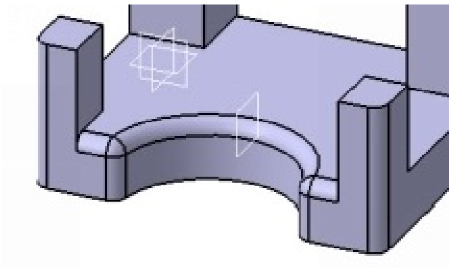


3. You can change the radius by typing a value in the **Radius** box available on the **Edge Fillet Definition** dialog. As you change the radius, all the selected edges will be updated. This is because they are all part of one instance. If you want the edges to have different radii, you must create rounds in separate instances. Select the required number of edges and click **OK** to finish this feature. The *Edge Fillet* feature will be listed in the Specification Tree.

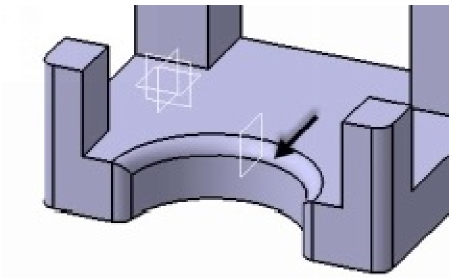


If you want to select all the edges that are tangentially connected, then select **Propagation > Tangency** on the dialog.

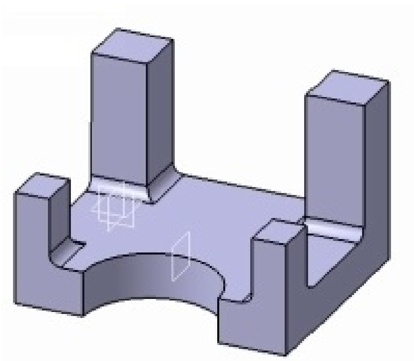




If you select **Propagation** > **Minimal** on the dialog, the selected will be filleted ignoring the connected ones.



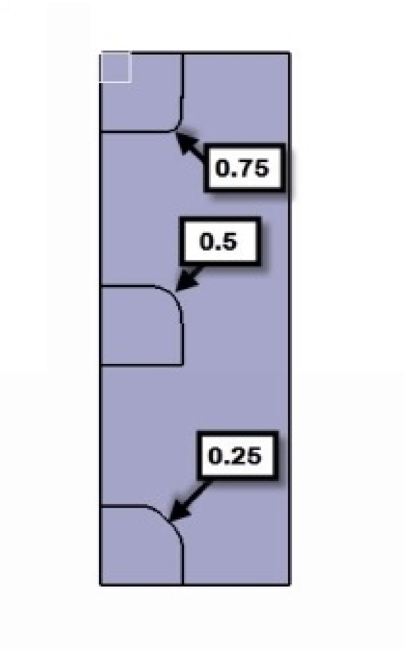
If you select **Propagation** > **Intersection**, then you can fillet the intersections between two features of the geometry.



Select **Propagation** > **Intersection with selected features** to fillet the intersections between the selected features only.

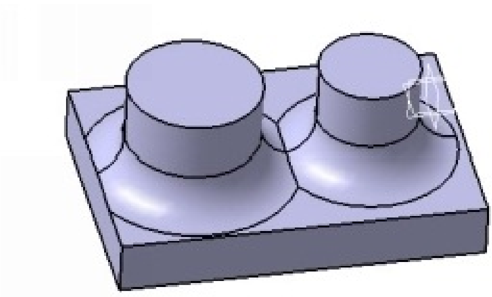
Conic fillets

By default, the edge fillets have a circular arc profile. However, if you want to create a fillet with conical arc profile, then check the **Conic parameter** option on the dialog. Next, type-in a value in the **Conic parameter** box. The fillets with different conic parameters are shown below.



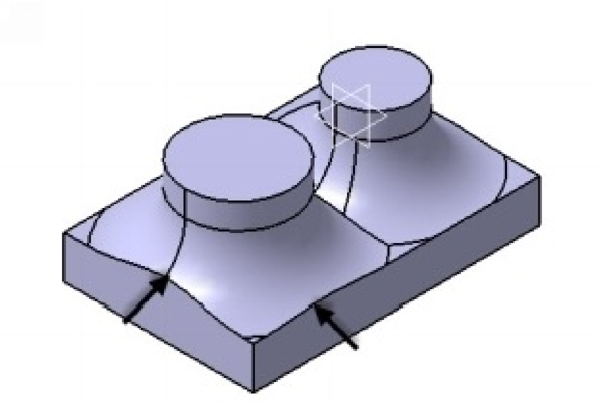
Trim ribbons

If you are creating fillets, which intersect each other, then you need to check the **Trim ribbons** option. This trims the intersecting portion.

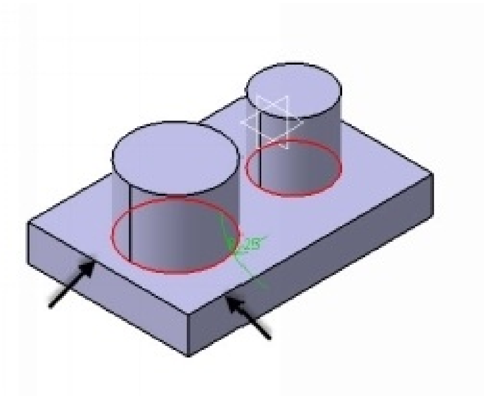


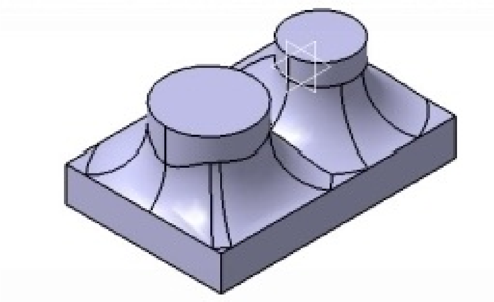
Edge(s) to keep

If you create fillets, which intersect with the adjacent edges, then this may deform the edges.



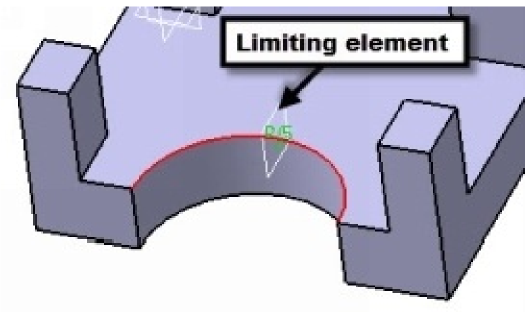
To avoid this, click the **More** button on the dialog and click in the **Edge(s) to keep** selection box. Now, select the adjacent edges.

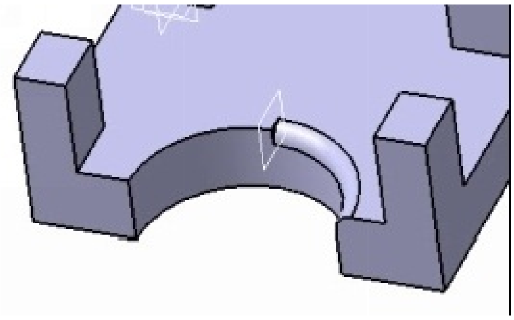




Limiting element(s)

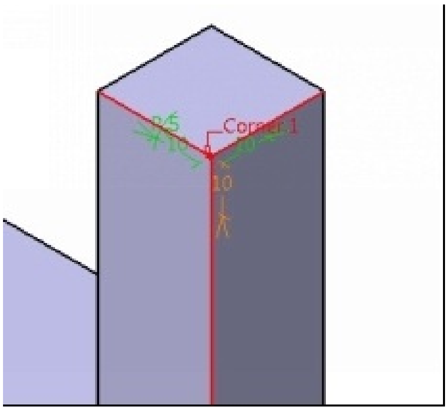
If you want to create an edge fillet only up to some distance, then expand the **Edge Fillet Definition** dialog and click in the **Limiting element(s)** selection box. Click on a reference element (plane or point) to define the limiting elements. If there is no reference element available, then click the right mouse button in the **Limiting element(s)** selection box and create one.



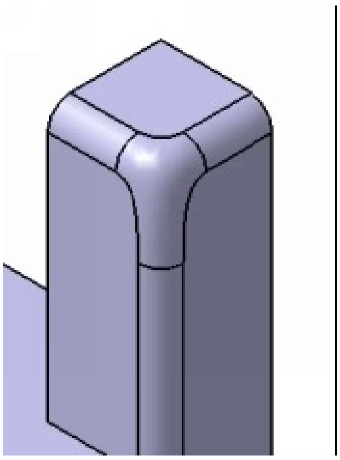


Blend corner(s)

If you create an edge fillet on three edges that come together at a corner, you have the option to control how these three fillets are blend together. Activate the **Edge Blend** command and select the three edges that meet together at a corner. On the **Edge Fillet Definition** dialog, click the right mouse button in the **Blend corner(s)** selection box, and then select **Create by edges**. If you select **Create by edges**, then the corner point will be selected, automatically. You can also select the vertex where the three fillets meet. You will notice that three setback distances appear at the corner.



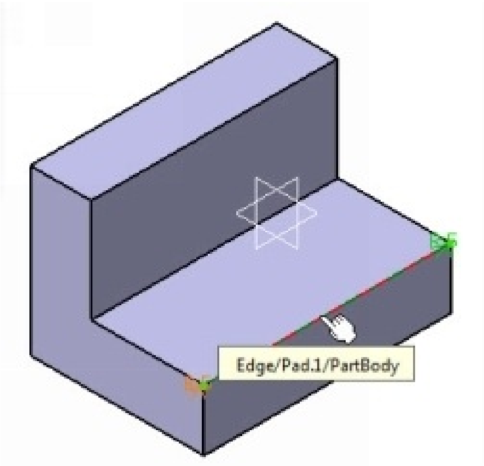
Double-click on the setback distances and type-in values in the **Parameter Definition** boxes.



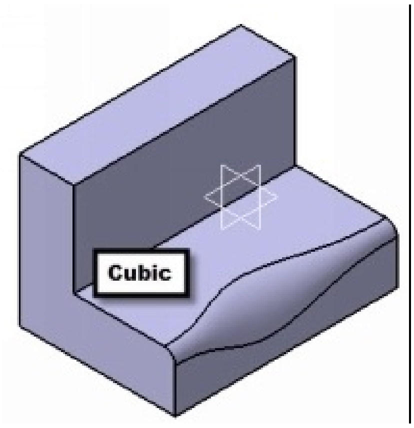
Variable Radius Fillet

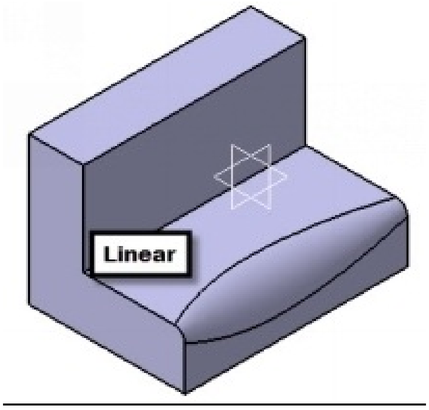
CATIA V5 allows you to create a fillet with a varying radius along the selected edge.

1. On the **Dress-Up Features** toolbar, click **Fillets** drop-down > **Edge Fillet** (or) click **Insert > Dress-Up Features > Edge Fillet** on the Menu bar.
2. On the **Edge Fillet Definition** dialog, click the **Variable** icon.
3. Click on the edge to fillet.
4. Click in the **Points** selection box to select variable radius points.
5. On the selected edge, select points to define variable radius. You will notice that radius values appear on the selected points.



6. Double-click on the radius values that appear on the selected points. The **Parameter Definition** box appears.
7. Type-in a value in the **Parameter Definition** box and click **OK**. You can also modify the radii by clicking the button next to the **Radius** box. This displays the **Fillet values** dialog. On this dialog, click on different points and change the radius values. Click **OK** to close the **Fillet values** dialog.
8. Click **Preview** to see how the variable radius fillet would look.
9. On the dialog, click **Variation > Cubic** to get a smooth fillet. Click **Variation > Linear** to get a straight transition fillet.

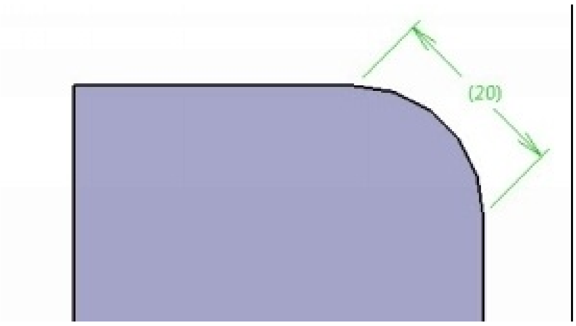




0. Click **OK** to create the variable radius fillet.

Chordal Fillet

This option helps you to create a fillet by specifying its chord length instead of a radius. The chord length is the distance between the endpoints of the fillet profile.

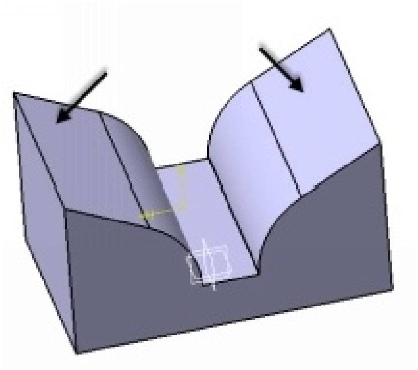


1. On the **Dress-Up Features** toolbar, click **Fillets** drop-down > **Edge Fillet** (or) click **Insert > Dress-Up Features > Edge Fillet** on the Menu bar.
2. On the **Edge Fillet Definition** dialog, click the **Chordal Length** icon available next to the **Radius** box.
3. Click on the edge to fillet.
4. Type-in a value in the **Chordal Length** box.
5. If you want to create a variable radius fillet, then click the **Variable** icon and select multiple points on the edge. Change the chordal length values of the points.

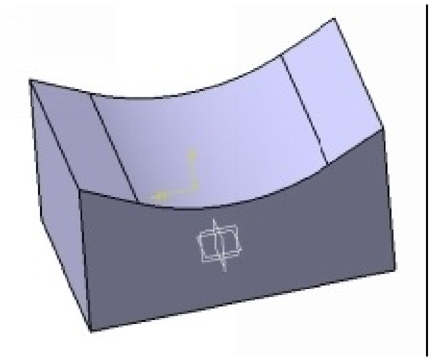
Face-Face Fillet

This command creates a fillet between two faces. The faces are not required to be connected with each other.

1. On the **Dress-Up Features** toolbar, click **Fillets** drop-down > **Face-Face Fillet** (or) click **Insert > Dress-Up Features > Face-Face Fillet** on the Menu bar.
2. Click on two faces.



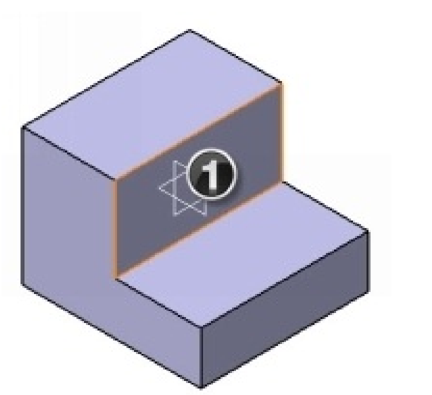
3. Type-in a value in the **Radius** box.
4. Click **OK**.

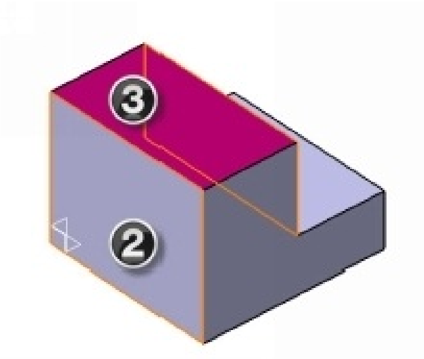


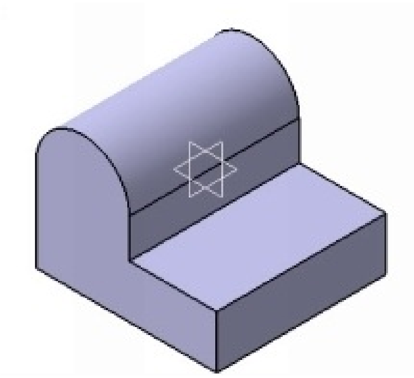
Tritangent Fillet

This command creates a fillet between three faces. It replaces the middle face with a fillet.

1. On the **Dress-Up Features** toolbar, click **Fillets** drop-down > **Tritangent Fillet** (or) click **Insert > Dress-Up Features > Tritangent Fillet** on the Menu bar.
2. Click on three faces of the model geometry.
3. Click **OK** to replace the middle face with a fillet.







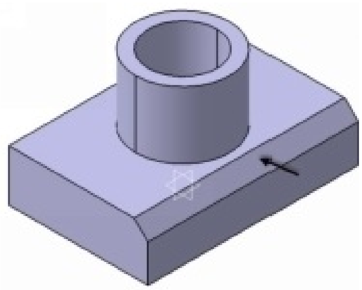
The Chamfer command

The **Chamfer** and **Edge Fillet** commands are commonly used to break sharp edges. The difference is that the **Chamfer** command adds a bevel face to the model. A chamfer is also a placed feature.

1. On the **Dress-Up Features** toolbar, click the **Chamfer** button (or) click **Insert > Dress-Up Features > Chamfer** on the Menu bar.
2. On the **Chamfer Definition** dialog, select chamfer **Mode**. You can select **Length1/Angle**, **Length1/Length2**, **Chordal Length/Angle**, or **Height/Angle**.

3. If you select **Mode > Length1/Angle**, then type-in the length and angle values of the chamfer.
4. Click on the edge(s) to chamfer.

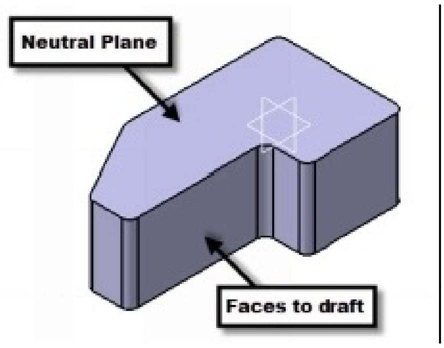
5. Click **OK**.



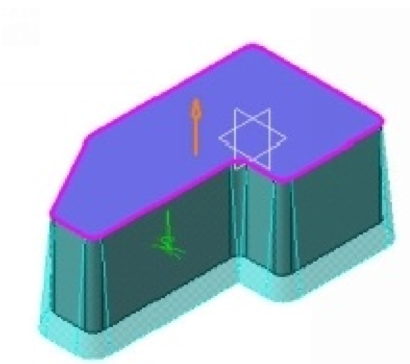
Draft Angle

When creating cast or plastic parts, you are often required to add draft on them so that they can be molded. A draft is an angle or taper applied to the faces of parts to make it easier to remove them from a mold. When creating *Drafted Filleted Pad* features, you can predefine the draft angle. However, most of the time, it is easier to apply the draft after the features are created.

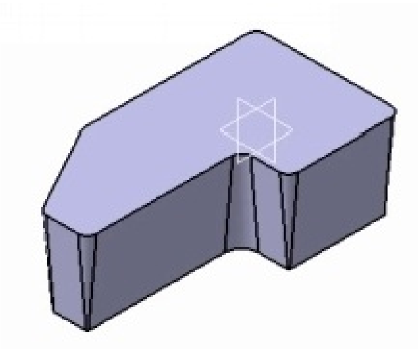
1. On the **Dress-Up Features** toolbar, click **Drafts** drop-down > **Draft Angle** (or) click **Insert > Dress-Up Features > Draft** on the Menu bar.
2. On the **Draft Definition** dialog, select **Draft Type > Constant**.
3. Click on the faces to draft.
4. Under the **Neutral Element** section, click in the **Selection** box and select a flat face or plane. This defines the neutral plane. The draft angle will be measured with reference to this face.



5. Type-in a value in the **Angle** box. This defines the draft angle.
6. Click **Preview**.

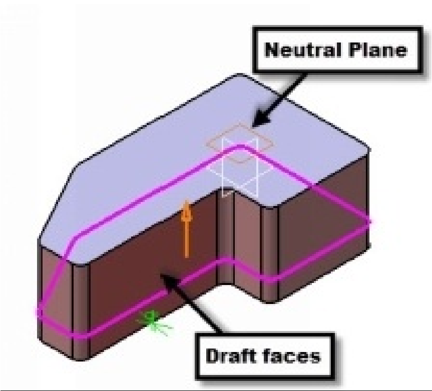


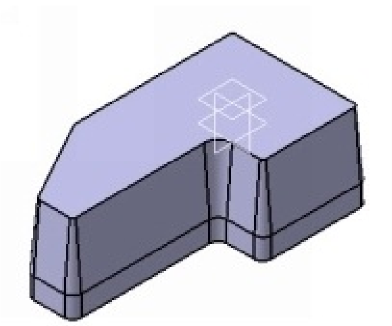
7. If you want to reverse the draft direction, then click the arrow that appears on the geometry.
8. Click **OK** to apply draft.



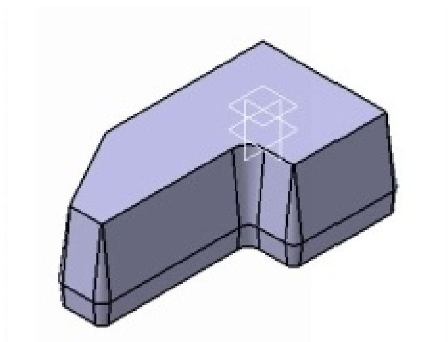
Parting=Neutral

If you want to add draft only up to a certain height, then create a plane at that height. Activate the **Draft Angle** command and select the faces to draft. Click in the **Neutral Plane** selection box and select the plane. On the dialog, click the **More** button and check the **Parting=Neutral** option.





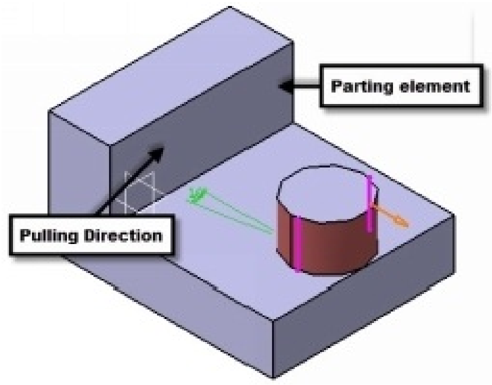
If you check the **Draft both sides** option, the draft will be applied on both sides of the parting plane.



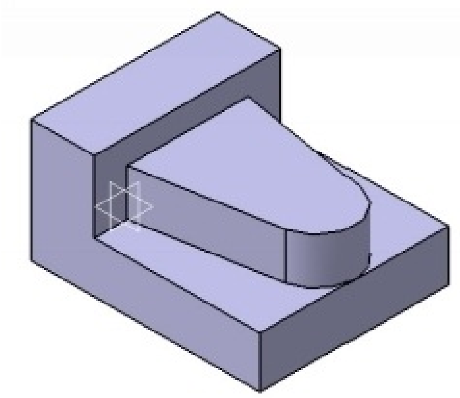
Draft Reflect Line

This command creates a draft by using the silhouette edges (reflected lines) of the curved feature.

1. On the **Dress-Up Features** toolbar, click **Drafts** drop-down > **Draft Reflect line** (or) click **Insert > Dress-Up Features > Draft reflect line** on the Menu bar.
2. Select a curved face to define the drafting face.
3. Click in the **Pulling Direction** selection box and select a flat face or plane. This defines the pulling direction.
4. Click the **More** button on the dialog and check the **Define parting element** option.
5. Select a flat face or plane to define the parting element.



6. Type-in a value in the **Angle** box, and then click **OK**.



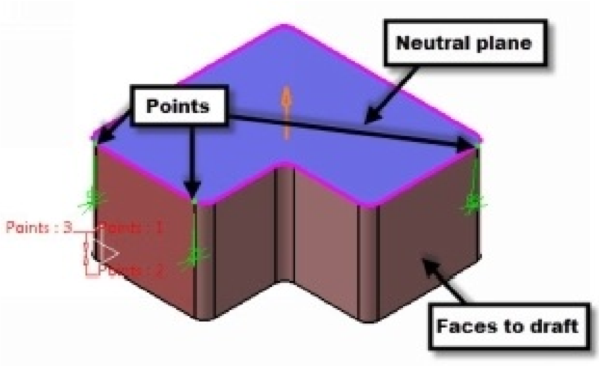
Variable Angle Draft

This command creates a variable angle draft.

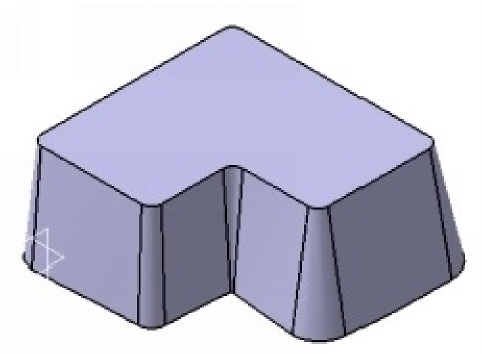
1. On the **Dress-Up Features** toolbar, click **Drafts** drop-down > **Variable Angle Draft** (or) click **Insert > Dress-Up Features > Variable Angle Draft** on the Menu bar.
2. Click on the face to draft. If you select multiple faces, you have to make sure that they are connected, tangentially.
3. Click in the **Neutral element** box and select a flat face or plane.
4. Click in the **Points** box and select multiple points to define different angles. You will

notice that angle values appear on the selected points.

5. Double-click on the angle values to change them.



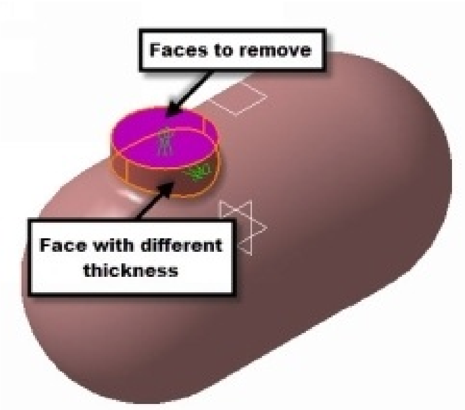
6. Click **OK**.



Shell

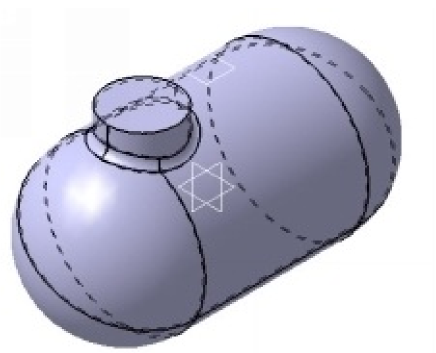
The **Shell** is another useful feature that can be applied directly to a solid model. It allows you to take a solid geometry and make it hollow. This can be a powerful and timesaving technique, when designing parts that call for thin walls such as bottles, tanks, and containers. This command is easy to use.

1. You should have a solid part, and then activate this command from the **Dress-Up Features** toolbar (or) click **Insert > Dress-Up Features > Shell** on the Menu bar.
2. Select the faces to remove.
3. Type-in the wall thickness in the **Default inside thickness** box.
4. If you want to add outside thickness, then type-in a value in the **Default inside thickness** box.
5. If you want add different thickness to some faces, then click in the **Other thickness faces** box, and then select the faces to add different thickness. You will notice that a thickness value appears on the selected face. Double-click on the value and change it.



6. Click **OK** to finish the feature.

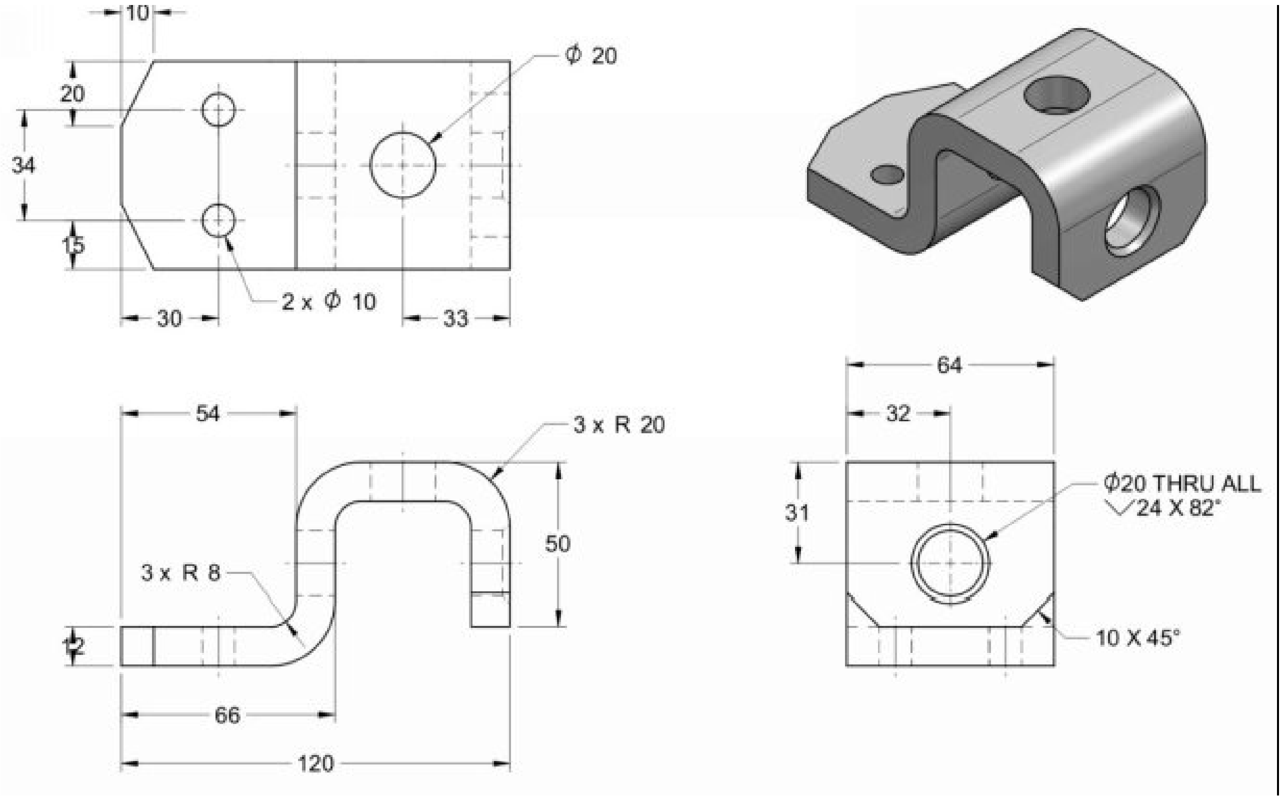
If you want to shell the solid body without removing any faces, then simply type-in a value in the **Default inner thickness** box and click **OK**. This creates the shell without removing the faces. Change the **Render style** to **Wireframe** or **Shading with Edges and Hidden Edges** to view the shell.



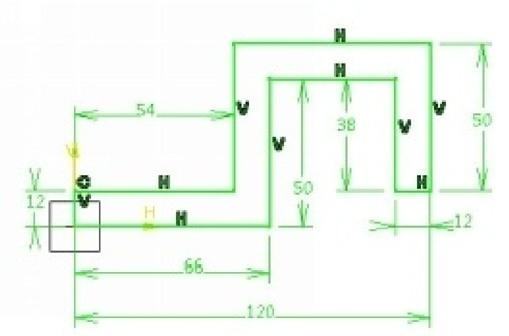
Examples

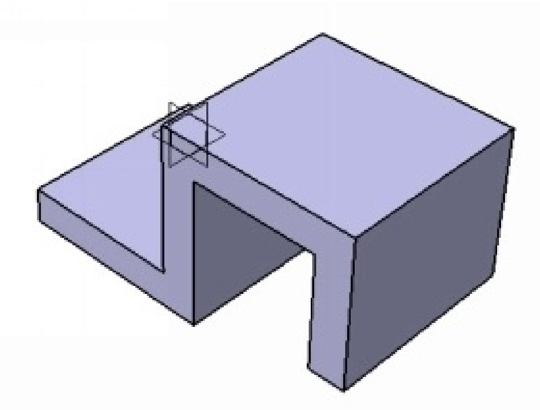
Example 1

In this example, you will create the part shown next.

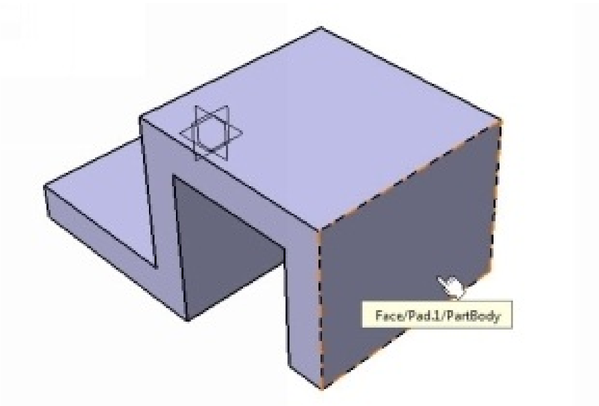


1. Start **CATIA V5-6R2015**.
2. On the **Standard** toolbar, click **New**. On the **New** dialog, select **List of Types > Parts**. Click **OK**.
3. On the **New Part** dialog, type-in **C04-Example1** and check the **Enable hybrid design** option. Click **OK**.
4. On the **Sketch Based Features** toolbar, click the **Pad** icon.
5. On the **Pad Definition** dialog, click the **Sketch** icon and select the YZ plane. Draw the sketch shown in figure and create the pad feature of 64 mm thickness.

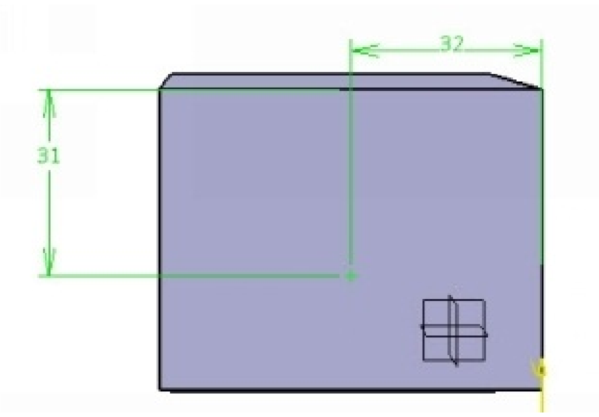




6. On the **Sketcher** toolbar, click the **Sketch** icon, and then select the right-side face.

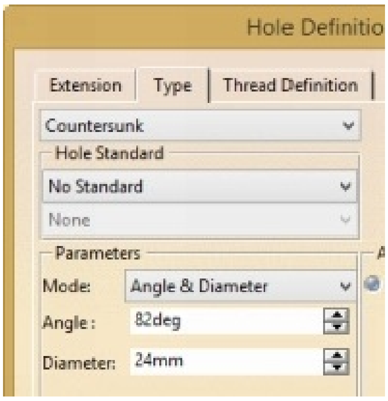


7. On the **Profile** toolbar, click **Points** drop-down > **Point by clicking** and place a point.
8. Add dimensions to locate the point.

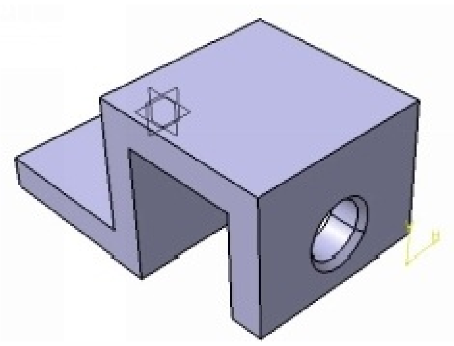


9. Click **Exit Workbench**.

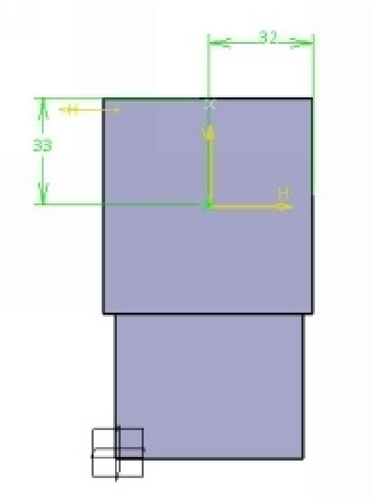
10. On the **Sketch Based Features** toolbar, click the **Hole** icon and select the sketch point.
11. Click on the right-side face.
12. On the **Hole Definition** dialog, select **Extension > Up to Last**.
13. Type-in 20 in the **Diameter** box.
14. Click the **Type** tab and **Countersunk** from the drop-down.
15. Under the **Parameters** section, select **Mode > Angle & Diameter**.
16. Set the **Angle** and **Diameter** values to **82** and **24**, respectively.



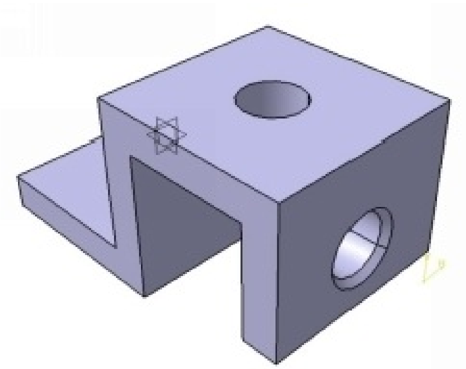
17. Click **OK** to complete the hole feature.



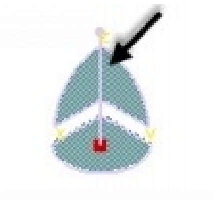
18. Activate the **Hole** command and click on the top face of the part geometry.
19. On the **Hole Definition** dialog, click the **Positioned Sketch** icon to activate the **Sketcher** Workbench.
20. Add dimensions to the sketch point and exit the **Sketcher** Workbench.



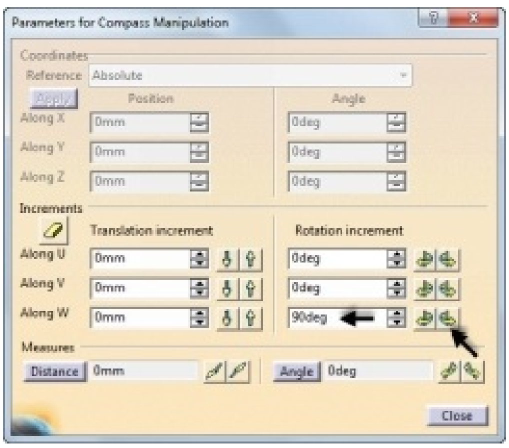
21. On the **Hole Definition** dialog, click the **Type** tab and select **Simple** from the drop-down.
22. Click **OK** to close the dialog.



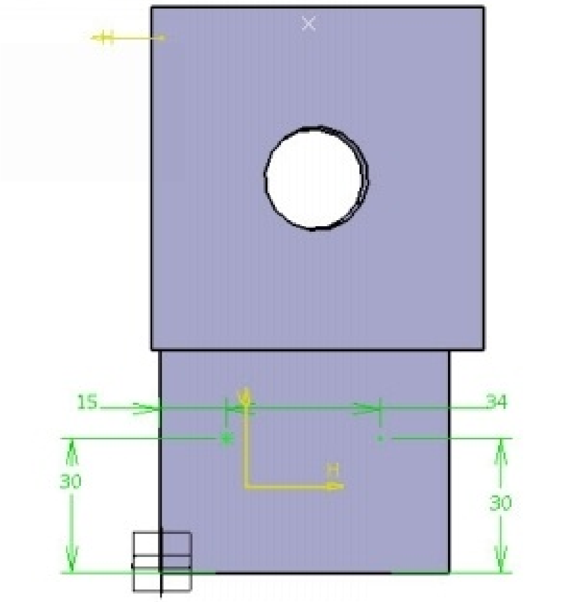
23. On the top-right corner of the graphic window, double-click the Z-axis of the Compass.



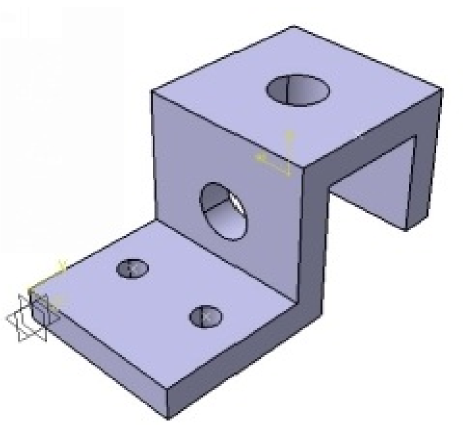
24. On the **Parameters for Compass Manipulation** dialog, type-in **90** in the **Rotation Increment** box along the W axis.
25. Click the rotation in positive direction button. Click **Close** to close the dialog.



26. On the **View** toolbar, click the **Fit All In** icon to fit the geometry in the graphics window.
27. Activate the **Sketch** command and click on the lower top face of the model geometry.
28. Activate the **Points by clicking** command and place two points.
29. Add dimensions to position the points.



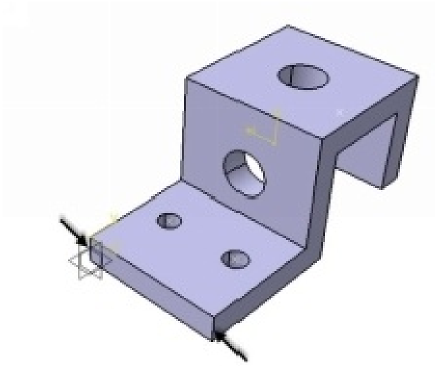
30. Exit the **Sketcher** Workbench.
31. Activate the **Hole** command and select anyone of the sketch points.
32. Click on the lower top face of the model geometry.
33. On the **Hole Definition** dialog, select **Extension > Up To Last**.
34. Type-in **10** in the **Diameter** box and click **OK**.
35. Create another hole using the remaining sketch point.



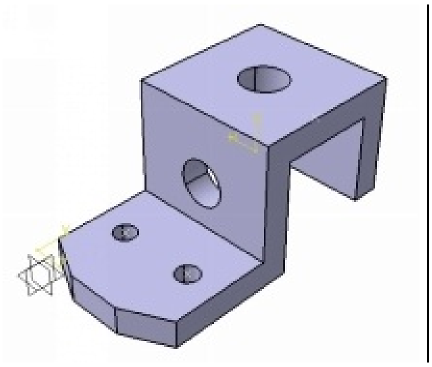
Chamfer Edges

1. On the **Dress-Up Features** toolbar, click the **Chamfer** icon (or) click **Insert > Dress-Up Features > Chamfer**.
2. On the **Chamfer Definition** dialog, select **Mode > Length1/Length2**.
3. Set the **Length1** and **Length 2** values to **20** and **10**, respectively.

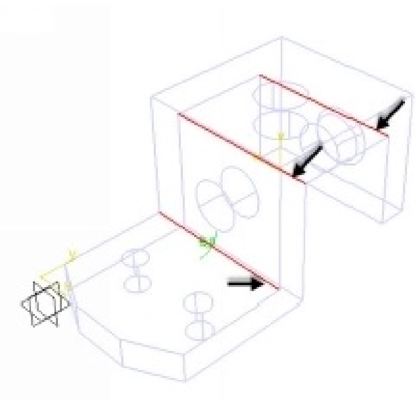
4. Click on the side vertical edges, as shown in figure.



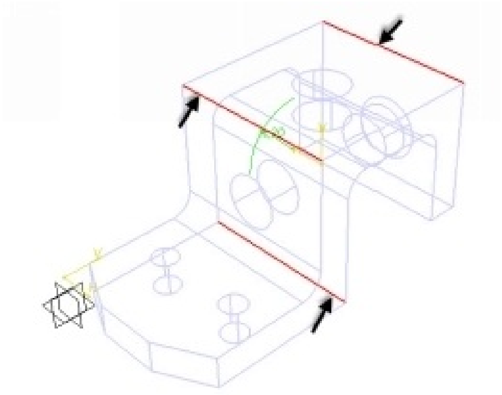
5. Click **OK** to apply chamfers.



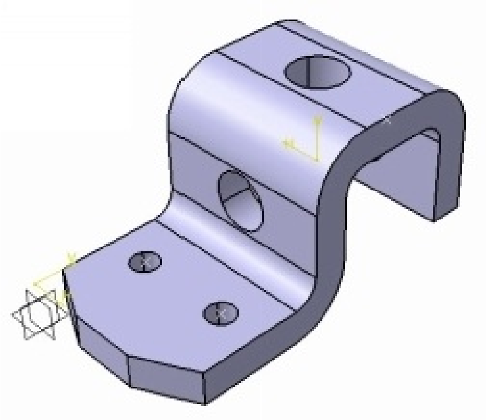
6. On the **Dress-Up Features** toolbar, click the **Edge Fillet** icon (or) click **Insert > Dress-Up Features > Edge Fillet**.
7. On the **Edge Fillet Definition** dialog, type-in **8** in the **Radius** box.
8. On the **View** toolbar, click **View Mode** drop-down > **Wireframe (NHR)**.
9. Click on the horizontal edges of the geometry, as shown below.



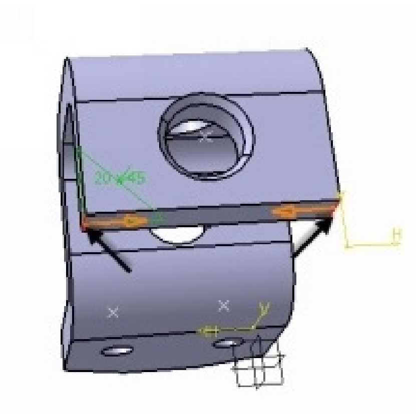
10. Click **OK** to add edge fillets.
11. Activate the **Edge Fillet** command and type-in 20 in the **Radius** box.
12. Click on the outer edges of the model, as shown below. Click **OK** to complete the edge fillet feature.



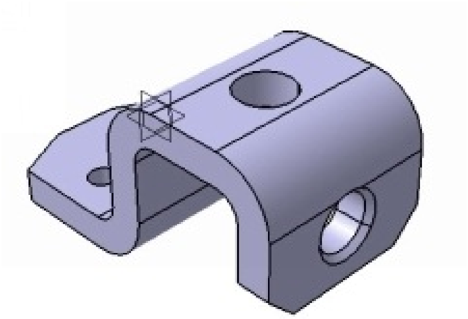
3. On the **View** toolbar, click **View Mode** drop-down > **Shading with Edges**.



4. On the **View** toolbar, click **Quick View Drop-down > Isometric View** to change the orientation of the model view to Isometric.
5. On the **Dress-Up Features** toolbar, click the **Chamfer** icon (or) click **Insert > Dress-Up Features > Chamfer**.
6. On the **Chamfer Definition** dialog, select **Mode > Length1/Angle**.
7. Click on the lower corners of the part geometry.



8. Type-in **10** and **45** in the **Length** and **Angle** boxes, respectively. Click **OK** to chamfer the edges.



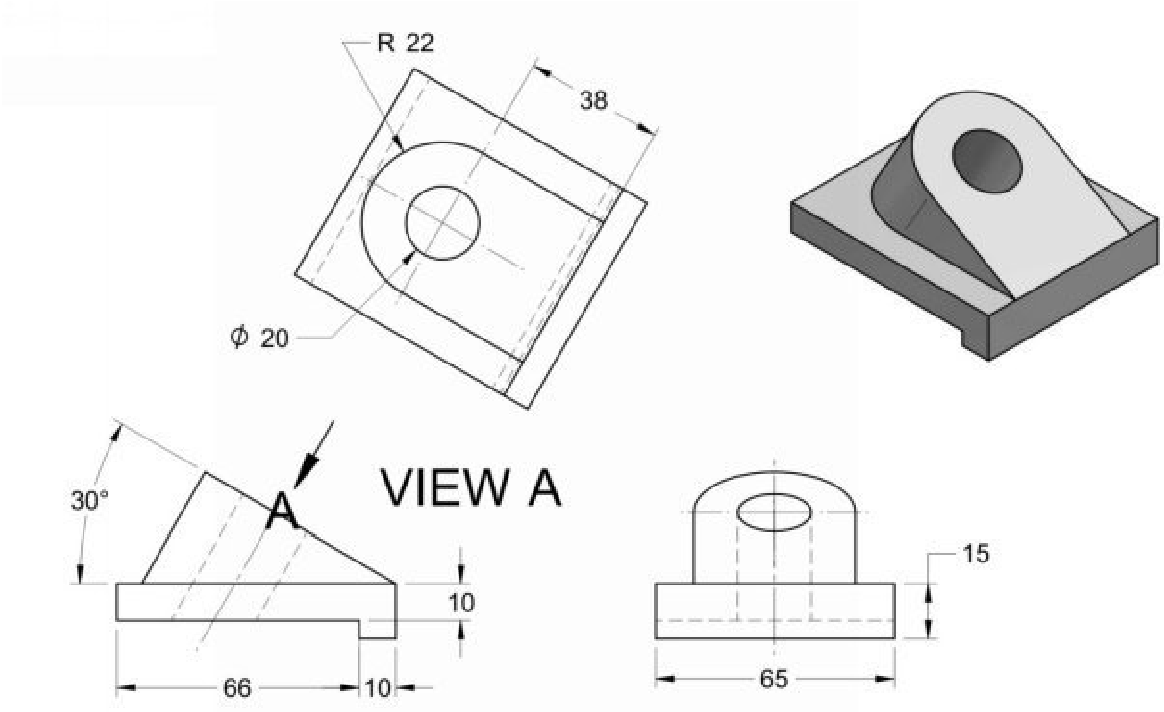
9. Save and close the file.

Questions

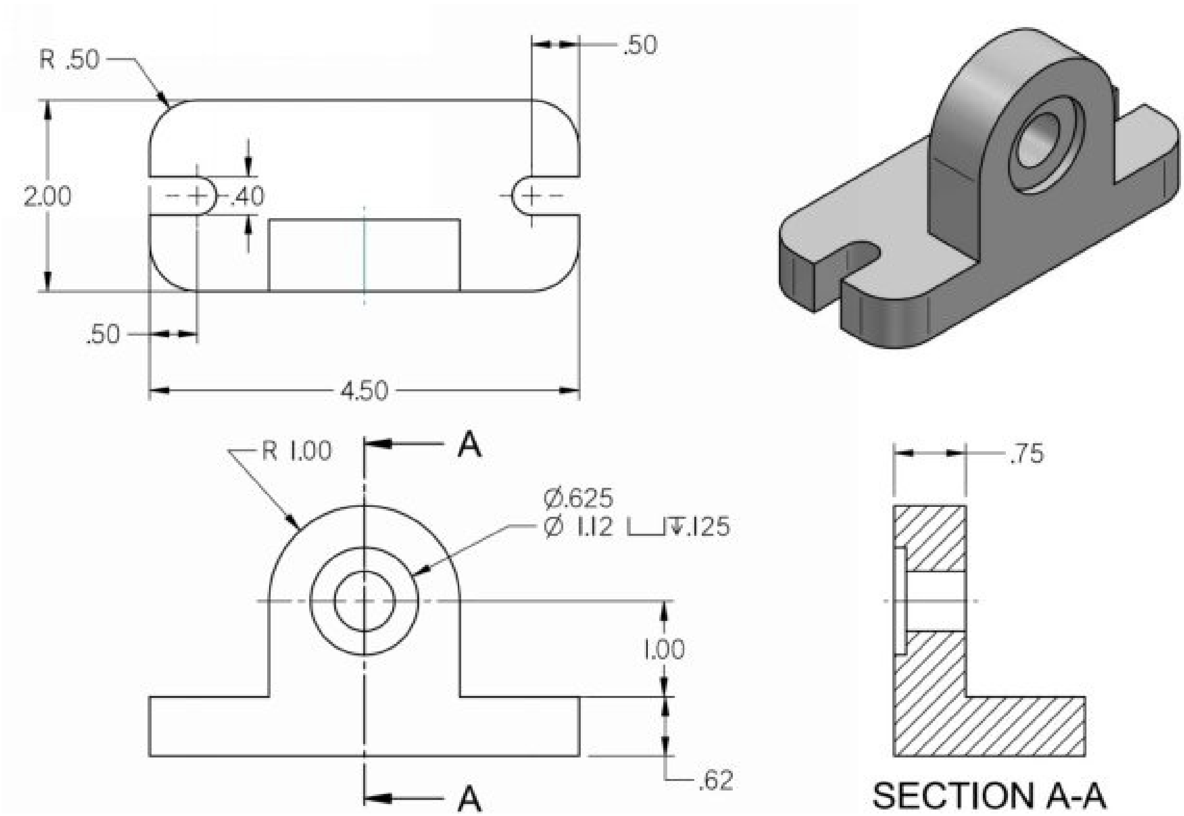
1. What are **Dress-Up** features?
2. Which option allows you to create a chamfer with unequal setbacks?
3. Which option allows you create a variable radius blend?
4. When you create a thread on a cylindrical face, the thread diameter will be calculated automatically or not?

Exercises

Exercise 1


R 22
38
Ø 20
30°
A
VIEW A
10
66
10
15
65

Exercise 2


R .50
.50
2.00
.40
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4.50
R 1.00
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Ø.625
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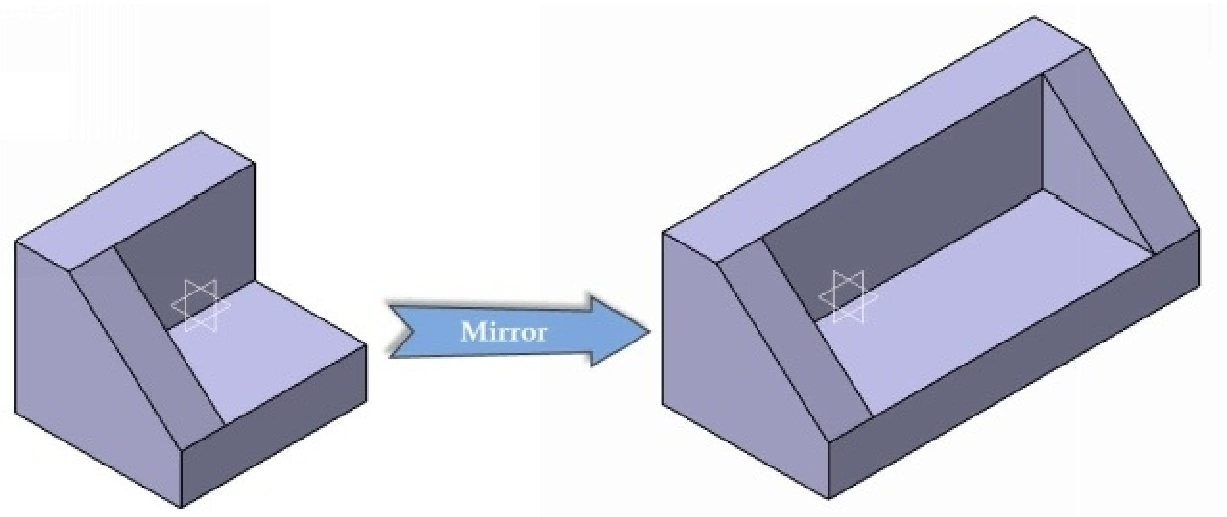
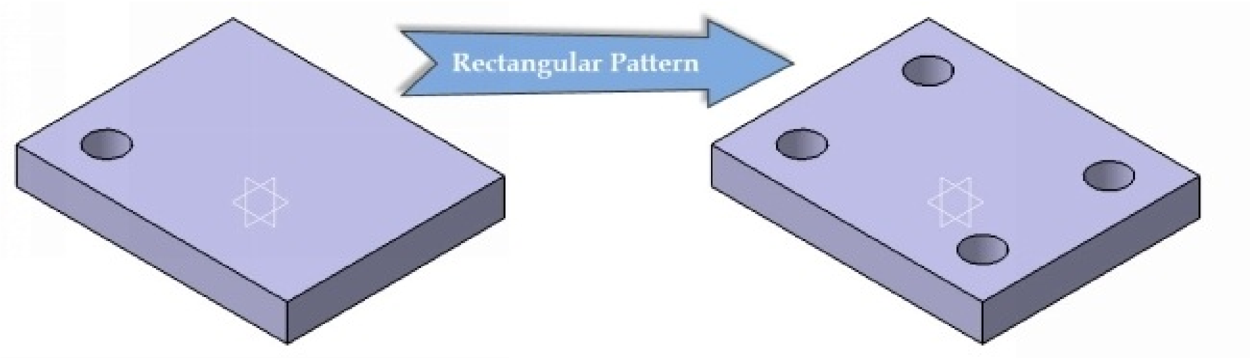
1.00
.62
A
.75
SECTION A-A

Chapter 5: Patterned Geometry

When designing a part geometry, oftentimes there are elements of symmetry in each part or there are at least a few features that are repeated multiple times. In these situations, CATIA V5 offers you some commands that save your time. For example, you can use mirror features to design symmetric parts, which makes designing the part quicker. This is because you only have to design a portion of the part and use the mirror feature to create the remaining geometry.



In addition, there are some transformation commands to replicate a feature throughout a part quickly. They save you time from creating additional features individually and help you to modify the design easily. If the design changes, you only need to change the first feature and the rest of the pattern features will update, automatically. In this chapter, you will learn to create mirrored and pattern geometries using the commands available in CATIA V5.



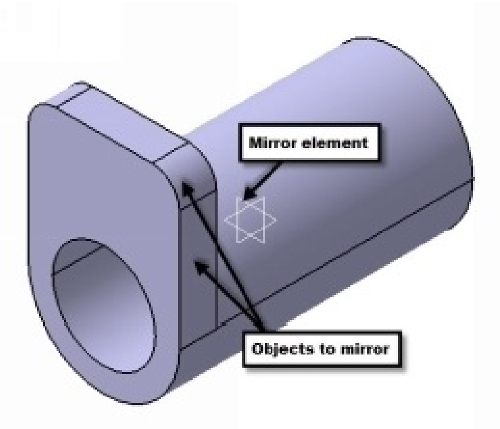
The topics covered in this chapter are:

- *Mirror* features
- *Rectangular Patterns*
- *Circular Patterns*
- *User Patterns*
- *Fill Patterns*

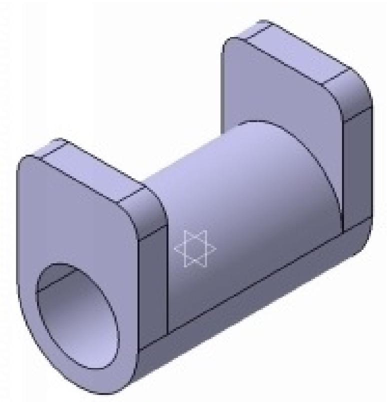
 The Mirror command

If you are designing a part that is symmetric, you can save time by using the **Mirror Feature** command. Using this command, you can replicate individual features of the entire body. To mirror features (3D geometry), you need to have a face or plane to use as a mirroring element. You can use a model face, default plane, or create a new plane, if it does not exist where it is needed.

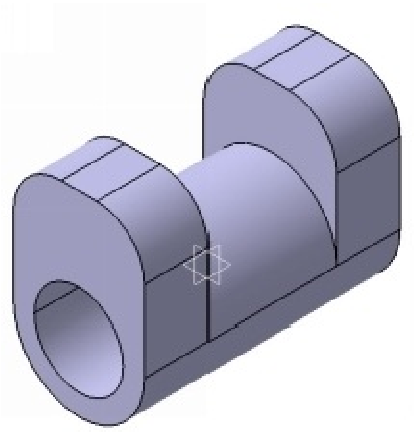
1. On the **Transformation Features** toolbar, click the **Mirror** button (or) click **Insert > Transformation Features > Mirror** on the Menu bar.
2. Select the reference plane about which the features are to be mirrored.
3. Click in the **Object to mirror** box and click on the features to mirror.



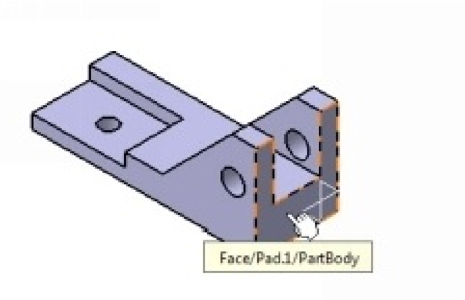
4. Click **OK**.

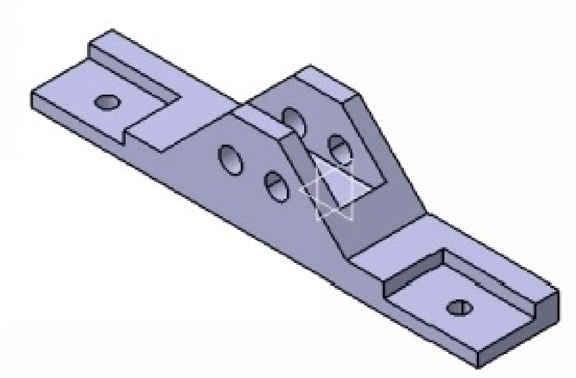


Now, if you make changes to original feature, the mirror feature will be updated automatically.



If the part you are creating is completely symmetric, you can save more time by creating half of it and mirroring the entire geometry rather than individual features. Activate the **Mirror** command and click on the mirror element. As the solid body is selected by default, you just need to click **OK** to mirror the complete body.

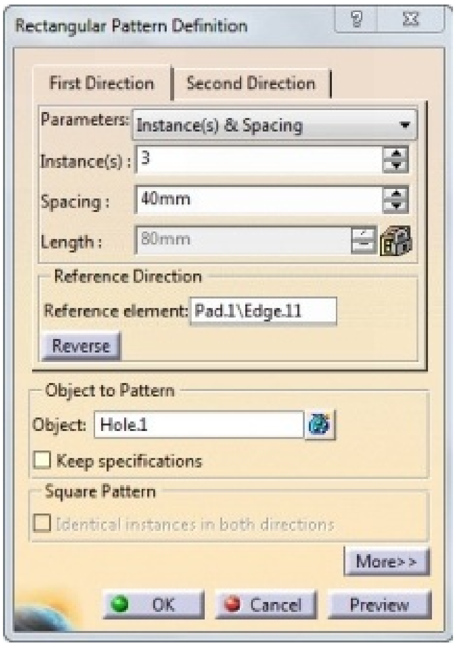




Rectangular Pattern

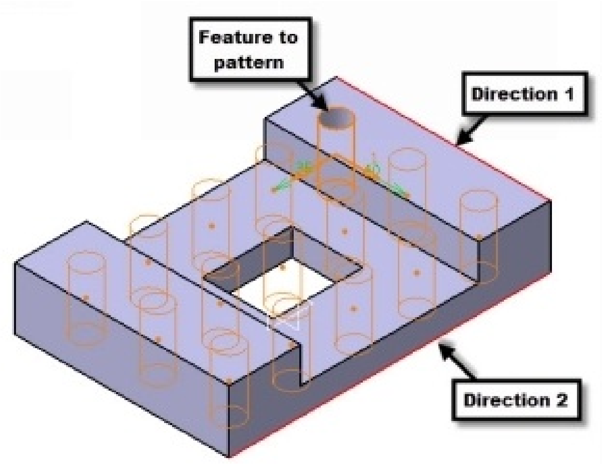
This command replicates a feature using a rectangular layout.

1. On the **Transformation Features** toolbar, click **Patterns** drop-down > **Rectangular Pattern** (or) click **Insert > Transformation Features > Rectangular Pattern** on the Menu bar.
2. On the **Rectangular Pattern Definition** dialog, click in the **Object** box and select the object to pattern.
3. Click in the **Reference element** box and click on an edge to define the first direction of the rectangular pattern. You will notice that a pattern preview appears on the model.
4. Now, select **Parameters > Instance(s) & Spacing** on the dialog and set the parameters of the pattern (**Instance(s)** and **Spacing**).
5. Click the **Reverse** button, if you want to reverse the pattern direction.

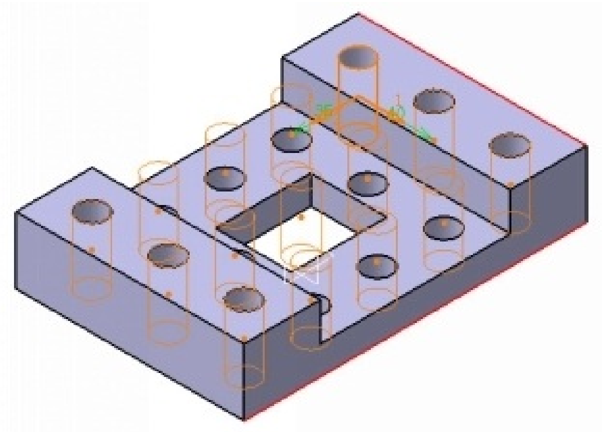


6. Click the **Second Direction** tab on the dialog.
7. Click in the **Reference element** box and click on an edge to define the second direction of the pattern.

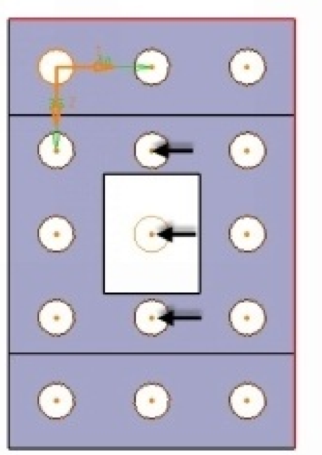
3. Set the parameters (**Instance(s)** and **Spacing**) of pattern in the second direction.
). Click the **Reverse** button, if you want to reverse the pattern direction.

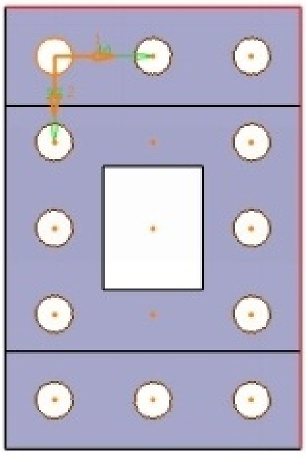


0. Click **Preview**.



If you want to suppress an instance of the pattern, then click on the orange dot on it. This suppresses the instance. If you want to unsuppress the instance, then click the orange dot again.

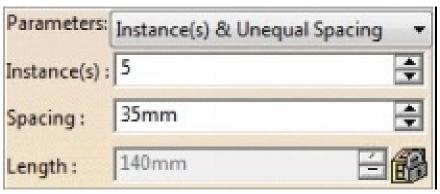




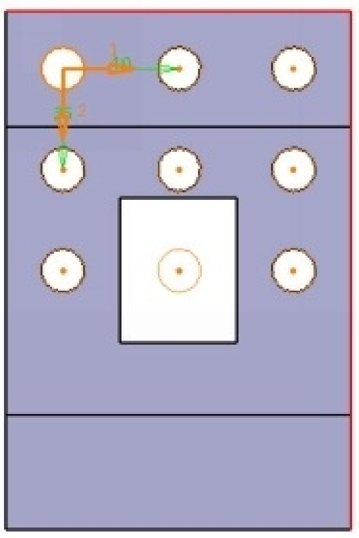
Select **Parameters > Instance(s) & Length** on the **Rectangular Pattern Definition** dialog, if you want to enter instance(s) and total length along the direction 1 or direction 2.

Select **Parameters > Spacing & Length**, if you want to enter the distance between individual instances of the pattern and total length along the directions.

Select **Parameters > Instance(s) & Unequal Spacing** to define different spacing values for each instance. You will notice that individual spacing values appear on the pattern. Double-click on these values and change them.

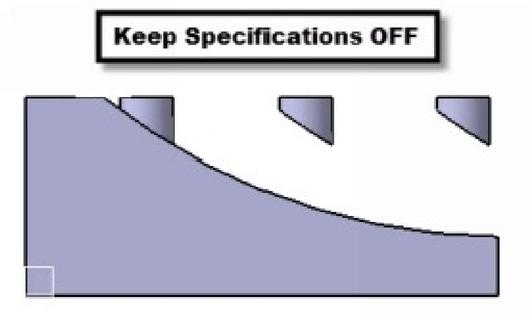


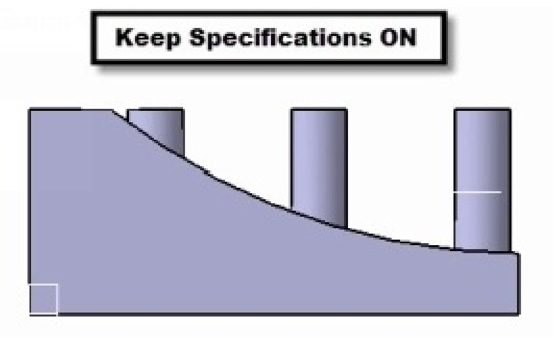
If you want to create a square pattern, then check **Identical instances in both directions** under the **Square Pattern** section.



Keep Specifications

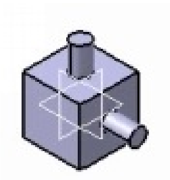
This option creates the rectangular pattern by keeping the specifications of the original feature. For example, if the original feature of pattern is created by extruding up to an irregular surface, then the **Keep Specifications** option creates all the features of the rectangular pattern by maintaining the extrusion condition.

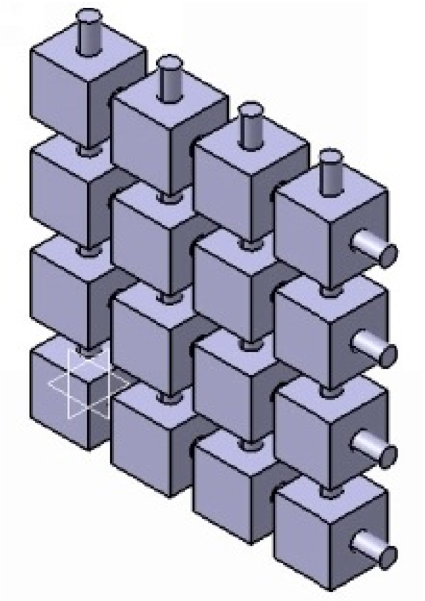




Patterning the entire geometry

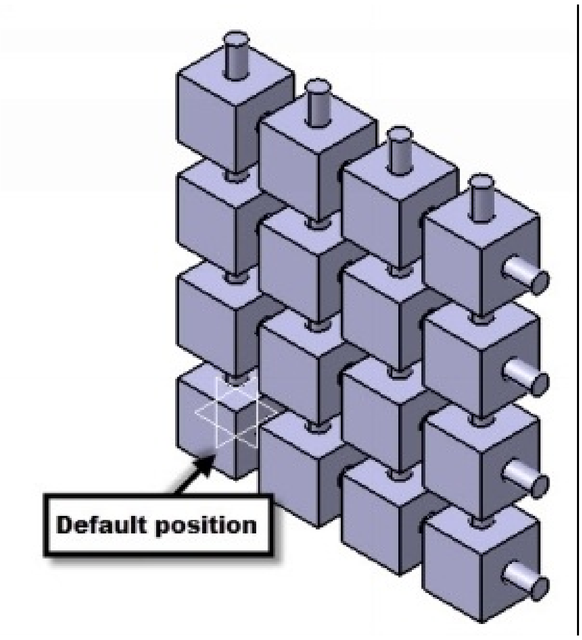
Patterning the entire geometry of a part is easier than patterning features. You need to activate the **Rectangular Pattern** command and define the direction, instant count, and spacing between the instances. There is no need to select the geometry as it is selected by default.

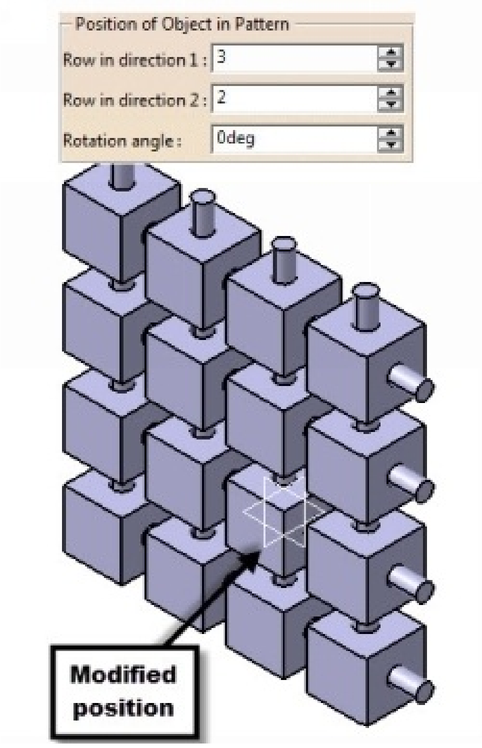




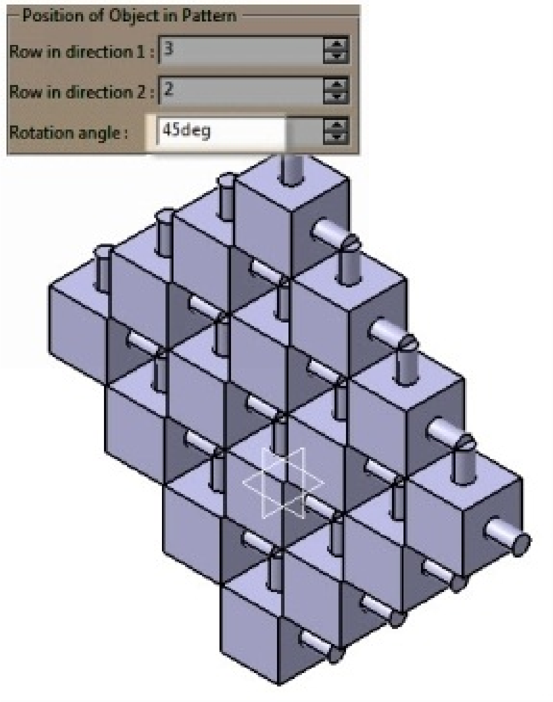
Position of Object in Pattern

On the **Rectangular Pattern Definition** dialog, there is an option to change the position of the original feature/body. Expand the **Rectangular Pattern Definition** dialog type-in values in **Row in direction 1** and **Row in direction 2** boxes. The position of the origin feature/body changes.



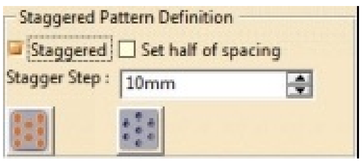


You can also change the angular position of the pattern feature/body by typing-in a value in the **Rotation angle** box.

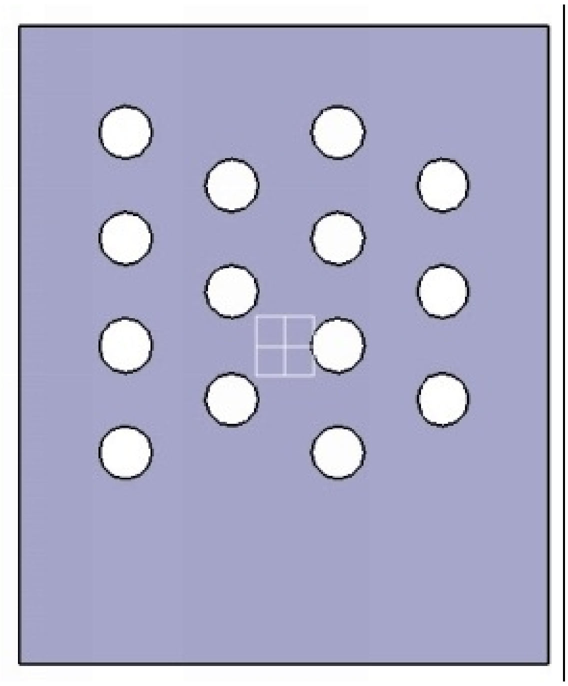


Staggered Pattern Definition

The **Rectangular Pattern** command has an option to create staggered pattern. In a staggered pattern, the position of one row is offset from another.



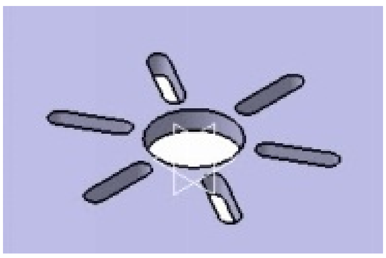
1. To create this kind of pattern, you need to activate the **Rectangular Pattern** command and select the feature to pattern.
2. Define the pattern parameters in two directions (make sure that there are more than two instances in both the directions).
3. Expand the **Rectangular Pattern Definition** dialog and check the **Staggered** option.
4. Type-in the offset value in the **Stagger Step** box (or) check the **Set half of spacing** option.
5. Click anyone of the two buttons available in the **Staggered Pattern Definition** section.
6. Click **OK** to create the staggered pattern.



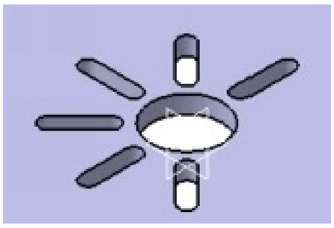
Circular Pattern

This command patterns the selected features in a circular fashion.

1. On the **Transformation Features** toolbar, click **Patterns** drop-down > **Circular Pattern** (or) click **Insert** > **Transformation Features** > **Circular Pattern** on the Menu bar.
2. Click in the **Object** box and select the feature to pattern from the model geometry.
3. Click in the **Reference element** box and select an axis, edge, plane, or cylindrical face. This defines the axis of circular pattern.
4. Select **Parameters** > **Complete Crown**.
5. Type-in a value in the **Instance(s)** box.



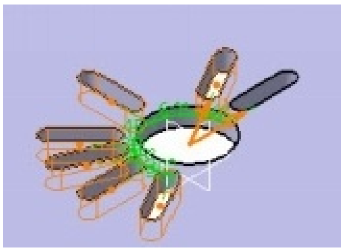
Select **Parameters** > **Instance(s) & angular spacing**, if you want to type-in the instance count and the angle between individual instances.



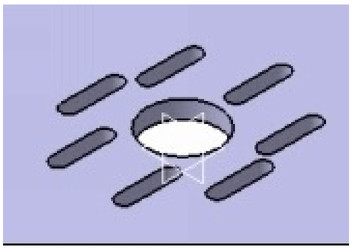
Select **Parameters > Instance(s) & total angle**, if you want to type-in the instance count and total angle. The angle between the instances will be calculated, automatically. For example, enter 5 in the Instance(s) box and 300 in the **Total angle** box. This creates five instances including the original one. The angle between the instances will be 60 (300/5). However, if you enter 6 and 360 in the **Instance(s)** and **Total angle** boxes, an error appears showing that one or more instances overlap each other.

Select **Parameters > Angular spacing & total angle**, if you want to type-in the angle between individual instances and the total angle of the circular pattern.

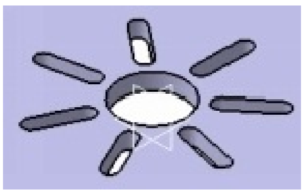
Select **Parameters > Instance(s) & unequal angular spacing**, if you want to define different angular spacing for each instance. You will notice that the individual spacing values appear on the pattern. Double-click on these values and change them.



Under the **Rotation of Instance(s)** section, uncheck **Radial alignment of instance(s)** to pattern the feature with the original orientation.



Check this option to change the orientation of the instances, as they are patterned in the circular fashion.

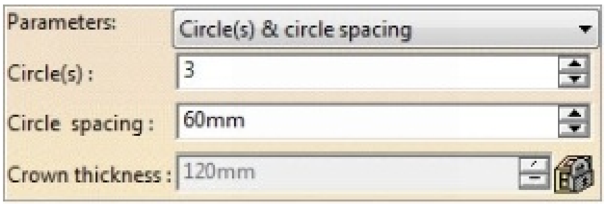


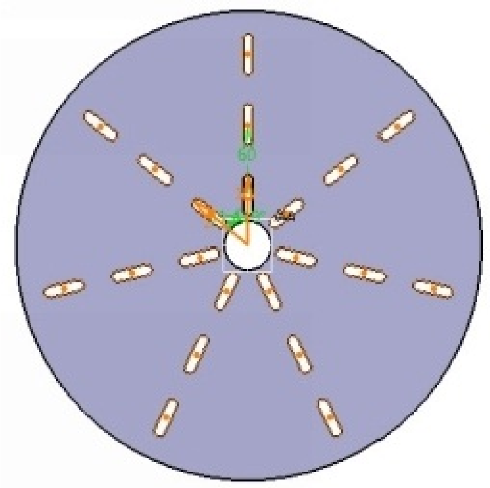
Crown Definition

The **Circular Pattern** command has options to radiate the circular pattern.

1. On the **Circular Pattern Definition** dialog, click the **Crown Definition** tab to view the options to radiate the circular pattern.

2. Select **Parameters > Circle(s) & circle spacing**.
3. Type-in values in the **Circle(s)** and **Circle spacing** boxes.

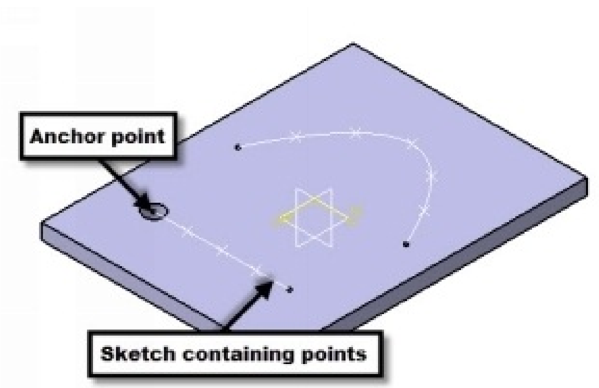




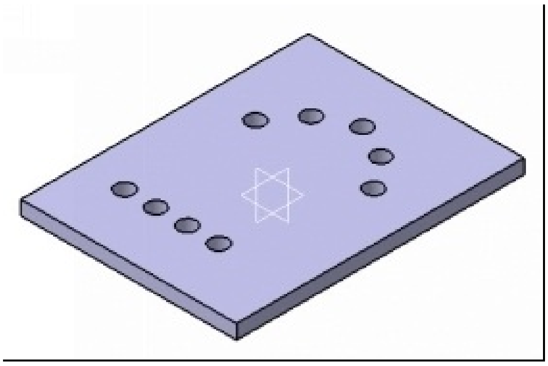
User Pattern

This command patterns the selected features by using user-defined points.

1. On the **Transformation Features** toolbar, click **Patterns** drop-down > **User Pattern** (or) click **Insert > Transformation Features > User Pattern** on the Menu bar.
2. On the **User Defined Pattern Definition** dialog, click in the **Object** box and select the feature to pattern.
3. Click in the **Anchor** box and select a point to define the origin point of the pattern.
4. Click on individual points (or) the sketch containing points.



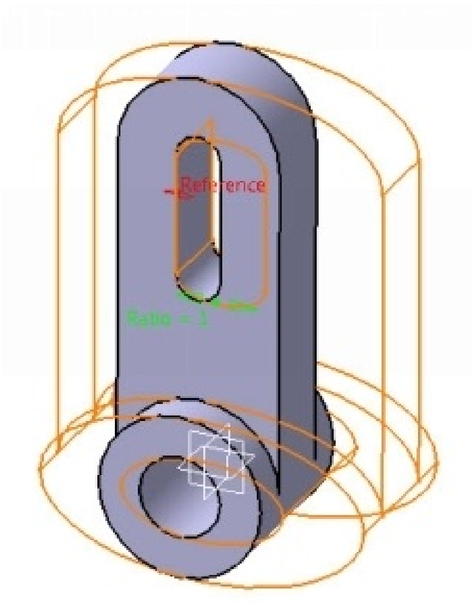
5. Click **OK** to create the pattern.



Scaling

This command scales the part geometry with reference to the face, point, or plane.

1. On the **Transformation Features** toolbar, click **Scale** drop-down > **Scaling** (or) click **Insert** > **Transformation Features** > **Scaling** on the Menu bar.
2. Click on the face or plane to define the scaling direction.
3. Type-in the scaling factor in the **Ratio** box.

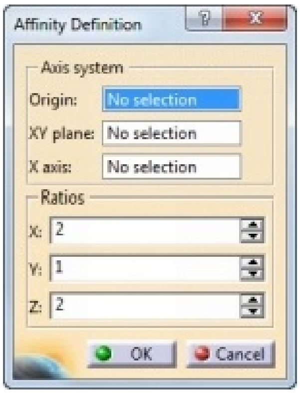


4. Click **OK** to scale the body.

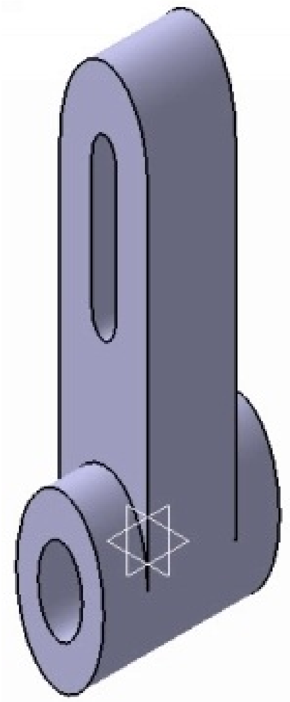
Affinity

This command scales the part geometry along three directions using the coordinate values that you specify.

1. On the **Transformation Features** toolbar, click **Scale** drop-down > **Affinity** (or) click **Insert** > **Transformation Features** > **Affinity** on the Menu bar.
2. On the **Affinity Definition** dialog, type-in values in the X, Y, Z boxes.


Affinity Definition
Axis system
Origin:
No selection
XY plane:
No selection
X axis:
No selection
Ratios
X: 2
Y: 1
Z: 2
OK
Cancel

3. Click **OK**.



Examples

Example 1

In this example, you will create the part shown next.

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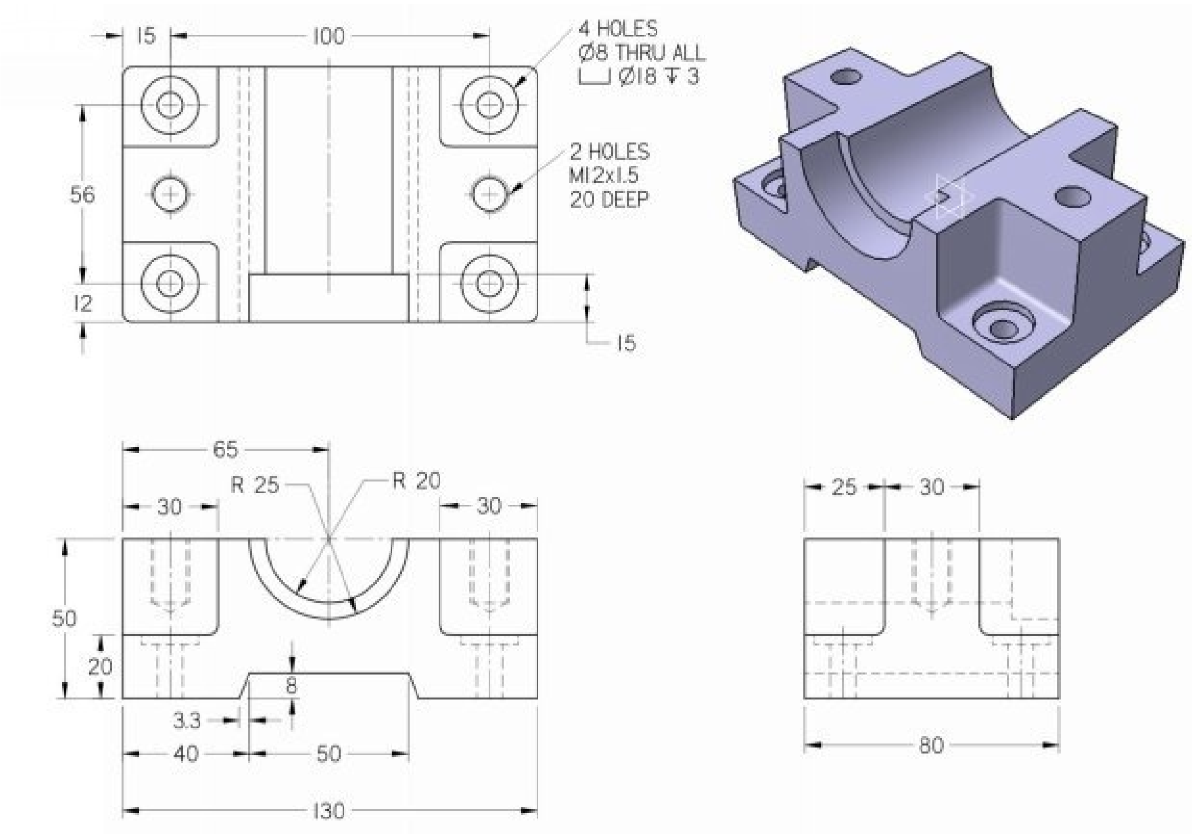
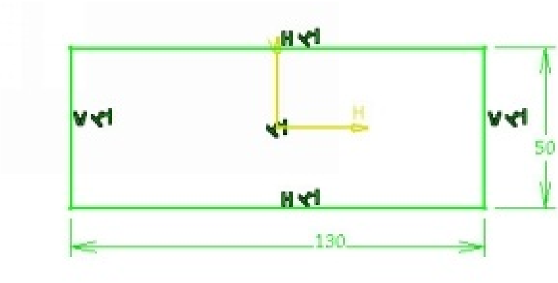
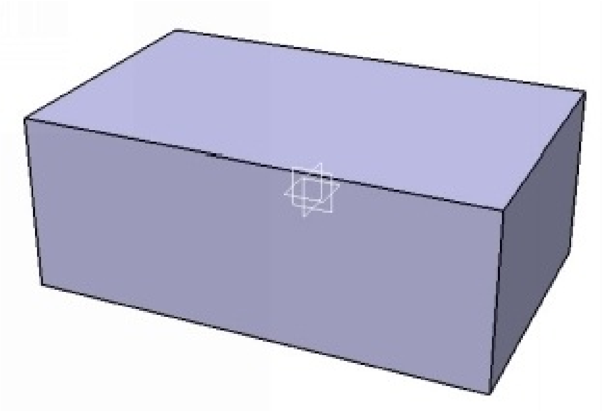


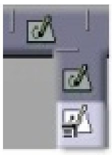
1. Start **CATIA V5-6R2015**.
2. Open a new part file.
3. Activate the **Pad** command and click the **Sketch** icon on the **Pad Definition** dialog.
4. Click on the YZ plane.
5. Create a rectangular sketch, and then click **Exit workbench**.



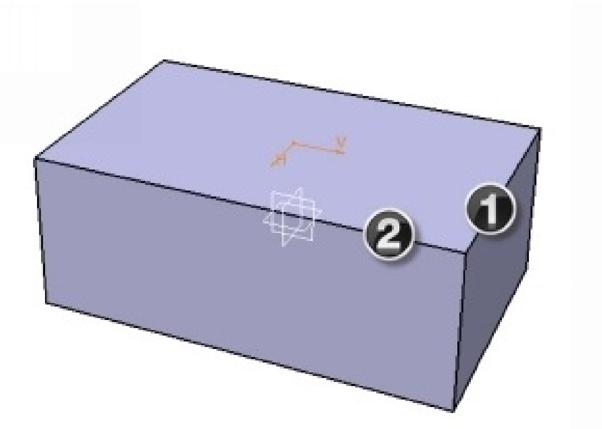
6. On the **Pad Definition** dialog, type-in **40** in the **Length** box.
7. Check the **Mirror extent** option.
8. Click **OK** to complete the *Pad* feature.



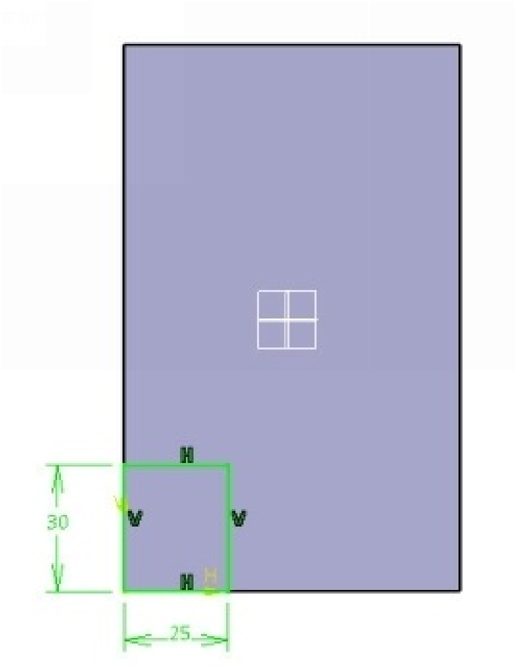
9. On the **Sketcher** toolbar, click **Sketch > Positioned Sketch**.



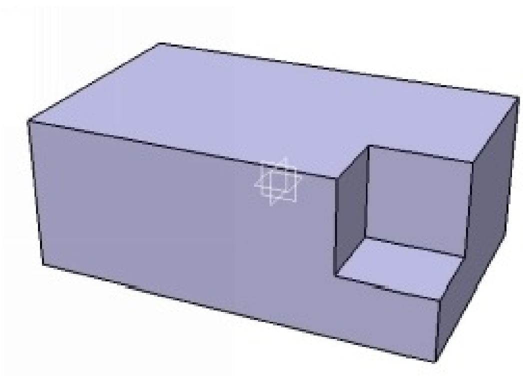
10. Click on the top face of the part geometry.
11. On the **Sketch Positioning** dialog, under the **Origin** section, select **Type > Intersection 2 Lines**.
12. Click on the edges of the part geometry, as shown below.



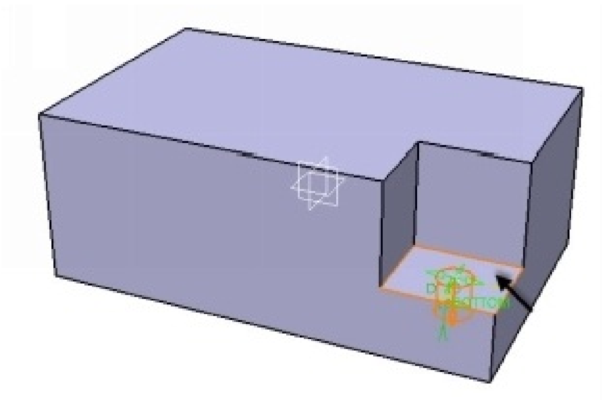
13. On the dialog, check the **Reverse H** and **Reverse V** options.
14. Click **OK** to start the sketch.
15. Create a rectangular sketch and click **Exit workbench**.



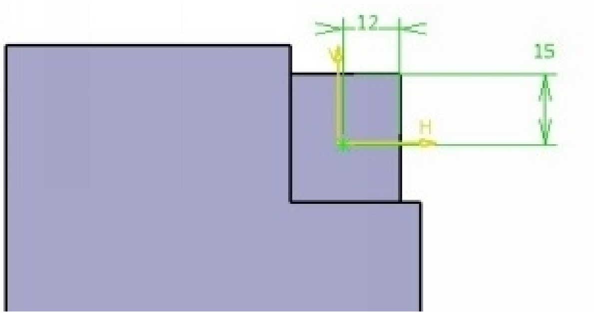
16. Activate the **Pocket** command.
17. Create the *Pocket* feature of **30 mm** depth.



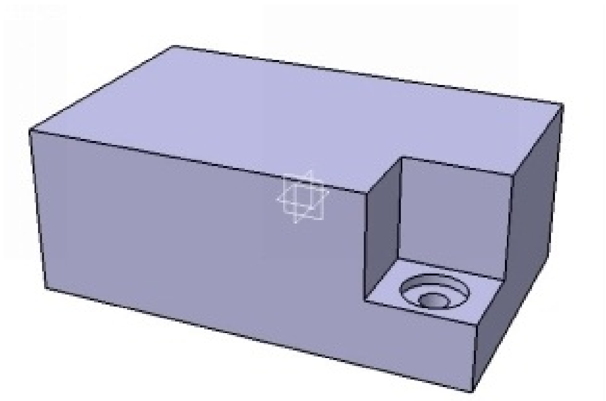
18. Activate the **Hole** command and click on the bottom face of the *Pocket* feature.

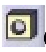


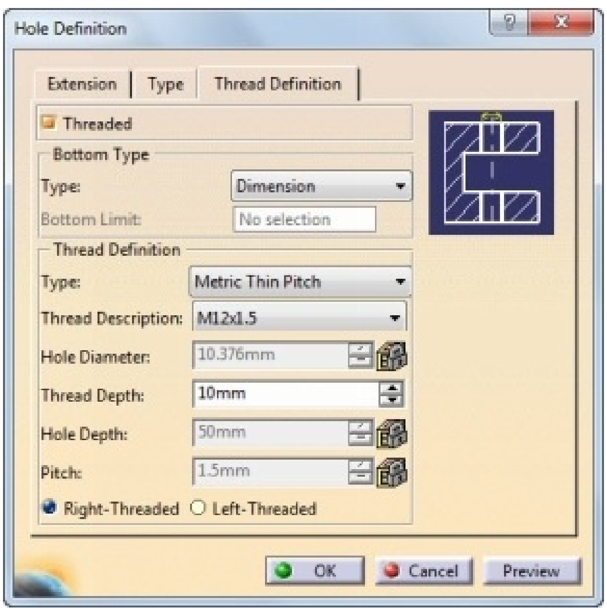
19. On the **Hole Definition** dialog, click **Extension > Up to last**.
20. Set the **Diameter** value to 8.
21. Click the **Type** tab and select **Counterbored** from the drop-down.
22. Type-in **18** and **3** in the **Diameter** and **Depth** boxes, respectively.
23. Click the **Extension** tab, and then click the **Positioning sketch** icon.
24. Apply constraints to position the hole.



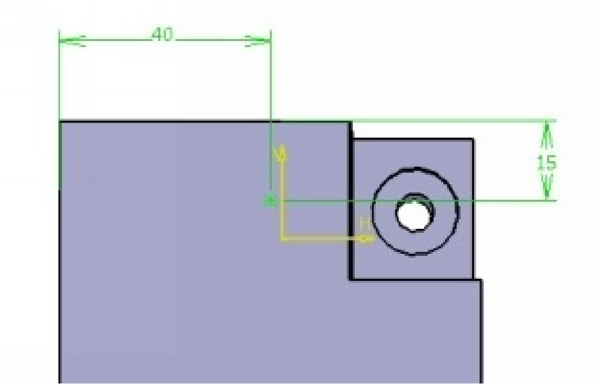
25. Click **Exit workbench**, and then click **OK** to create the counterbored hole.



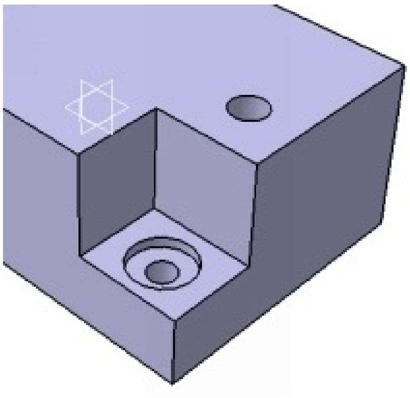
26. Activate the **Hole** command and click on the top face of the part geometry.
27. Click the **Type** tab and select **Simple** from the drop-down.
28. Click the **Thread Definition** tab and check the **Threaded** option.
29. Under the **Thread Definition** section, select **Type > Metric Thin Pitch**.
30. Click **OK** on the **Warning** message.
31. Select **Thread Description > M12X1.5**.



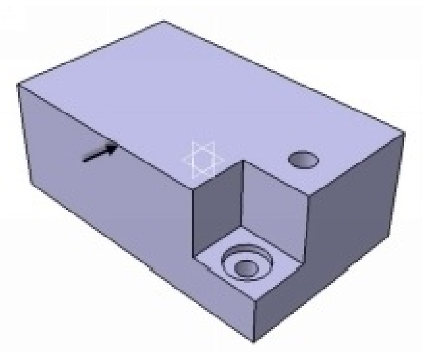
32. Click the **Extension** tab, and the click the **Positioning sketch** icon.
33. Add constraints to define the hole location.



34. Click **Exit workbench**, and then click **OK** to create a threaded hole.

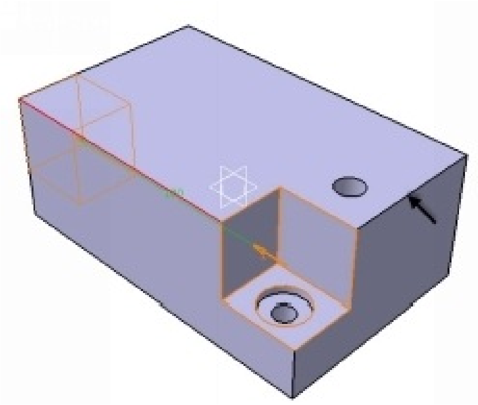


35. On the **Transformation Features** toolbar, click **Pattern** drop-down > **Rectangular Pattern** (or) click **Insert > Transformation Features > Rectangular Pattern** on the Menu.
36. On the **Rectangular Pattern Definition** dialog, under the **First Direction** tab, select **Parameters > Instance(s) & Spacing**.
37. Type-in **2** and **100** in the **Instance(s)** and **Spacing** boxes.
38. Under the **Reference Direction** section, click in the **Reference element** selection box, and then click on the top front edge of the part geometry.

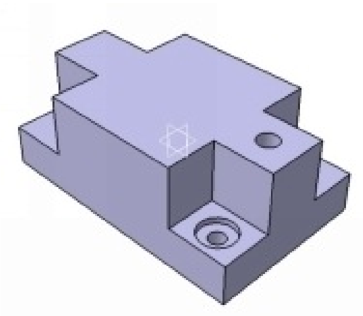


39. Under the **Object to Pattern** section, click in the **Object** selection box and select the *Pocket* feature.
40. Under the **Second Direction** section, type-in **2** and **55** in the **Instance(s)** and **Spacing** boxes, respectively.

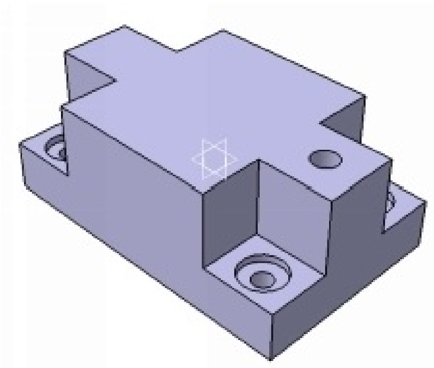
41. Under the **Reference Direction** section, click in the **Reference element** box and click on the top side edge of the part geometry.



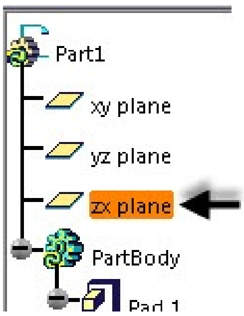
42. Click the **Reverse** button, and then click **OK** to pattern the *Pocket* feature.



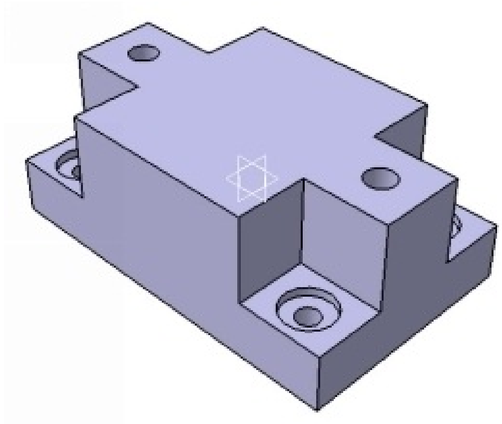
43. Likewise, pattern the counterbored hole. The pattern parameters are same.



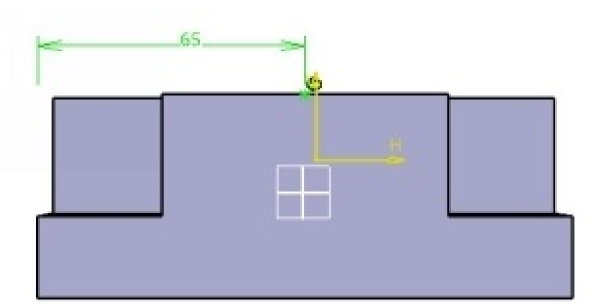
44. On the **Transformation Features** toolbar, click the **Mirror** icon (or) click **Insert > Transformation Features > Mirror** on the Menu.
45. In the Specification Tree, click the **zx plane** to define the mirror plane.



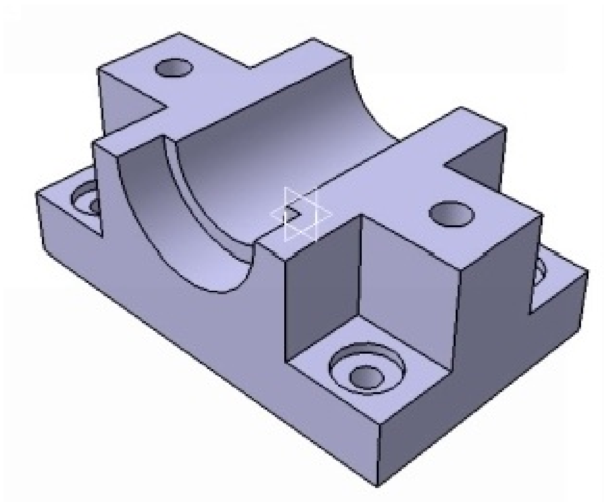
46. On the **Mirror Definition** dialog, click in the **Object to mirror** selection box and select the threaded hole feature.
47. Click **OK** to mirror the selected features.



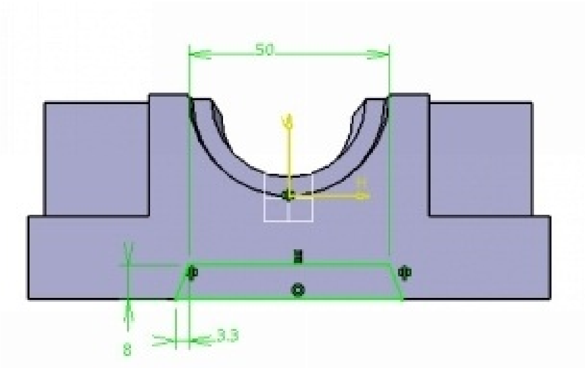
48. Activate the **Hole** command and select the front face of the part geometry.
49. On the **Hole Definition** dialog, click **Extension > Up to last**.
50. Type-in **40** in the **Diameter** box
51. Under the **Type** tab, select **Counterbored** from the drop-down and type **50** and **15** in the **Diameter** and **Depth** boxes, respectively.
52. Click the **Thread Definition** tab and uncheck the **Threaded** option.
53. Under the **Extension** tab, click the **Positioning sketch** icon, and then add constraints to define the location of the hole.

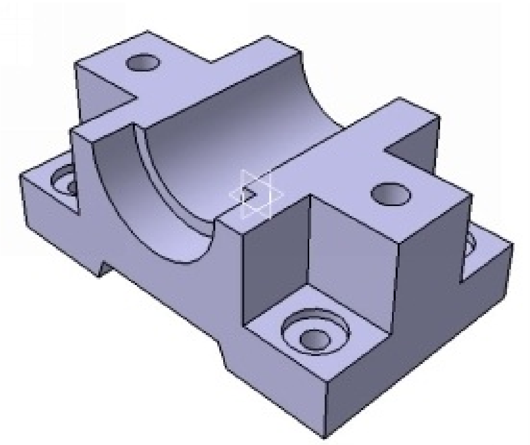


54. Exit the sketch, and then click **OK**.

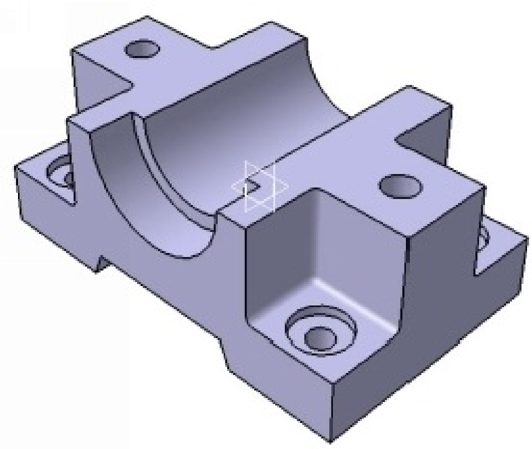


55. Draw a sketch on the front face of the pat geometry and create a *Pocket* throughout the geometry.





56. Fillet the sharp edges of the *Pocket* features. The fillet radius is 2 mm.



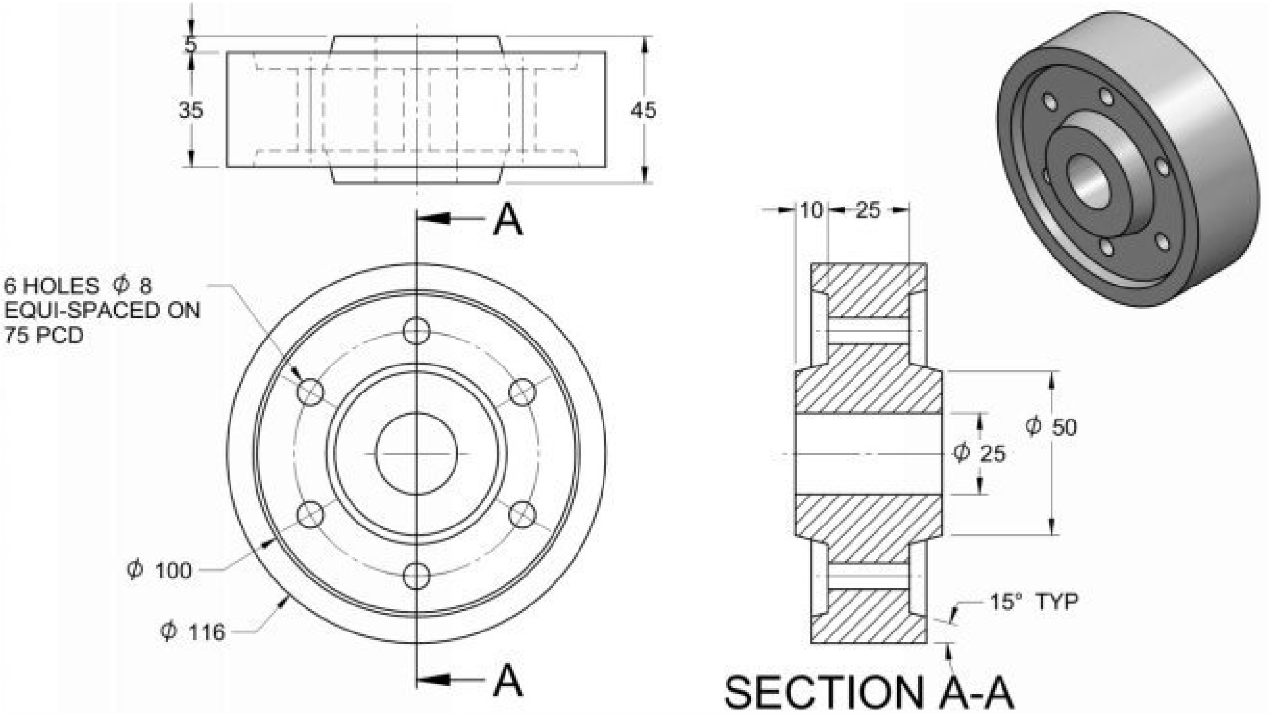
57. Save and close the part file.

Questions

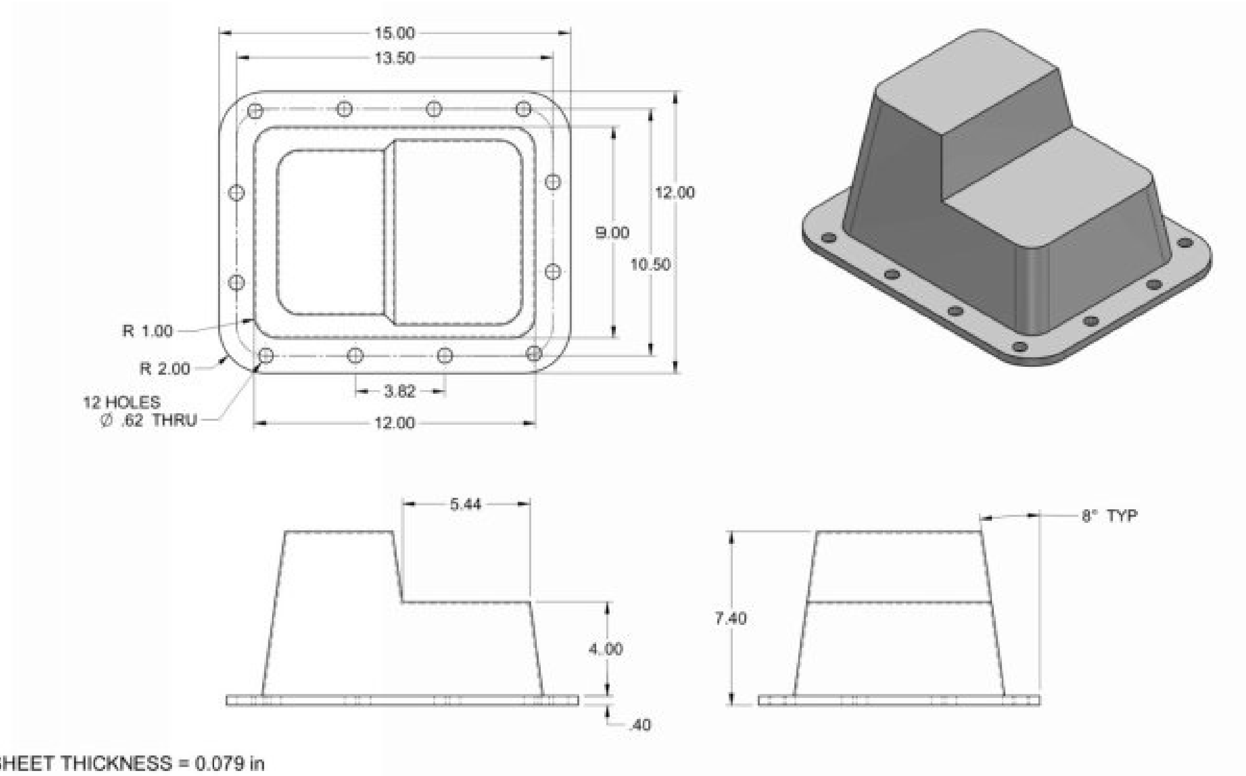
1. Describe the procedure to create a mirror feature.
2. List any two layouts to create patterns.
3. What is the difference between the **Mirror Feature** and **Mirror Geometry** command?
4. Describe the procedure to create a helical pattern.
5. List the methods to define spacing in a linear pattern.

Exercises

Exercise 1



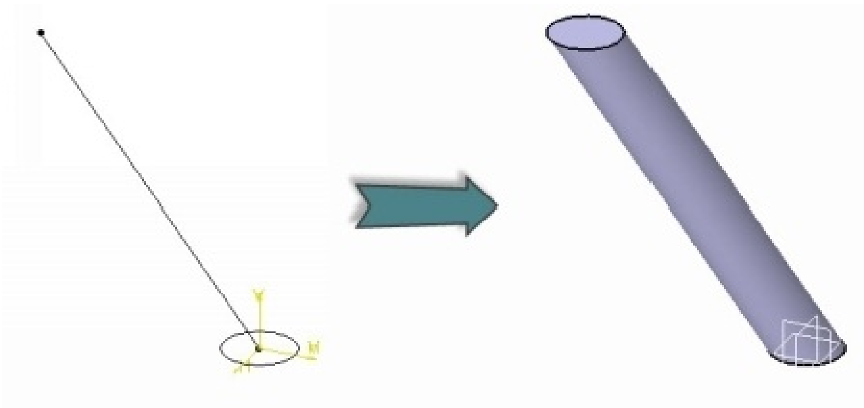
Exercise 2



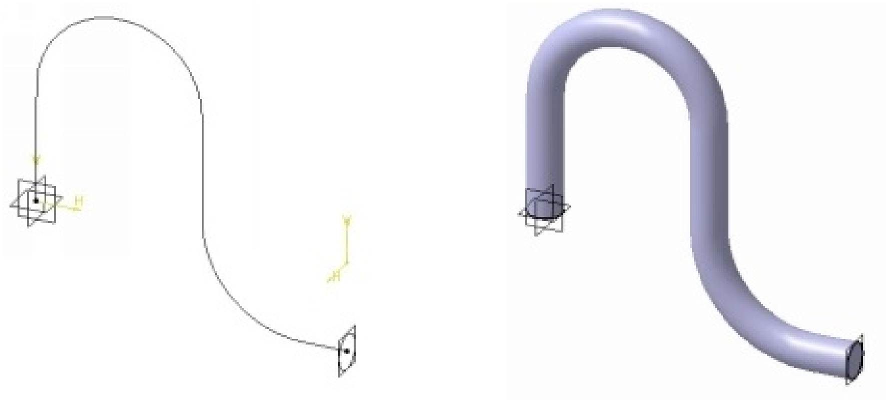
SHEET THICKNESS = 0.079 in

Chapter 6: Rib Features

The **Rib** command is one of the basic commands available in CATIA that allow you to generate a solid geometry. It can be used to create simple geometry as well as complex shapes. A rib is composed of two items: a cross-section and a path. The cross-section controls the shape of rib while the path controls its direction. For example, look at the angled cylinder shown in figure. This is created using a simple rib with the circle as the profile and an angled line as the path.



By the making the path a bit more complex, you can see that a rib allows you to create the shape you would not be able to create using commands such as Pad or Revolve.



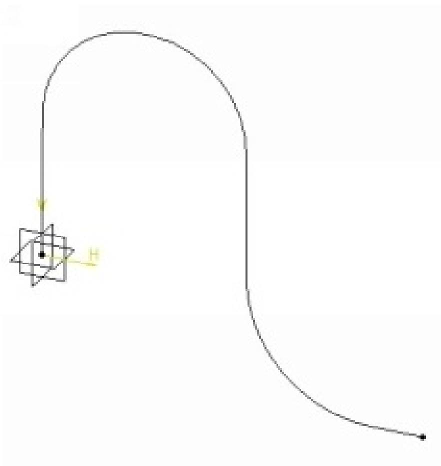
The topics covered in this chapter are:

- *Create Simple rib features*
- *Avoiding errors and intersections*
- *Various types of center curves that can be used to create rib features*
- *Merging end faces of the rib*
- *Create Slot features*

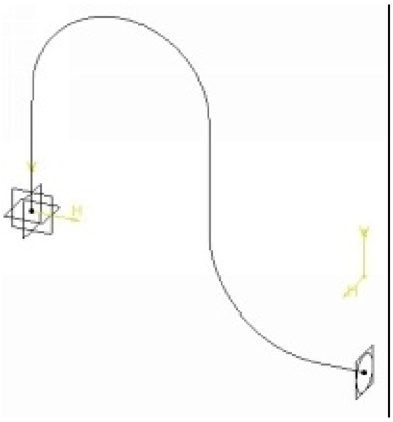
The Rib command

This type of rib requires two elements: a center curve and profile. The profile defines the shape of the rib along the center curve. A center curve is used to control the direction of the profile and it can be a sketch or an edge.

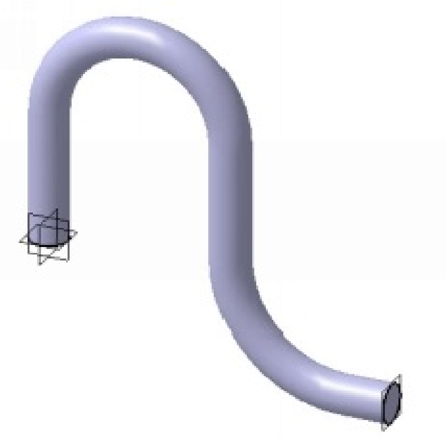
1. To create a rib, you must first create a center curve and a profile.
2. Create a center curve by drawing a sketch. It can be an open or closed sketch.



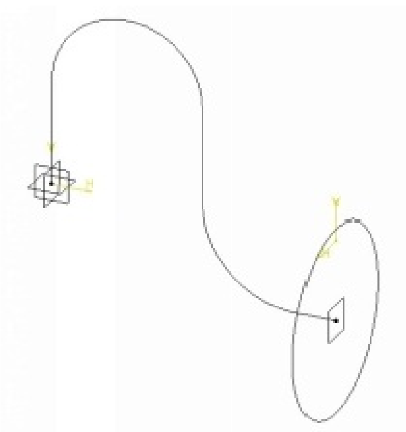
3. Activate the **Plane** command and create a plane normal to the path.
4. Sketch the profile on the plane normal to the path.

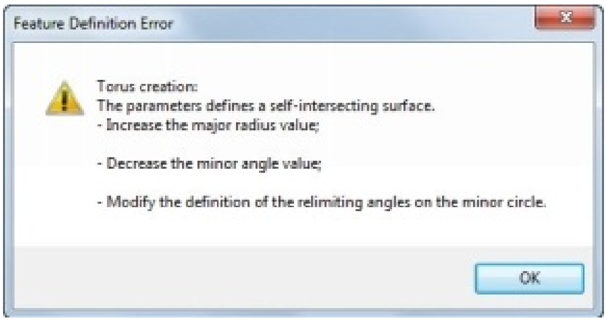


5. On the **Sketch-Based Features** toolbar, click the **Rib** button (or) click **Insert > Sketch-Based Features > Rib** on the Menu bar. As you activate the **Rib** command, a dialog appears showing different options to create the rib.
6. Select the profile and center curve.
7. Click **OK**.

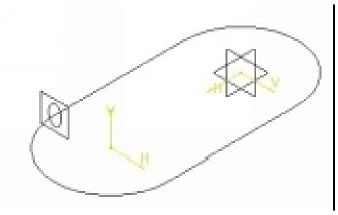


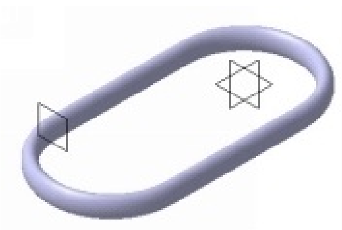
CATIA will not allow the sweep to result in a self-intersecting geometry. As the profile is swept along a path, it cannot comeback and cross itself. For example, if the profile of the sweep is larger than the curve, the resulting geometry will intersect and the sweep will fail.

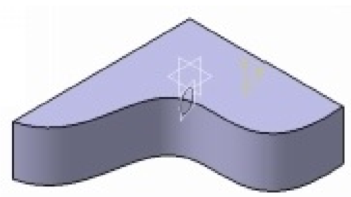


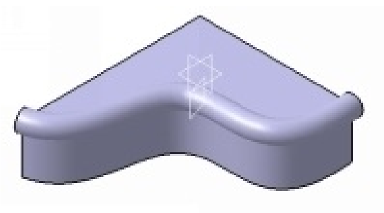


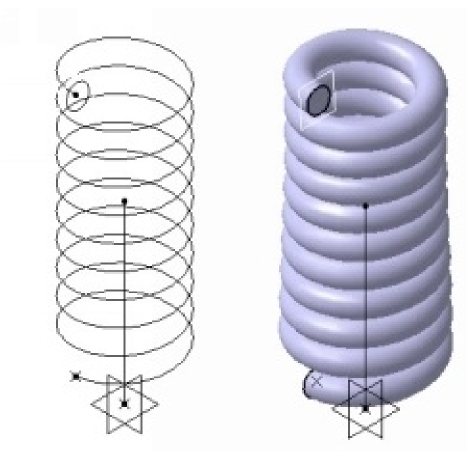
A rib profile must be created as a sketch. However, a center curve can be a sketch, curve, or edge. The following illustrations show various types of center curves and resultant rib features.





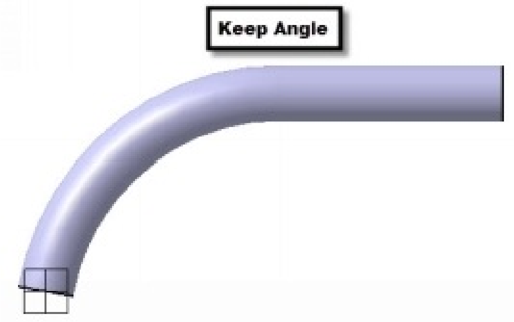


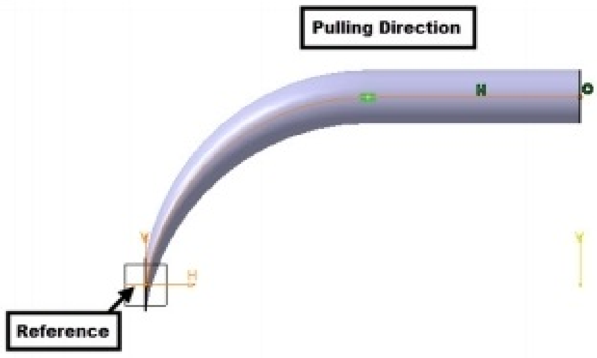




Profile Control

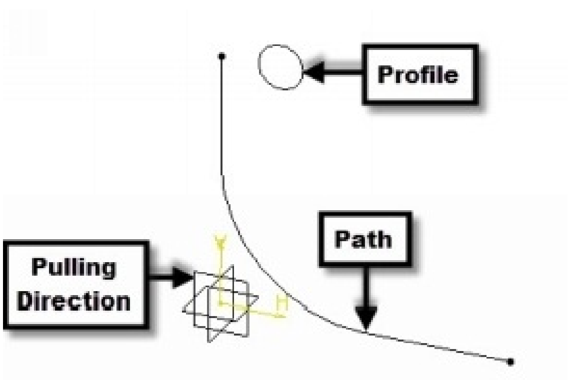
The profile control options define the orientation of the resulting geometry. The **Keep angle** option sweeps the profile in the direction normal to the center curve. The **Pulling Direction** option sweeps the profile in the along the direction that you define.

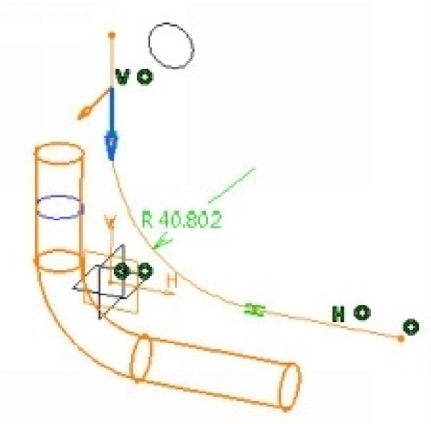




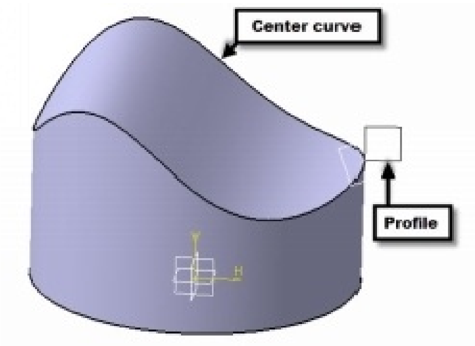
You can also use the **Pulling Direction** option to sweep the profile and path, which are not normal to each other.

1. Activate the **Rib** command and select **Profile Control > Pulling Direction**.
2. Select the plane parallel to the profile and path. This defines the pulling direction.
3. Check the **Move profile to path** option.
4. Select the profile and path.

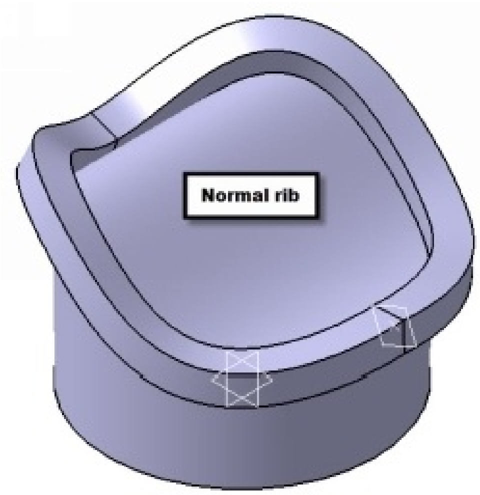


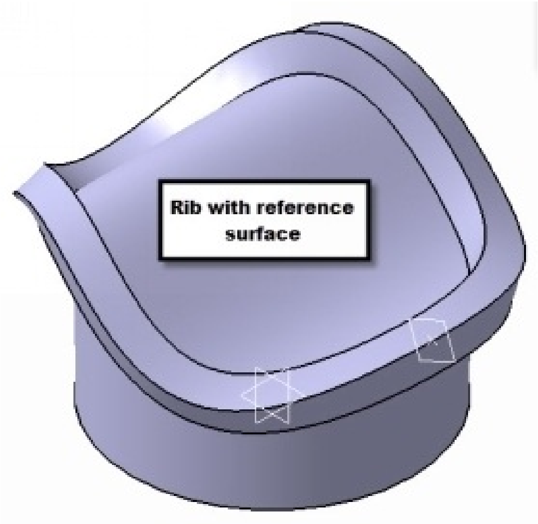


The **Reference Surface** option under the **Profile Control** section will be useful while sweeping a profile along a non-planar path. For example, define a path and cross-section similar to the one shown in figure. Select the profile and path

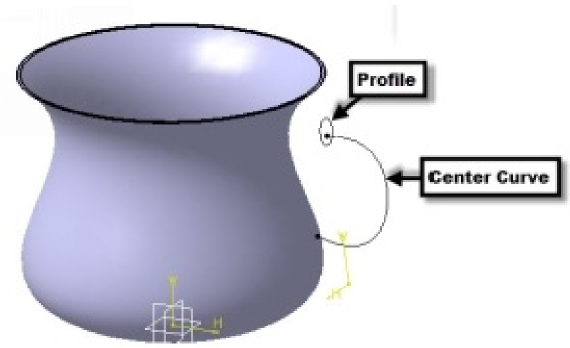


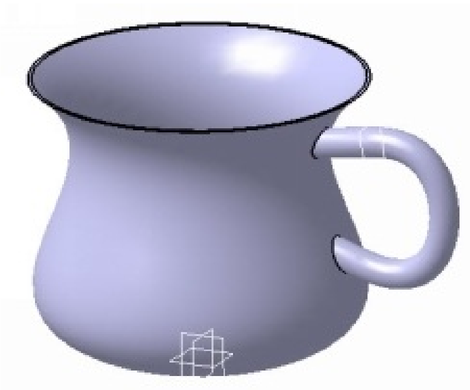
On the dialog, select **Profile Control > Reference Surface** and click on the top surface. The rib will be created by maintaining contact with the top surface.



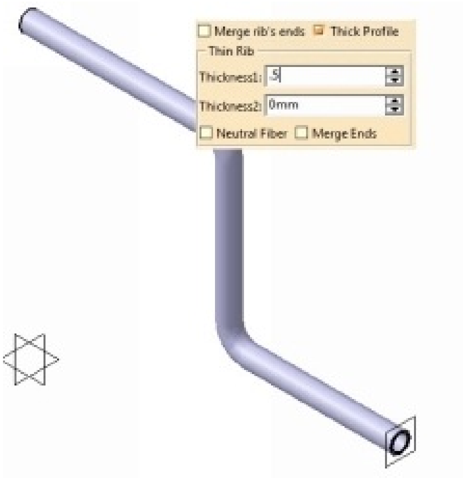


Use the **Merge end faces** option to merge the end faces of the rib feature with the adjacent model faces. You can use this option for the cases similar to the following one.





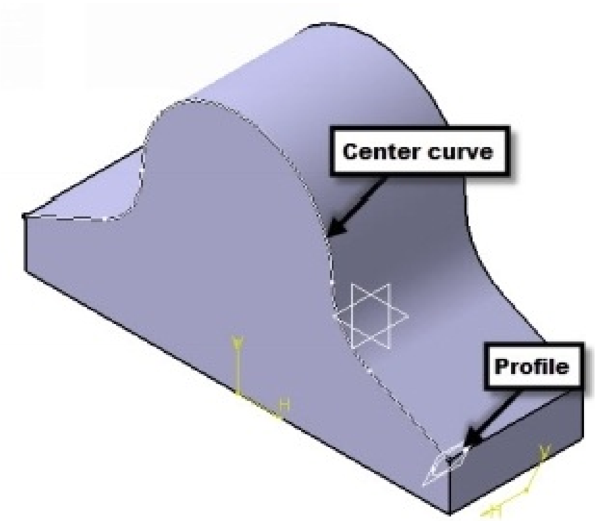
Use the **Thick Profile** option to create a shelled rib feature. This option can be used to create pipes.



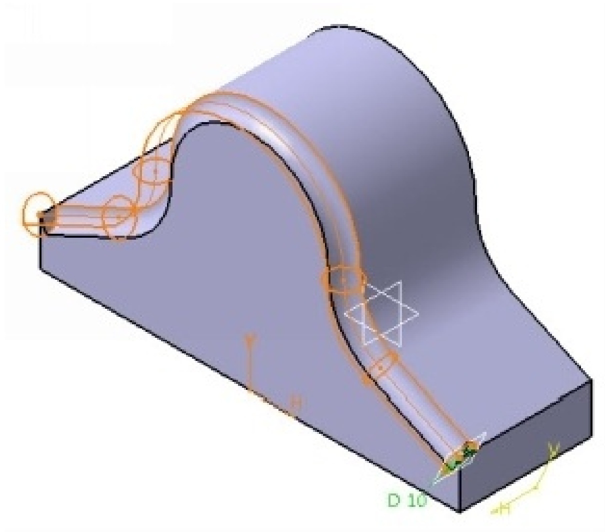
 The Slot command

In addition to adding rib features, CATIA allows you to remove geometry using the **Slot** command.

1. On the **Sketch-Based Features** toolbar, click the **Slot** button (or) click **Insert > Sketch-Based Features > Slot** on the Menu bar.
2. Select the profile and center curve from the model geometry.

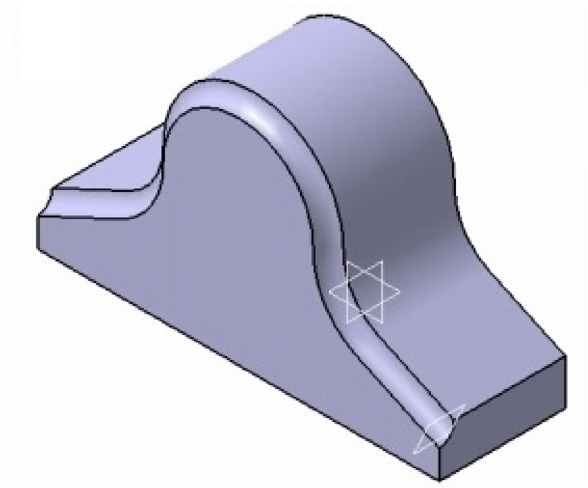


3. Click **Preview** on the dialog. You will notice that slot is not created throughout the



geometry.

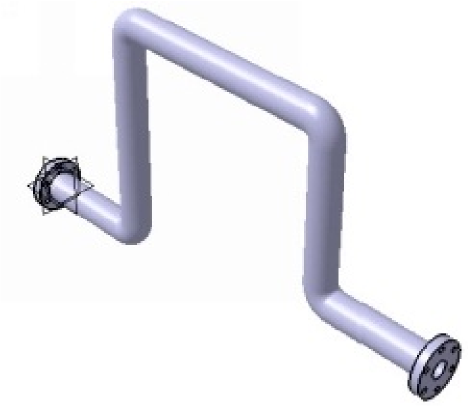
4. On the dialog, check the **Merge slot's ends** option. The resultant swept cutout will be throughout the geometry. Click **OK**.

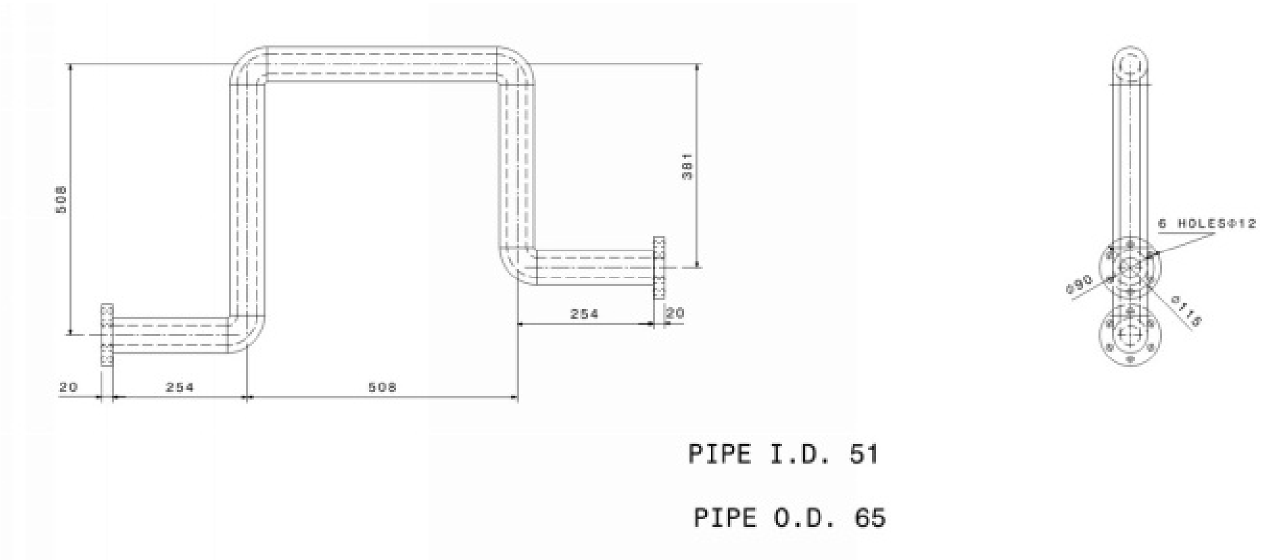


Examples

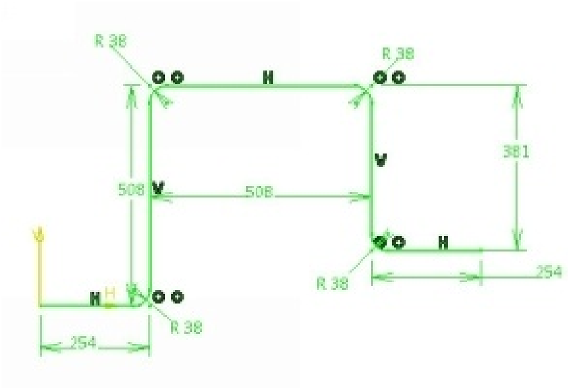
Example 1

In this example, you will create the part shown below.



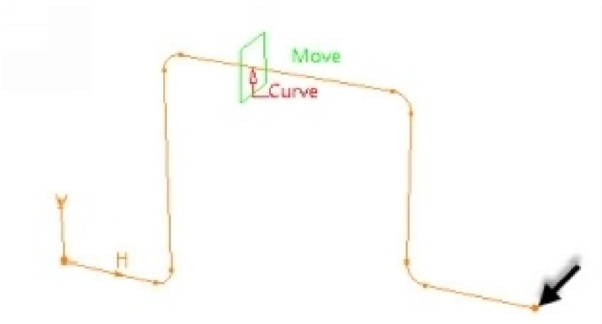


1. Start **CATIA V5-6R2015**.
2. Open a new part file.
3. Draw the sketch on the YZ plane, as shown in figure.
4. Click **Exit workbench** to complete the sketch.

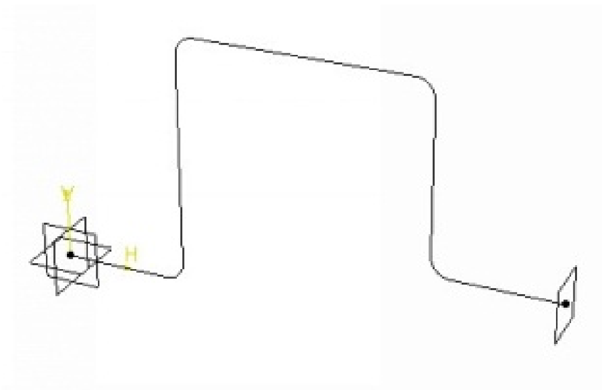


5. On the **Reference Elements** toolbar, click the **Plane**  icon.

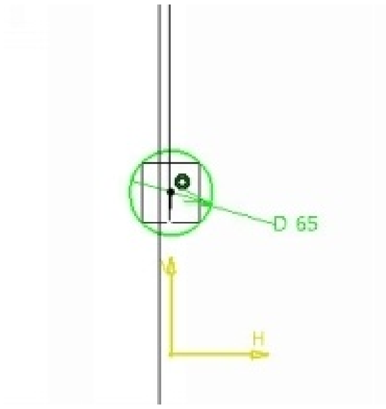
6. On the **Plane Definition** dialog, click **Plane type > Normal to curve**.
7. Click on the end-point of the sketch to define the plane location.



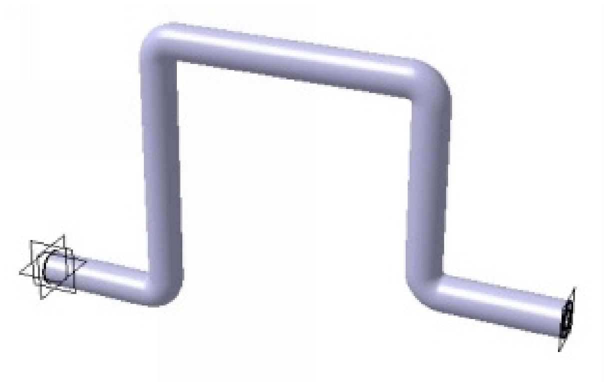
8. Click **OK** to create the plane.



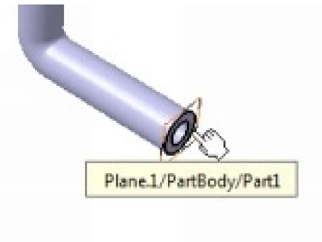
9. Start a sketch on the new plane.
10. Draw a circle and make its center coincident with endpoint of the previous sketch.



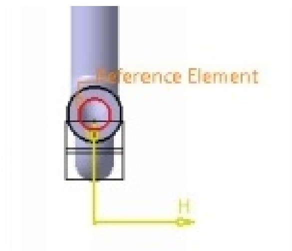
1. Click the **Exit workbench** icon.
2. On the **Sketch-Based Features** toolbar, click the **Rib** icon (or) click **Insert > Sketch-Based Features > Rib** on the Menu.
3. Select the first sketch to define the Center curve.
4. On the **Rib Definition** dialog, check the **Thick Profile** option and type-in 14 in the **Thickness 1** box.
5. Click **OK** to create the *Rib* feature.



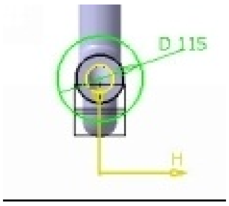
6. Activate the **Pad** command and click the **Sketch** icon
7. Click on the front-end face of the *Rib* feature.



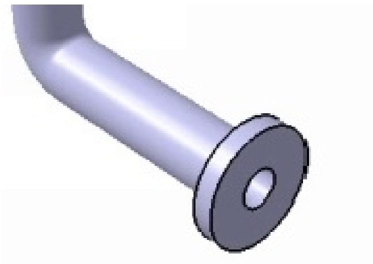
8. On the **3D Geometry** toolbar, click **Project 3D Elements** and click on the inner circular edge.



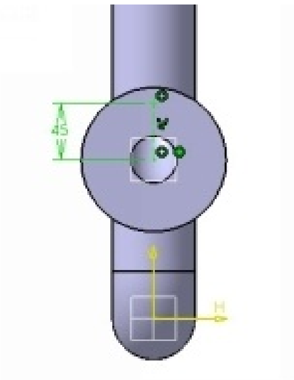
9. On the **Projection** dialog, click **OK** to project the curve onto the sketch plane.
0. Draw a circle of 115 diameter.

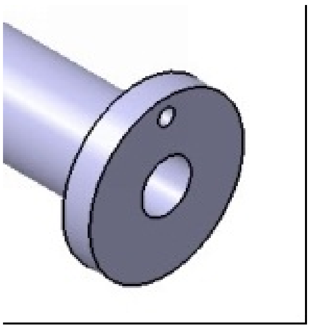


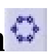
1. Click **Exit workbench** to complete the sketch.
2. Type-in **20** in the **Length** box. Click the Reverse Direction and **OK** to complete the *Pad* feature.



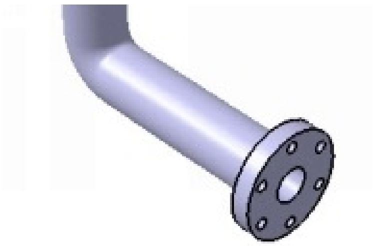
3. Create a hole of 12 diameter on the *Pad* feature.



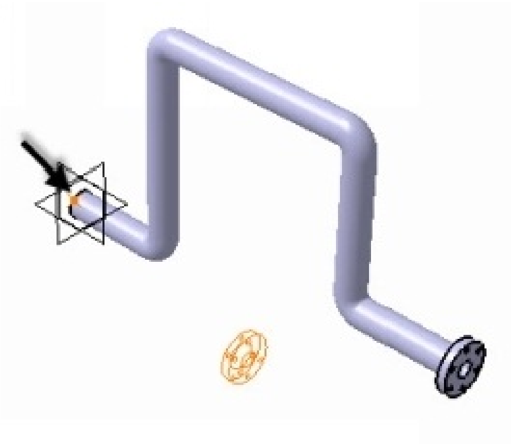


4. On the **Transformation Features** toolbar, click **Pattern** drop-down > **Circular Pattern**.
5. On the **Circular Pattern Definition** dialog, select **Parameters** > **Instance(s) & angular spacing**.
6. Type-in **6** and **60** in the **Instance(s)** and **Angular spacing** boxes, respectively.
7. Under the **Reference Direction** section, click in the **Reference element** box and select the outer cylindrical face of the *Pad* feature.
8. Under the **Object to Pattern** section, click in the **Object** box and select the hole.

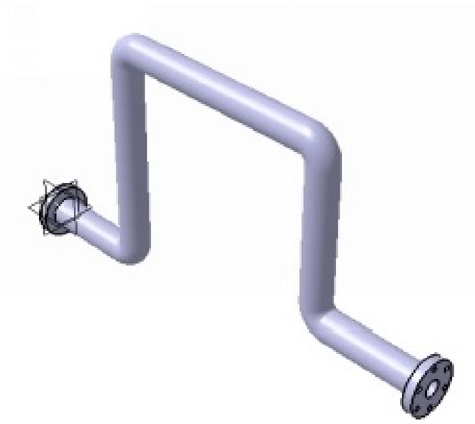
9. Click **OK** to pattern the hole.



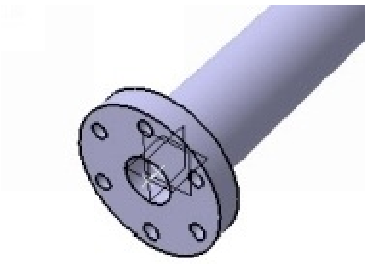
0. On the **Reference Elements** toolbar, click the **Point**  icon.
1. On the **Point Definition** dialog, type-in 0, -20 and 0 in X, Y, and Z boxes and then click **OK**.
2. On the **Transformation Features** toolbar, click **Pattern** drop-down > **User Pattern**  .
3. Click on the new reference point.



4. On the **User Pattern Definition** dialog, click in the **Object** box and select the *Pad* feature.
5. Click **OK** to pattern the *Pad* feature.



6. On the **Reference Elements** toolbar, click the **Point**  icon.
7. On the **Point Definition** dialog, type-in 0, -10 and 0 in X, Y, and Z boxes and then click **OK**.
8. Activate the **User Pattern** command and select the reference point.
9. Click in the **Object** box and select the Circular pattern from the Specification tree.
0. Click **OK** to pattern the circular pattern.



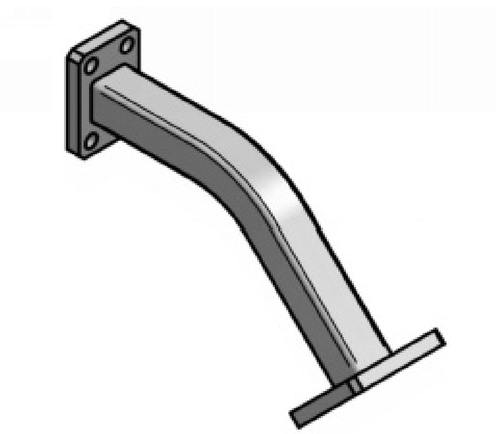
1. Change the model view orientation.
2. Save and close the part file.

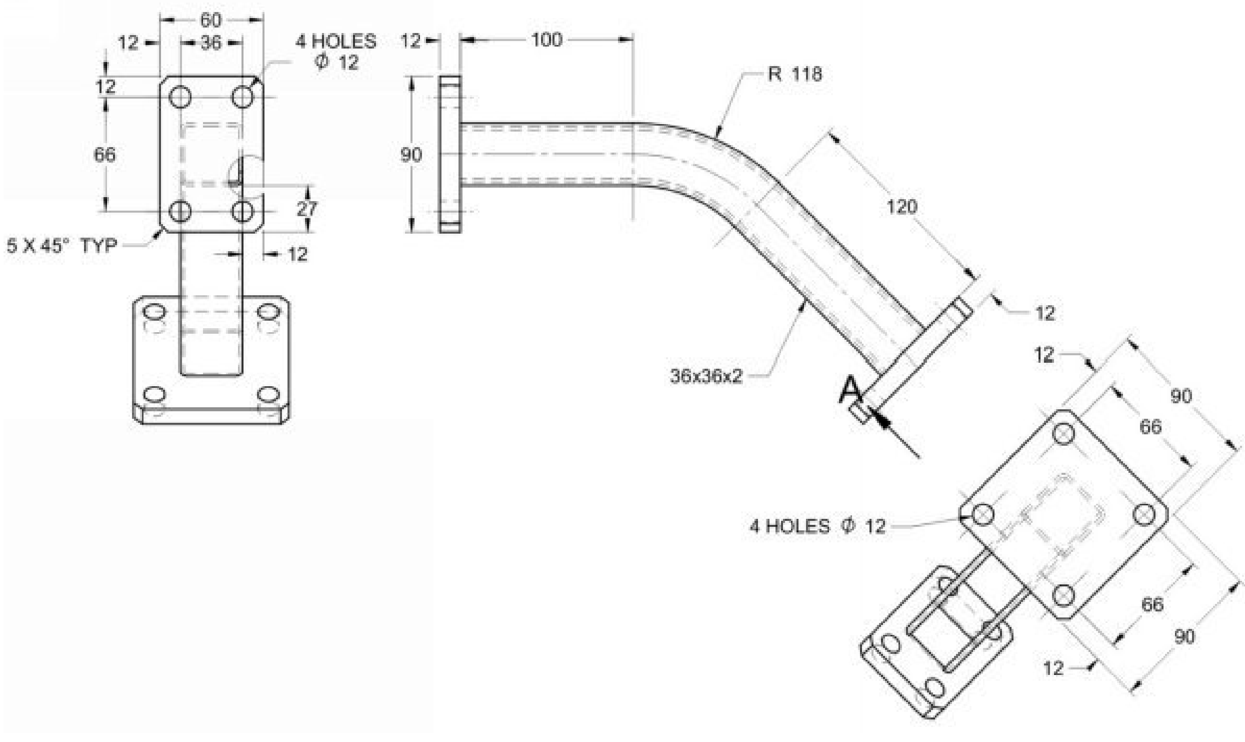
Questions

1. List the types of path that can be used to create *Rib* features.
2. What is the use of **Merge slot ends** option?
3. Why do we use the **Reference Surface** option in the **Rib** command?
4. What is the use of **Pulling Direction** option?

Exercises

Exercise1




60
12
36
4 HOLES
Φ 12
12
66
27
5 X 45° TYP
12
12
100
90
R 118
120
12
36x36x2
A
12
90
66
4 HOLES Φ 12
66
90
12

Chapter 7: Multi Section Solids

The **Multi Section Solid** command is one of the advanced commands available in CATIA that allows you to create simple as well as complex shapes. A basic multi section solid is created by defining two cross-sections and joining them together. For example, if you create a loft feature between a circle and a square, you can easily change the cross-sectional shape of the solid. This ability is what separates the multi section solid feature from the rib feature.

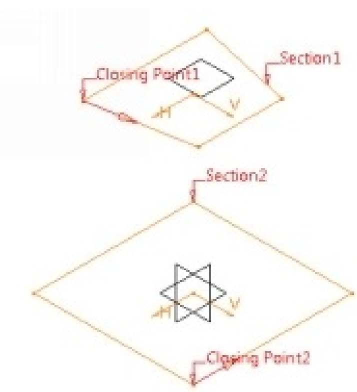
The topics covered in this chapter are:

- *Basic Multi-sections Solid*
- *Multi Section Solids*
- *Removed Multi Section Solids*

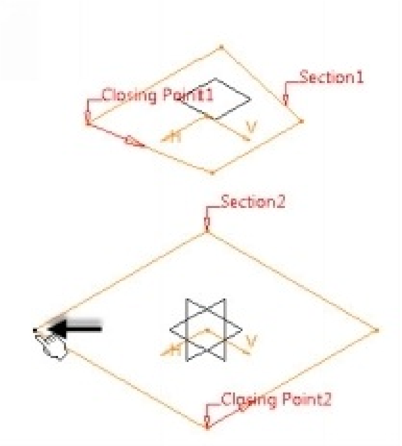
The Multi-sections Solid command

This command creates a solid feature between different cross-sections.

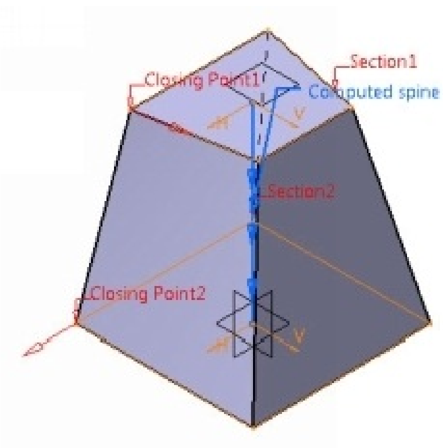
1. To create this type of feature, first create two or more sections on different planes. The planes can be parallel or perpendicular to each other.
2. On the **Sketch-Based Features** toolbar, click the **Multi-sections Solid** button (or) **Insert > Sketch-Based Features > Multi-sections Solid** on the Menu bar.
3. Select two or more sections to define the multi section solid.



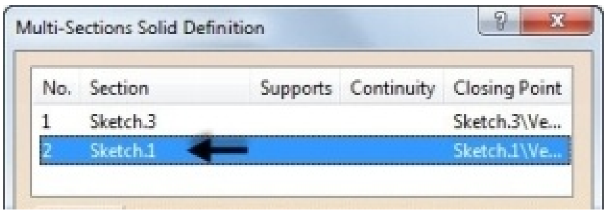
You have to ensure that the closing points of the cross-sections should be on same corners. For example, if the closing point of the first section is on the left corner, then the closing points on other cross-sections should also be on the left corner. If they are not on same corners, then click the right mouse button on anyone of the cross-sections on the dialog and select **Replace Closing Point**. Click on the corner point matching with the other section.

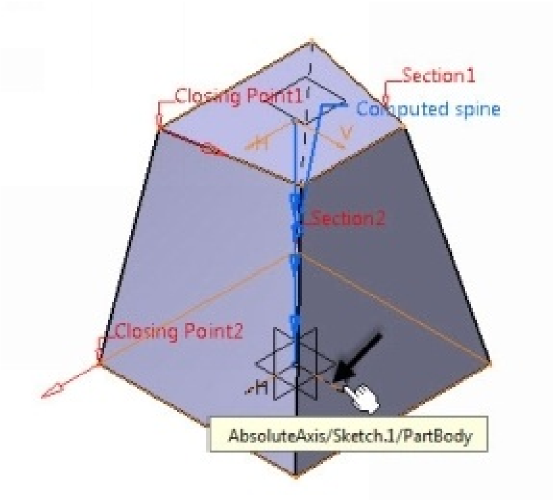


4. Click the **Preview** button. The model preview updates immediately, as shown below.

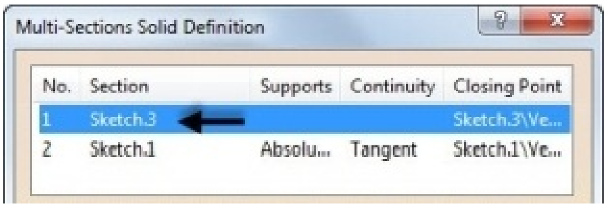


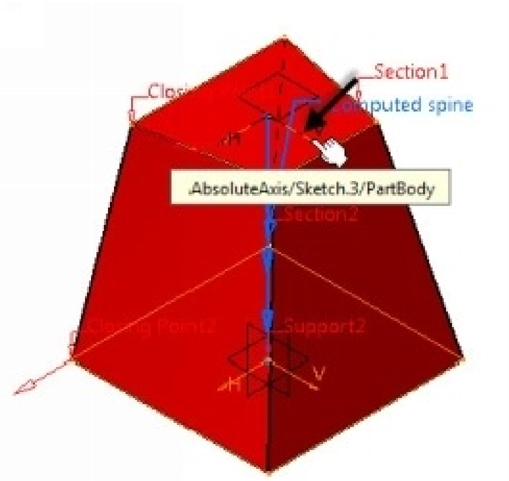
5. Now, you have to define supports for the cross-sections. Click on anyone of the sections on the dialog and select the absolute coordinate system of the section.



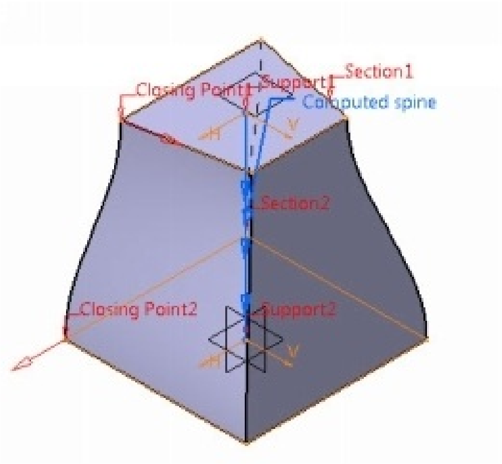


5. Likewise, and click on the other sections and click on its absolute coordinate system.

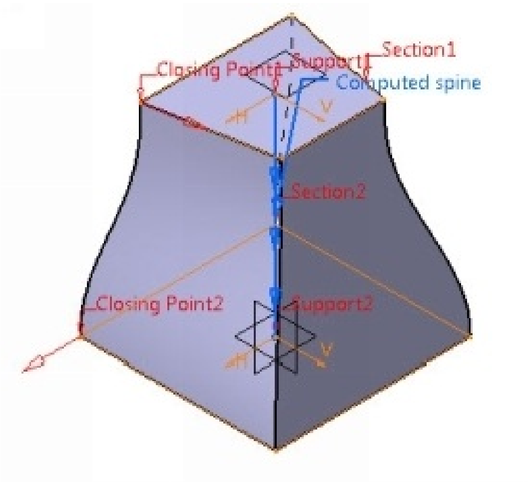




The shape of a simple multi section solid is controlled by the cross-sections and the plane location. However, the behaviour of the side faces can be controlled by the **Continuity** between the cross-sections. If you would like to change the appearance of the side faces, you can change the **Continuity** of the cross-sections of the multi section solid. For instance, click on the first cross-section in the dialog, and then select **Continuity > Tangent**. Click **Preview** to view the changes. You can notice that the beginning of the solid starts in a direction normal to the cross-section.



If you select **Continuity > Curvature**, the side faces will maintain curvature continuity with the first cross-section.



Likewise, select the second cross-section from the dialog and change the continuity type.

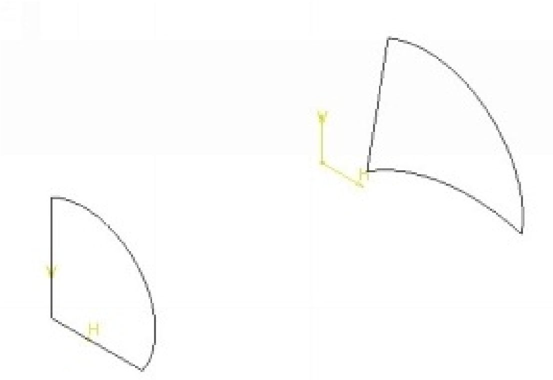
7. Click **OK** to complete the feature.

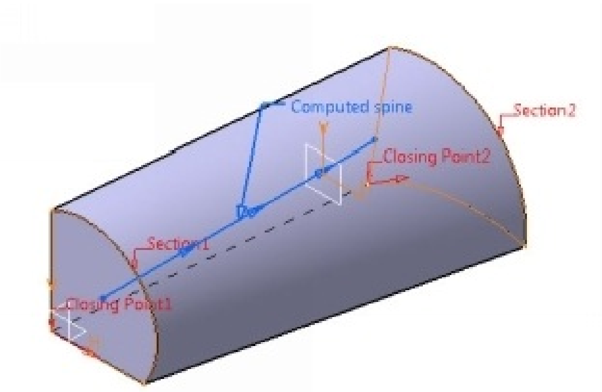
Types of the Cross-sections

In addition to 2D sketches, you can also use different element types to define cross-sections by using different element types. For instance, you can use existing model faces, surfaces, and curves

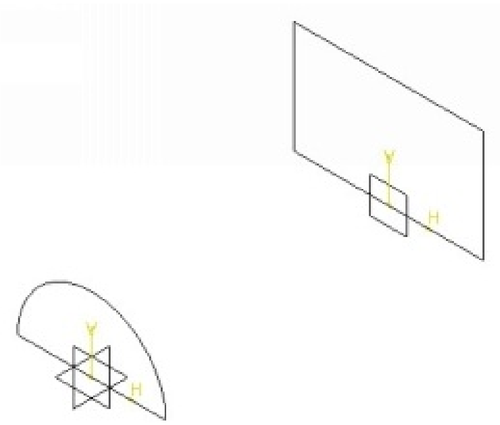
Couplings

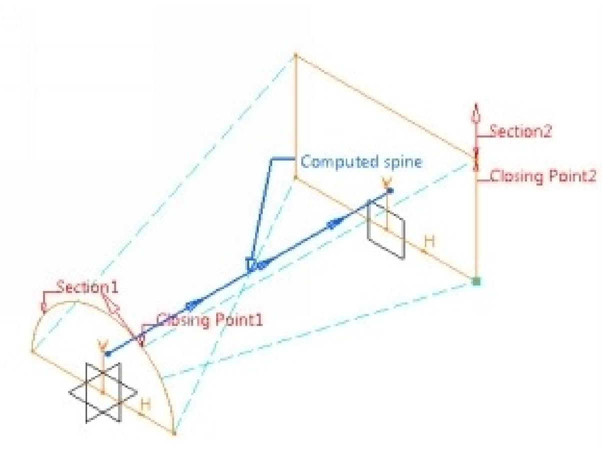
Sections used for creating multi section solids should have a matching number of segments. For example, a three-sided section will loft nicely to another three-sided section despite the differences in the shape of the individual segments. The **Multi Section Solid** command does a good job of generating smooth faces to join them.





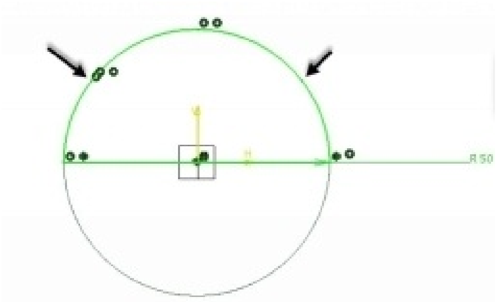
On the other hand, a four-sided section and two-sided section will result in an error.

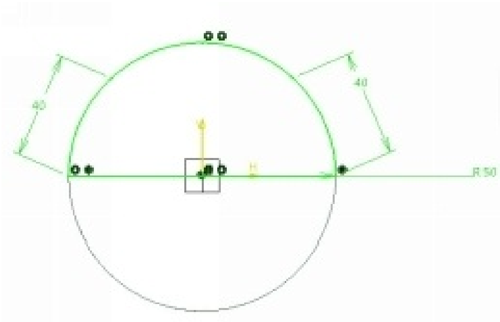




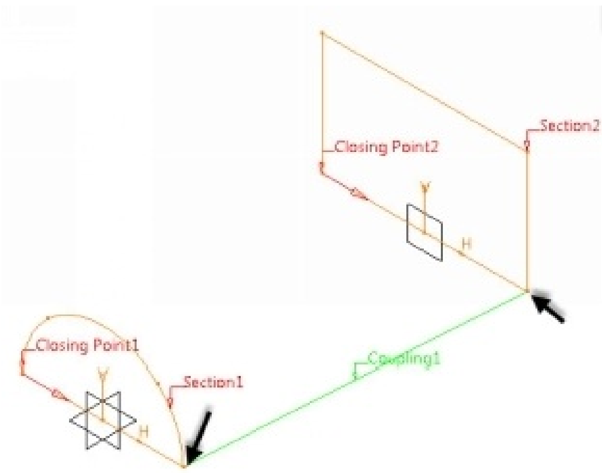
To get the desired result, you have to break one of the sections so that they have equal number of segments.

1. Click **Cancel** on the **Multi-sections Solid Definition** dialog and double-click on the arc to activate its sketch.
2. Activate the **Break** command (click **Insert > Operations > Relimitations > Break** on the Menu bar) and break the arc into three segments. You can also use dimensions to define the exact location of the split points.

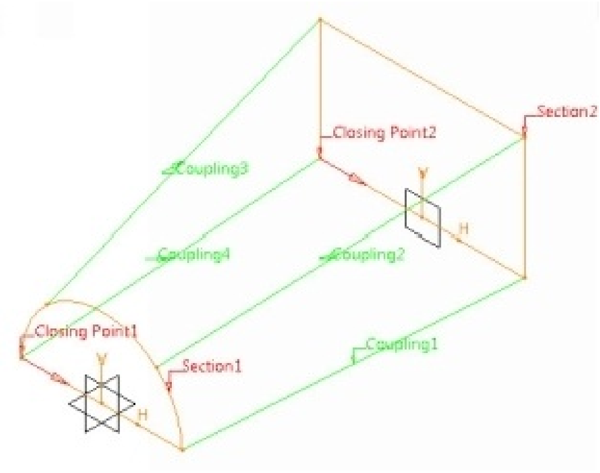




3. Now, exit the **Sketcher** workbench and activate the **Multi-sections Solid** command.
4. Select the sections one by one.
5. On the **Multi-sections Solid Definition** dialog, click the **Coupling** tab.
6. Select the vertices of the section 1 and section 2, as shown below. This creates a coupling.



7. Likewise, create other couplings using the vertices.

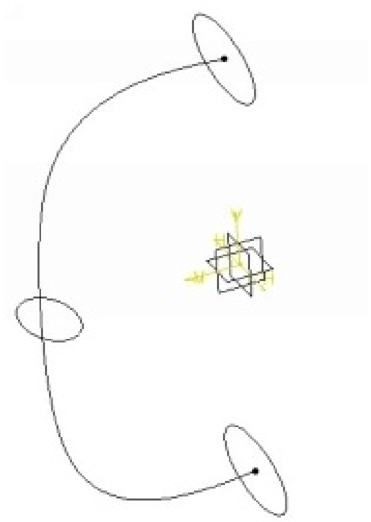


3. Click **OK** to complete the feature.

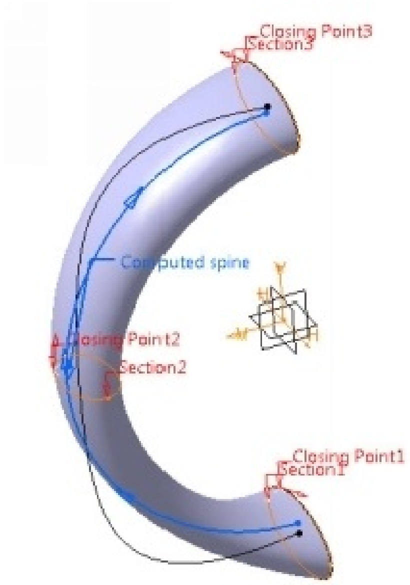
Spines

When you create a multi section solid, a spine is defined between the sections, automatically. The spine controls the way the multi section solid is transformed between the sections. You can also define the spine using a curve or sketch element.

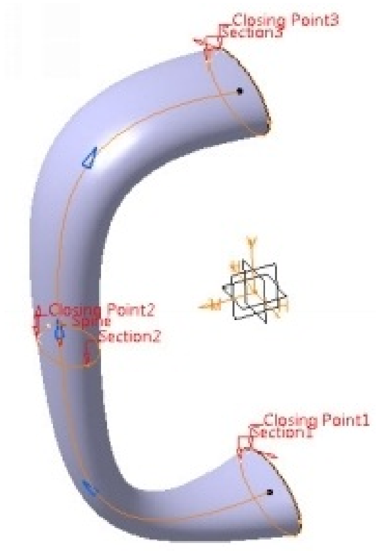
1. Activate the **Multi-sections Solid** command and select the sections.



2. Ensure that the arrows on the section point in the right direction.
3. Click **Preview** to view the resulting solid.

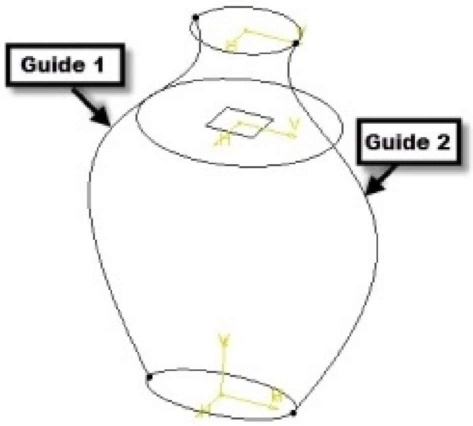


4. Click on the **Spine** tab and select the curve passing through the section. This defines the spine.
5. Click **OK** to create the solid.

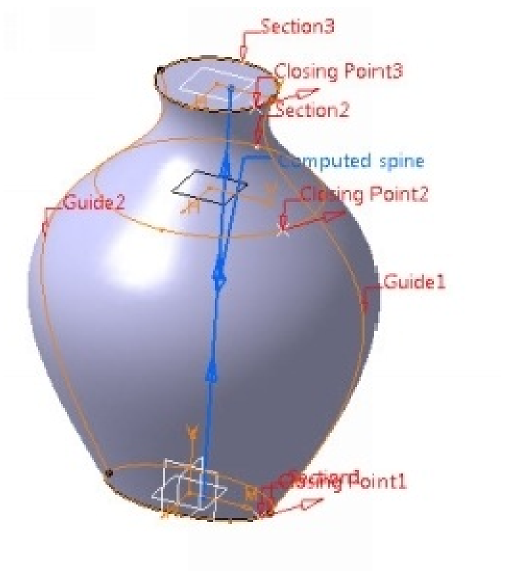


Guides

Similar to **Continuity** options, guides allow you to control the behaviour of a multi sections solid between cross-sections. You can create guides by using 2D sketches. You can also use the **3D Spline** command to create guided curves. Ensure that the guides touch the cross-sections.

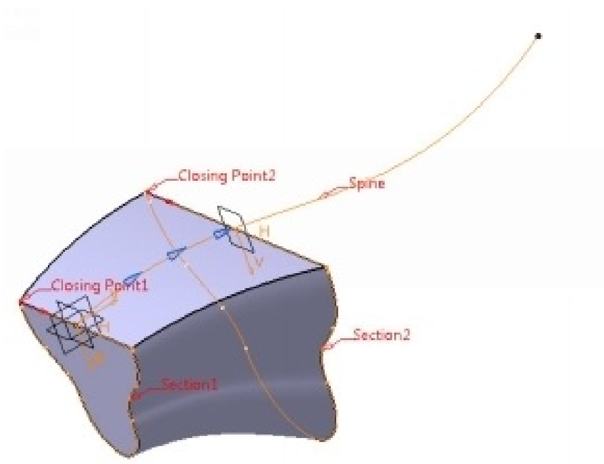


1. Now, activate the **Multi Section Solid** command and select the cross-sections.
2. To select guide curves, click the **Guides** tab on the dialog.
3. Click in the **Guides** list and select the first and second guides.
4. Click the **Preview** button. You will see that the preview updates.



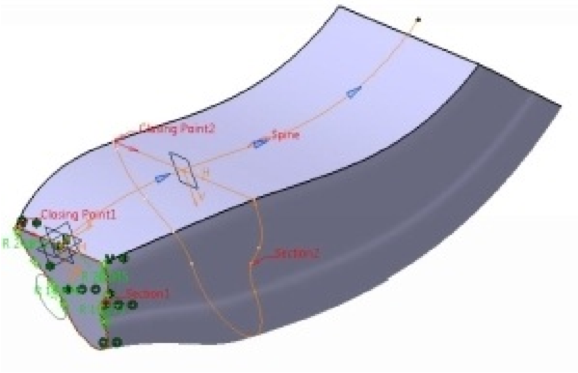
Relimitation

The **Relimitation** options limits the transformation of the multi sections solids to the start and end sections, as shown below.



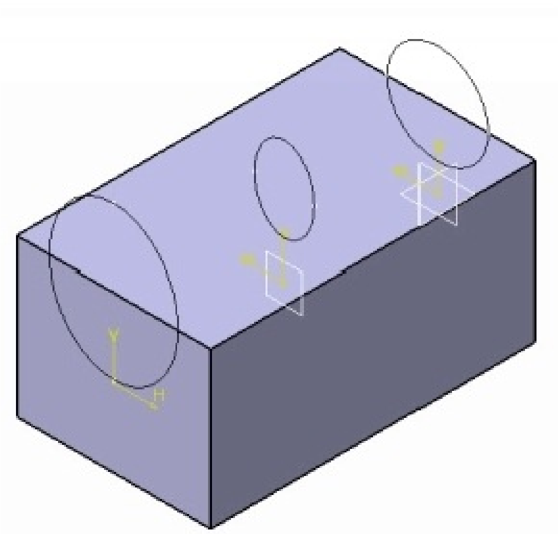
If you want to create the multi section solid up to the complete length of the spine or guides, then click the **Relimitation** tab on the dialog. Uncheck the **Relimited on start**

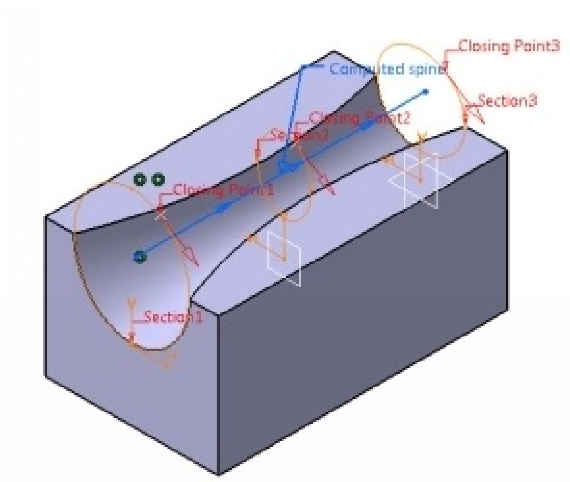
section and **Relimited on end section** options. Click **Preview** to view the relimited solid.



Removed Multi-sections Solid

Like other standard features such as pad, revolve and rib, the multi-sections solid feature can be used to add material. However, it can also be used to remove material. You can do this by using the **Removed Multi-sections Solid** command. Activate this command (click the **Removed Multi-sections Solid** button on the **Sketch-Based Features** toolbar) and select the cross-sections. Ensure that the arrows on the cross-sections point in the same direction. Click **Preview** and **OK** to create the feature.

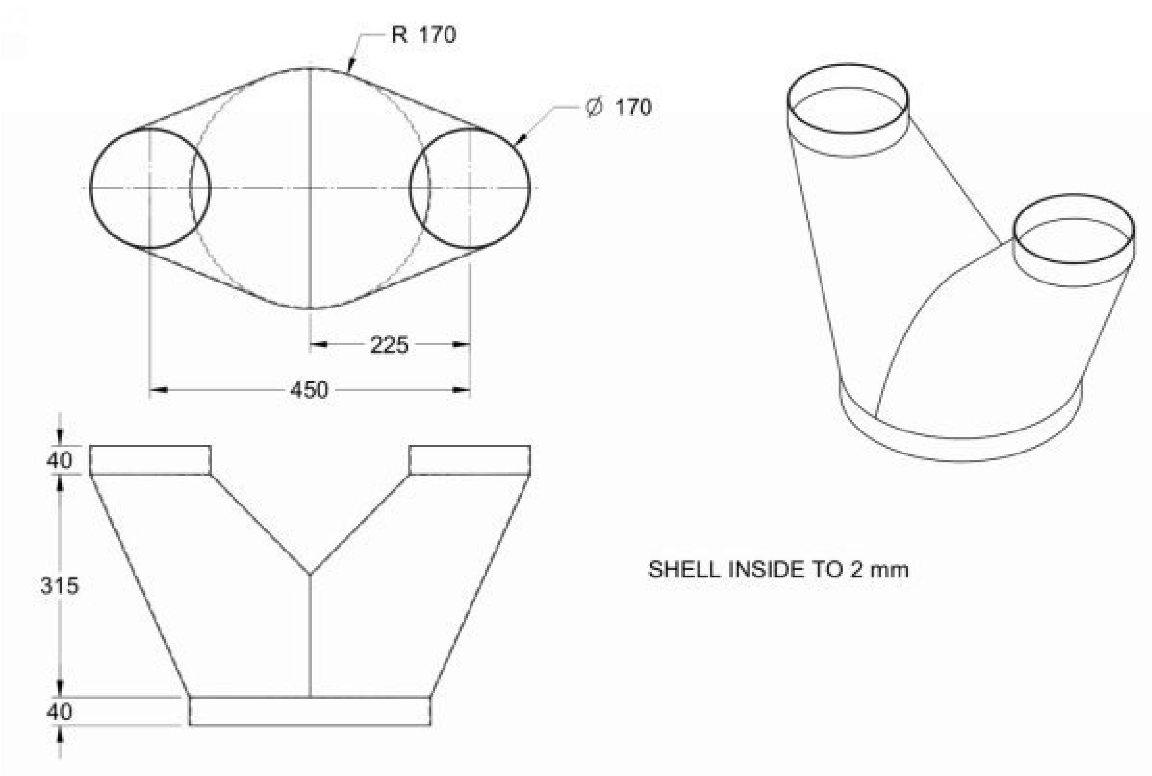




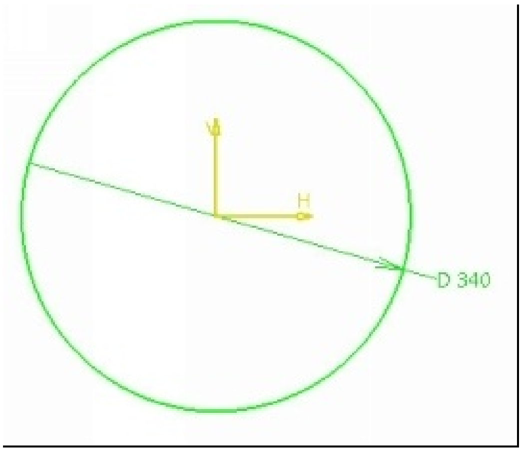
Examples

Example 1

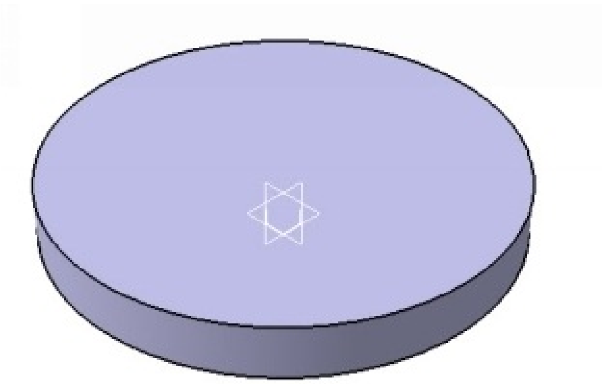
In this example, you will create the part shown below.



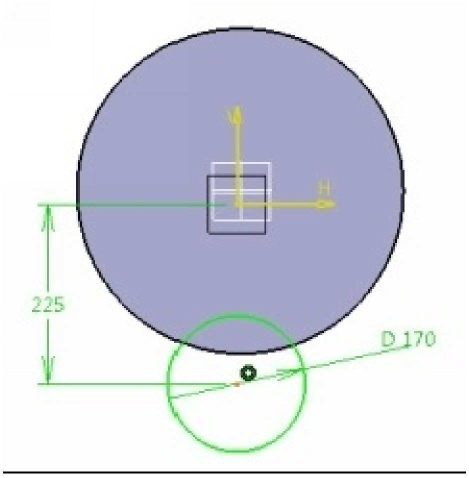
1. Start **CATIA V5-6R2015**.
2. Open a new part file.
3. Start a new sketch on the xy plane and draw a circle of 340 mm diameter.



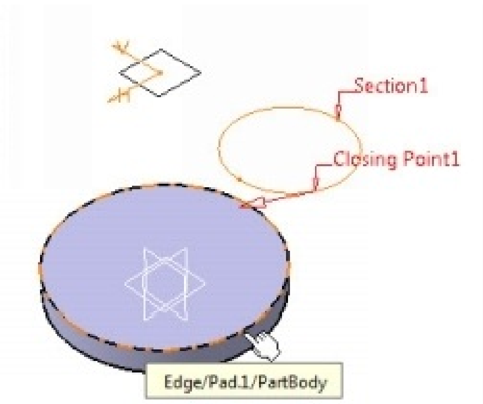
4. Exit the sketch.
5. Create the *Pad* feature with 40 mm thickness.



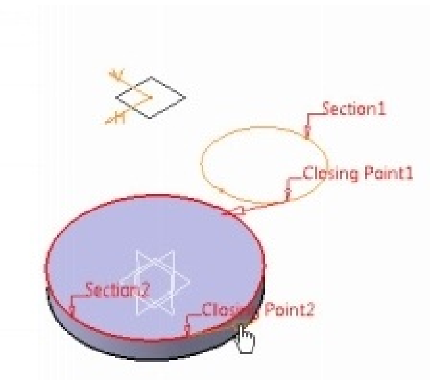
6. On the **Reference Elements** toolbar, click the **Plane** icon.
7. Click on the top face of the geometry and type-in 315 in the **Offset** box.
8. Click **OK** to create an offset plane.
9. Start a sketch on the offset plane.
0. Draw a circle of 170 mm diameter and add dimensions to it. Exit the sketch.



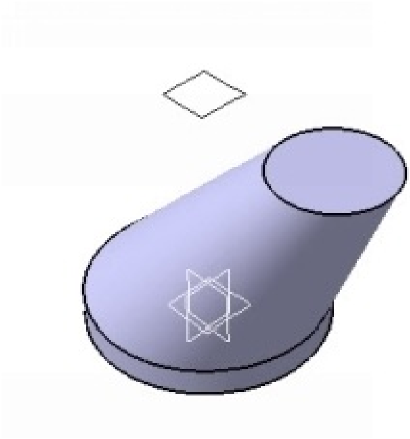
1. On the **Sketch-Based Features** toolbar, click the **Multi-sections Solid** icon. The first section will be selected, automatically.
2. Click on the circular edge of the base to define the second section.



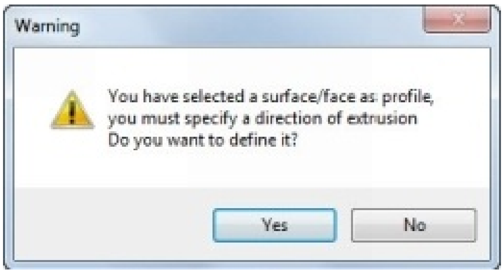
3. Ensure that the arrows point in same direction. If they point in opposite direction, then click on anyone of the arrow reverse its direction.



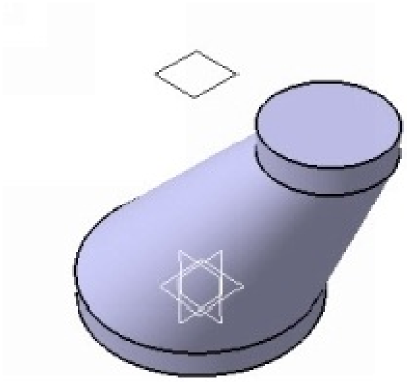
14. Click **OK** to create the *Multi-section solid* feature.



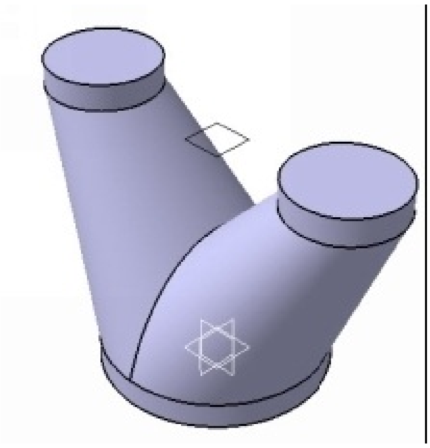
5. Activate the **Pad** command and click on the top face of the *Multi-section solid* feature.
6. Click **Yes** on the **Warning** message.



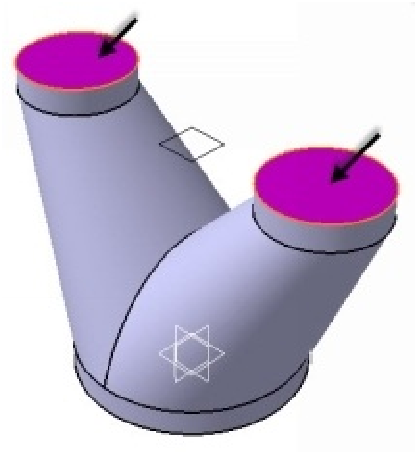
7. Select the xy plane on the Specification tree.
8. Type-in 40 in the **Length** box, and then click **OK** to create the *Pad* feature.

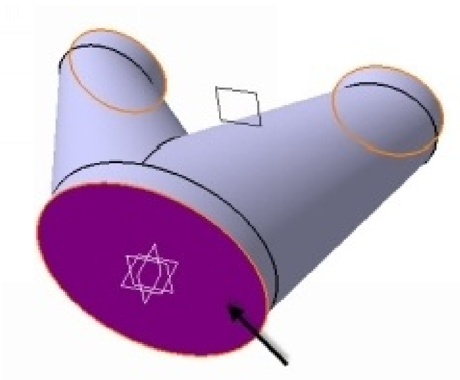


9. On the **Transformation Features** toolbar, click the **Mirror**  icon, and then select the zx plane to define the mirroring element.
0. Click **OK** to mirror the entire solid body.

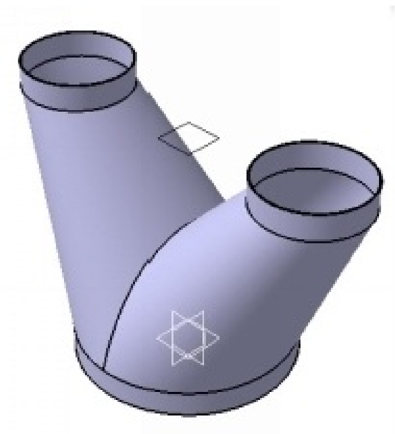


1. On the **Dress-up Features** toolbar, click the **Shell** icon.
2. Click on the flat faces of the model geometry.





3. On the **Shell Definition** dialog, type-in 2 in the **Default inside thickness** box.
4. Click **OK**. The part geometry is shelled.



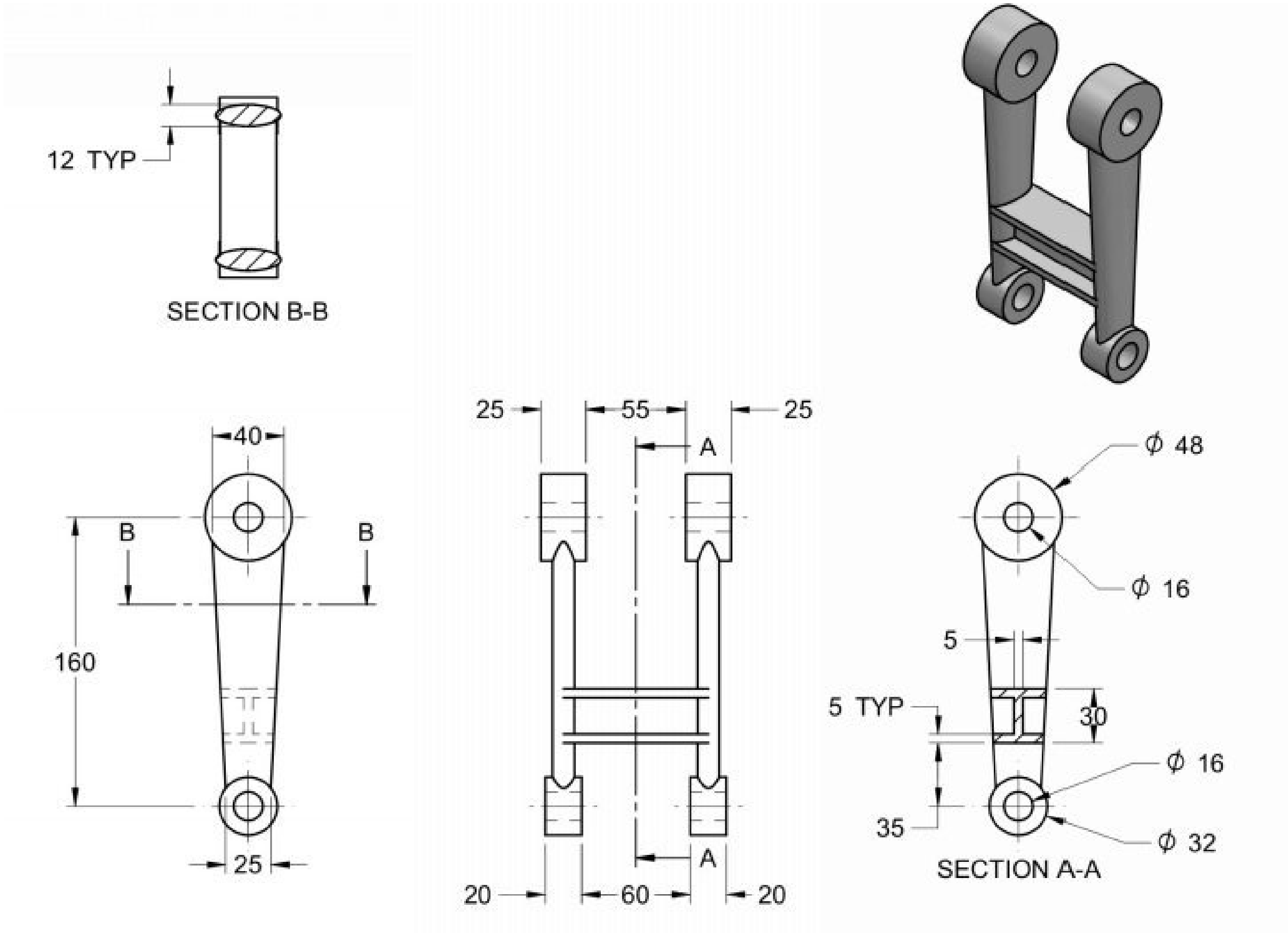
5. Save and close the part file.

Questions

1. Describe the procedure to create a *Multi-sections solid* feature.
2. List the **Continuity** options.
3. List the type of elements that can be selected to create a *Multi-sections solid* feature.

Exercises

Exercise 1



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SECTION A-A

Chapter 8: Additional Features and Multibody Parts

CATIA V5 offers you some additional commands and features which will help you to create complex models. These commands are explained in this chapter.

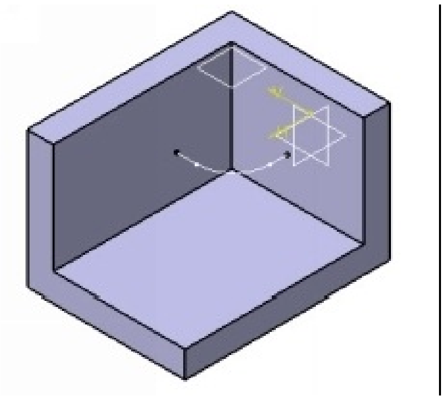
The topics covered in this chapter are:

- *Stiffeners*
- *Solid Combine*
- *Multi-body parts*
- *Boolean Operations*

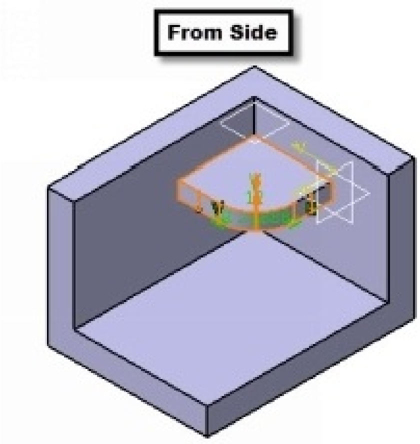
Stiffener

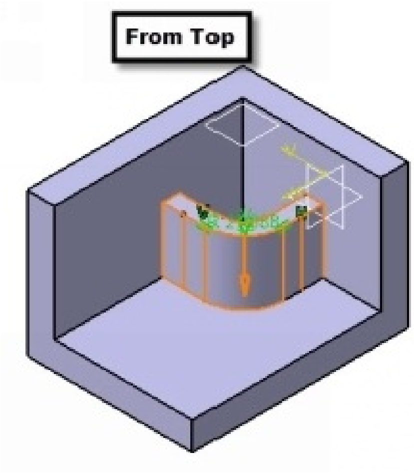
This command creates stiffening features to add structural stability, strength and support to your designs. Just like any other sketch-based feature, a stiffener requires a two dimensional sketch.

1. Create a sketch, as shown in figure.



2. On the **Sketch-Based Features** toolbar, click **Advanced** drop-down > **Stiffener** (or) click **Insert > Sketch-Based Features > Stiffener** on the Menu bar.
3. Select the sketch; the preview of the geometry appears. You can add material to either side of the sketch line or evenly to both sides.
4. Check the **Neutral Fiber** option to add material to both sides of the sketch line.
5. Type-in the thickness value of the stiffener feature in the **Thickness1** box.
6. You can click the **Reverse direction** button to change the side of the material (optional).
7. You can define the direction of the rib feature by using the **From Side** or **From Top** option.



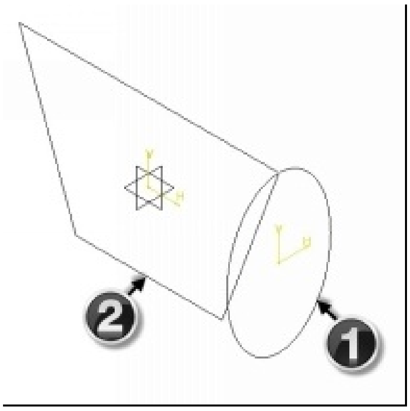


3. Click **OK** to complete the feature.

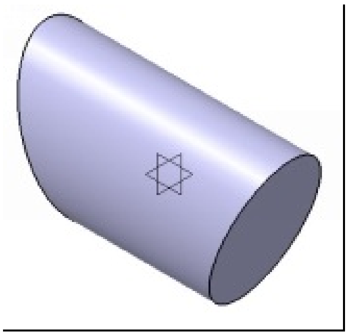
Solid Combine

This command creates a solid body by using two sketches, which are perpendicular to each other.

1. Create two sketches, as shown in figure.



2. On the **Sketch-Based Features** toolbar, click **Advanced** drop-down > **Solid Combine** (or) click **Insert > Sketch-Based Features > Solid Combine** on the Menu bar.
3. Select the two sketches.
4. Click **OK**.



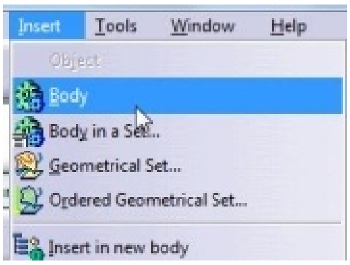
Multi-body Parts

CATIA V5 allows the use of multiple bodies when designing parts. This opens the door to several design techniques that would otherwise not be possible. In this section, you will learn some of these techniques.

Creating Multi-bodies

The number of bodies in a part can change throughout the design process. CATIA V5 makes it easy to create separate single bodies and multiple bodies, and combine multiple bodies into single bodies.

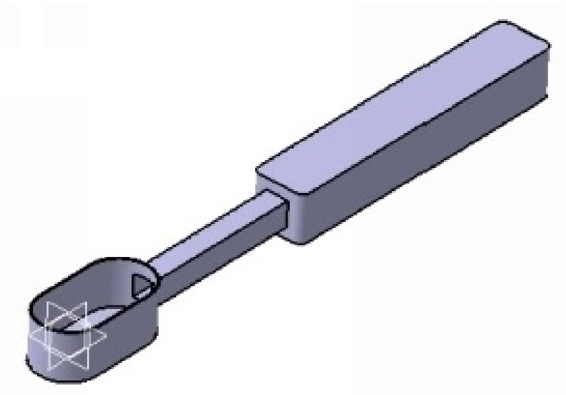
1. In order to create multiple bodies in a part, first create a solid body, and then click **Insert > Body** on the Menu bar. A new body is added to the Specification Tree and the previous body is hidden.

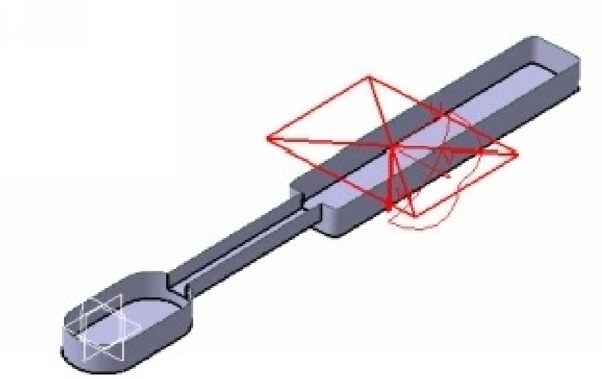


2. Now, create another solid body using anyone of the modeling tools.

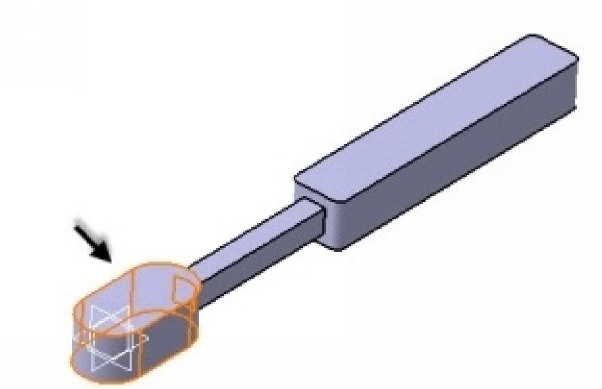
Insert in new body

The **Insert in new body** command can be used to separate a feature of the body, and then insert it in a new body. This command can be used to perform local operations. For example, if you apply the shell feature to the front portion of the model shown in figure, the whole model will be shelled. To solve this problem, you must split the solid body into multiple bodies.





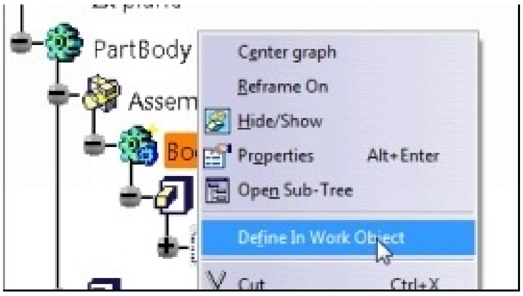
1. To perform shell operation to the front portion, you must create it as a separate feature.
2. Expand the Specification Tree and select the feature of the front portion.



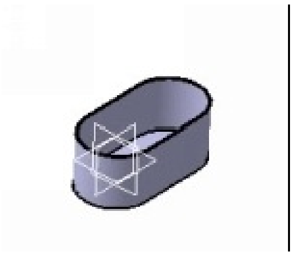
3. On the Menu bar, click **Insert > Insert in new body**.

You will notice that a new body is created and the selected feature is inserted into that body. In addition, the Assemble operation is performed between the new body and main body.

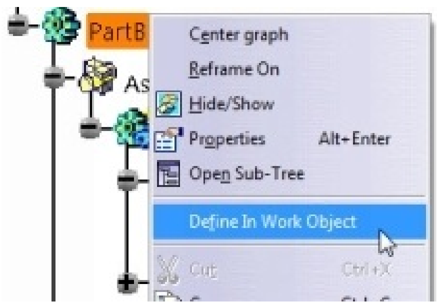
4. Now, click the right mouse button on the new body and select **Define In Work Object**. This activates the new body.

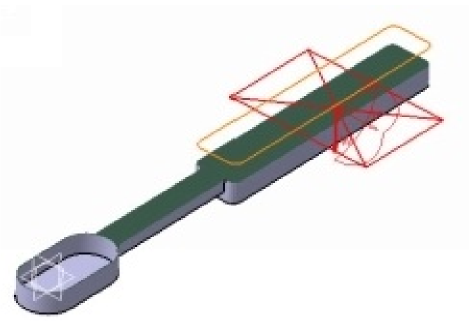


5. Now, activate the **Shell** command and perform the shell operation.



6. Click the right mouse button on the main body and select **Define In Work Object**.



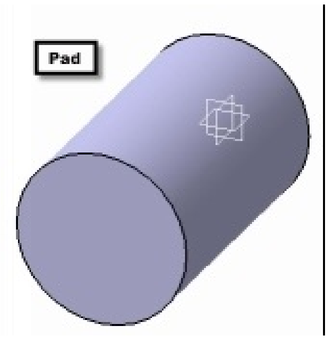


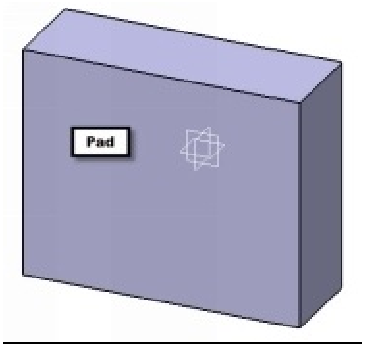
 Assemble

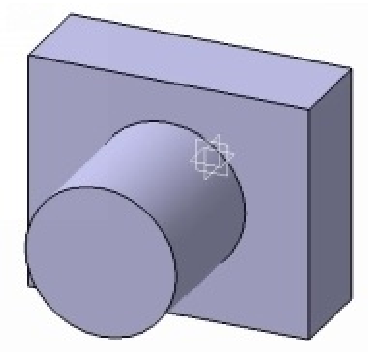
This command assembles two bodies into a single one.

1. Create two or more bodies.
2. On the **Boolean Operations** toolbar, click the **Assemble** button (or) click **Insert > Boolean Operations > Assemble** on the Menu bar.
3. Click on the body to assemble, and then click on main body.
4. Click **OK**.

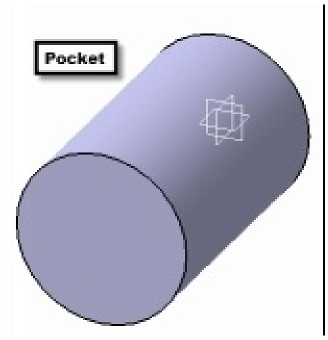
The geometry of the resultant body will depend upon the type of bodies used. For example, if you assemble, a body created using a *Pad* feature; resultant body will be the addition of two bodies.

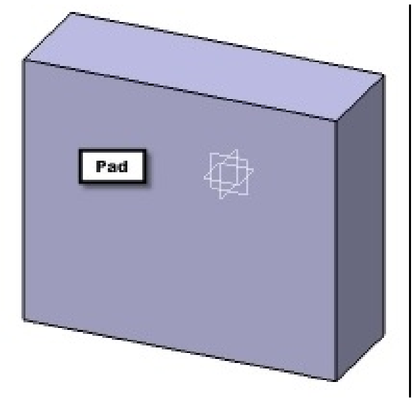


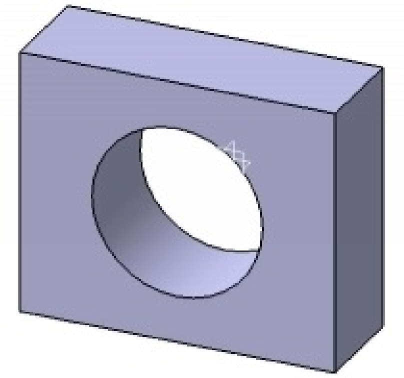




If you assemble a body created using a *Pocket* Feature, then it will result in a pocket in the resultant body.







 Add

This command combines two separate bodies.

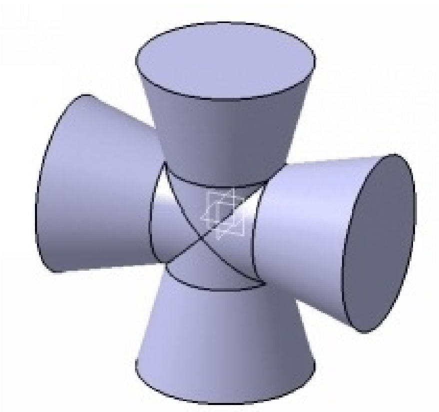
1. On the **Boolean Operations** toolbar, click **Boolean Operations** drop-down > **Add** (or) click **Insert > Boolean Operations > Add** on the Menu bar.
2. Select the two bodies and click **OK**.

Remove

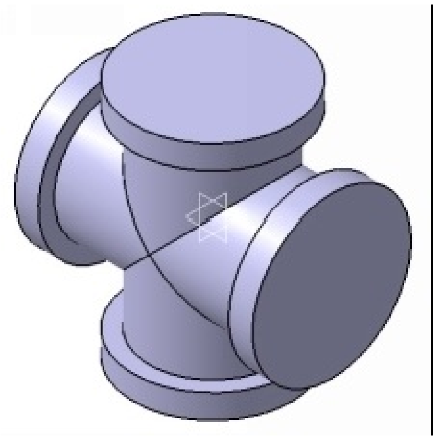
This command performs the function of subtracting one solid body from another.

1. On the **Boolean Operations** toolbar, click **Boolean Operations** drop-down > **Remove** (or) click **Insert > Boolean Operations > Remove** on the Menu bar.
2. Select tool body and the target body.

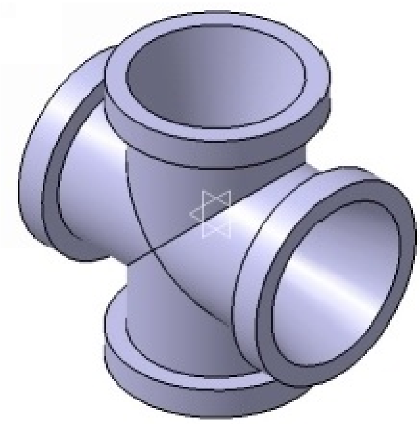
   Tool



Target



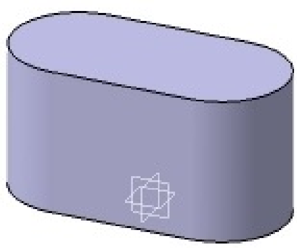
3. Click **OK** to subtract the tool body from the target.

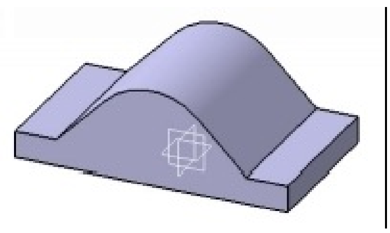


 Intersect

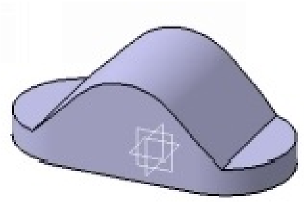
By using the **Intersect** command, you can generate bodies defined by the intersecting volume of two bodies.

1. On the **Boolean Operations** toolbar, click **Boolean Operations** drop-down > **Intersect** (or) click **Insert > Boolean Operations > Intersect** on the Menu bar.
2. Select two bodies.





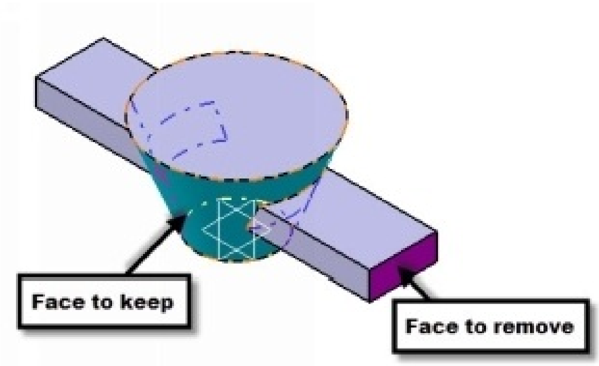
3. Click **OK** to see the resultant single solid body



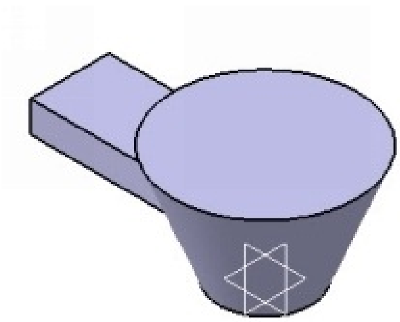
Union Trim

By using the **Union Trim** command, you can combine two bodies and trim the unwanted portion.

1. Create two bodies.
2. Ensure that the **Partbody** is not active.
3. On the **Boolean Operations** toolbar, click the **Union Trim** button (or) click **Insert > Boolean Operations > Union Trim** on the Menu bar.
4. Click on the second body. The **Trim Definition** dialog appears.
5. On the **Trim Definition** dialog, click the **Faces to remove** box and select the faces to remove.
6. Click the **Faces to Keep** box and select faces to keep.



7. Click **OK**.



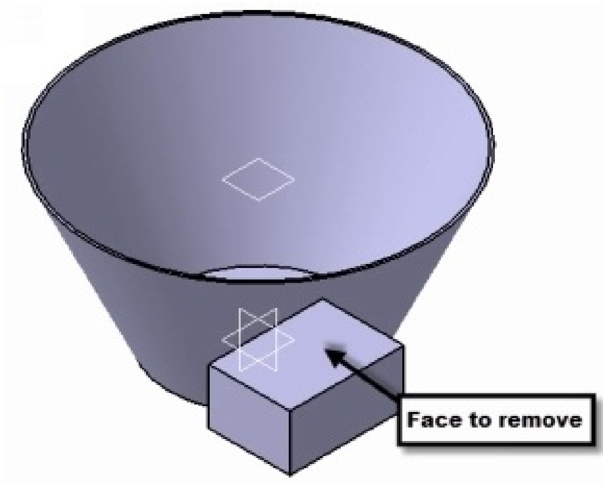
 Remove Lump

This command trims a lump of body from the main part body.

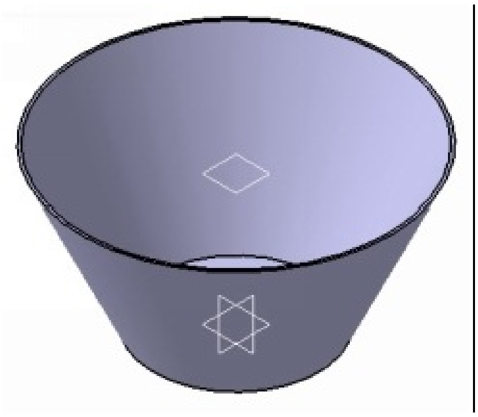
1. Create two bodies and combine them.
2. On the **Boolean Operations** toolbar, click the **Remove Lump** button (or) click **Insert**

> Boolean Operations > Remove Lump on the Menu bar.

3. Click on the part body.
4. On the **Remove Lump** dialog, click the **Faces to remove** box.
5. Select the face of the body to be trimmed.



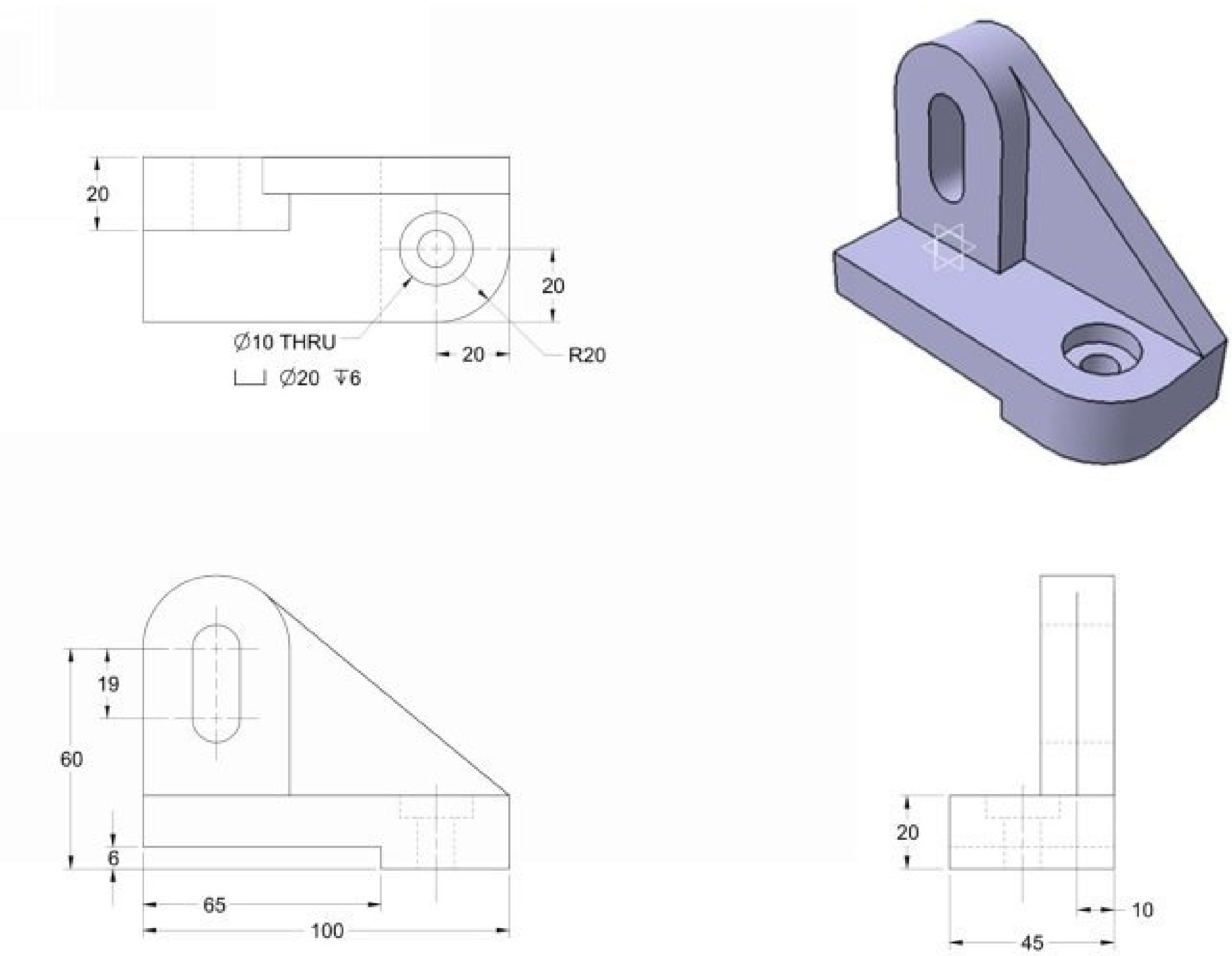
6. Click **OK**.



Examples

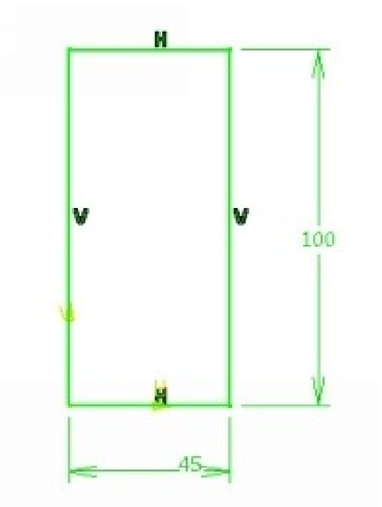
Example 1 (Millimetres)

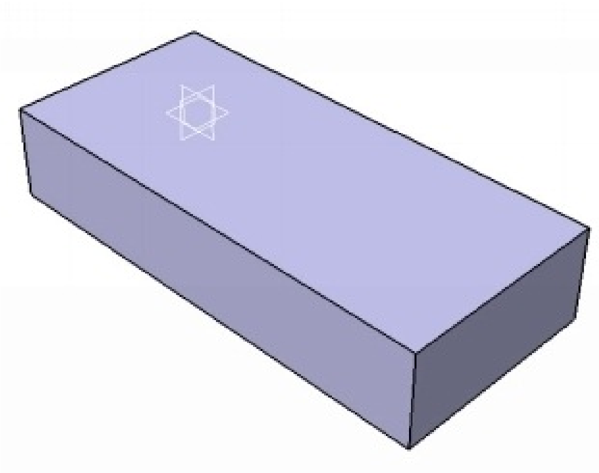
In this example, you will create the part shown next.



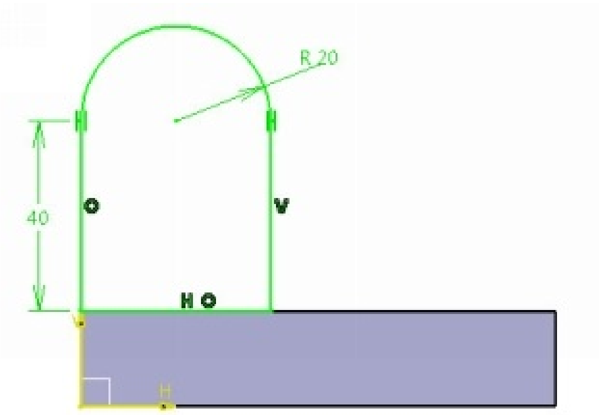
1. Start **CATIA V5-6R2015**.

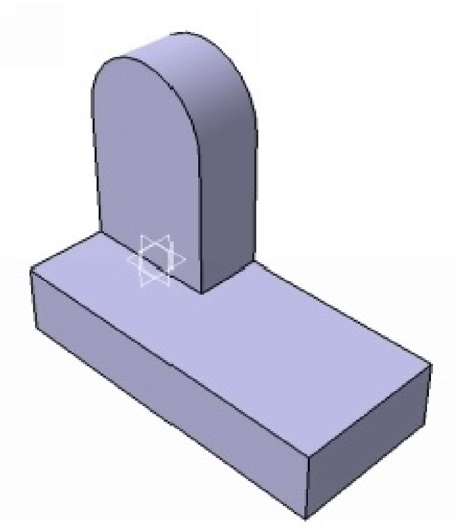
2. Open a new part file.
3. Create the rectangular base on the XY plane. The extrusion depth is 20 mm.



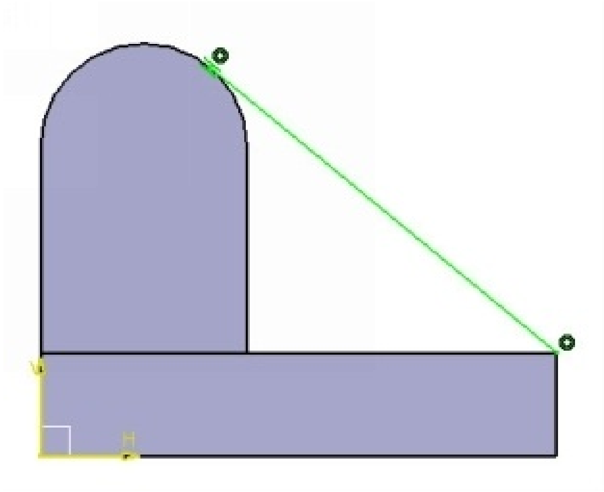


4. Construct the second feature on the YZ plane. The extrusion depth is 20 mm.

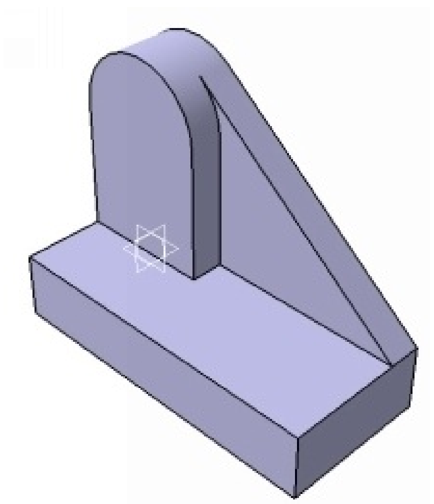




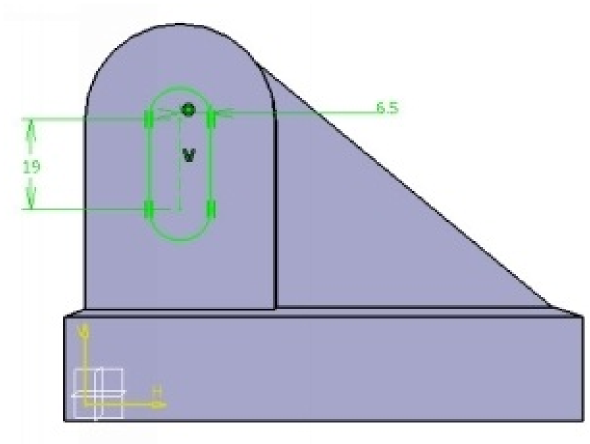
5. On the **Sketch-Based Features** toolbar, click **Advanced drop-down > Stiffener** (or) click **Insert > Sketch-Based Features > Stiffener** on the Menu.
6. On the **Stiffener Definition** dialog, click on the **Sketch** icon and select the yz plane.
7. Draw a line, which is tangent to the curved face of the second feature and connected to the top right vertex of the first feature.



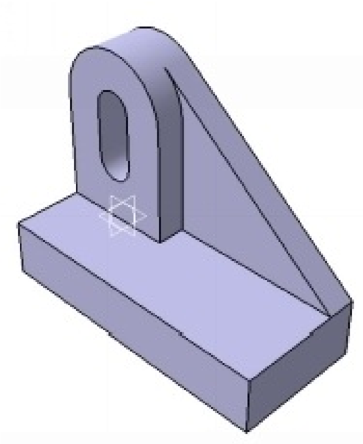
3. Click **Exit workbench**.
9. On the **Stiffener Definition** dialog, under the **Thickness** section, uncheck the **Neutral Fiber** option and click the **Reverse direction** button.
0. Type-in 10 in the **Thickness** box and click **OK** to create the stiffener.



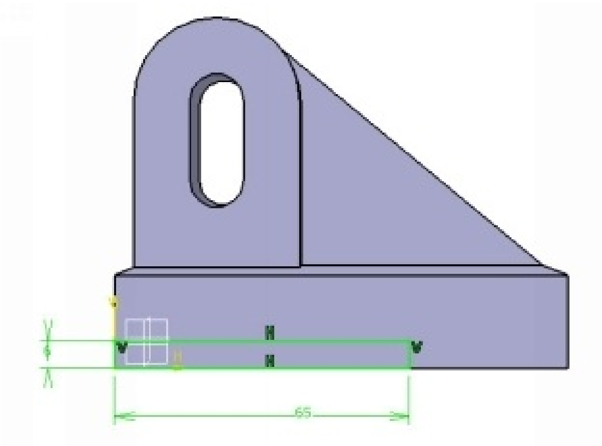
1. Activate the **Pocket** command and click the **Sketch** icon on the dialog.
2. Click on the front face of the second feature, and then draw the sketch for the slot feature.

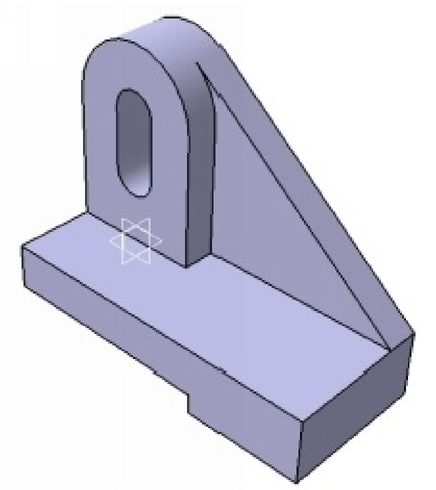


3. Exit the sketch and create the *Pocket* feature.

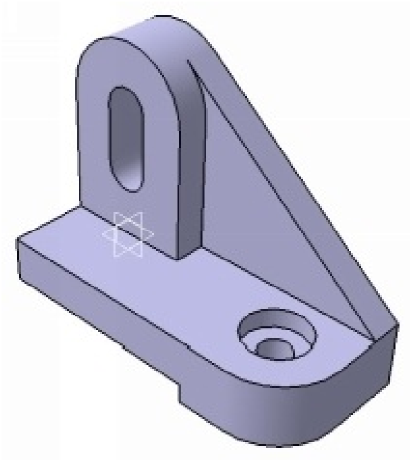


4. Create another *Pocket* feature on the front face of the base.





5. Add an edge fillet and counterbored hole to the model geometry.



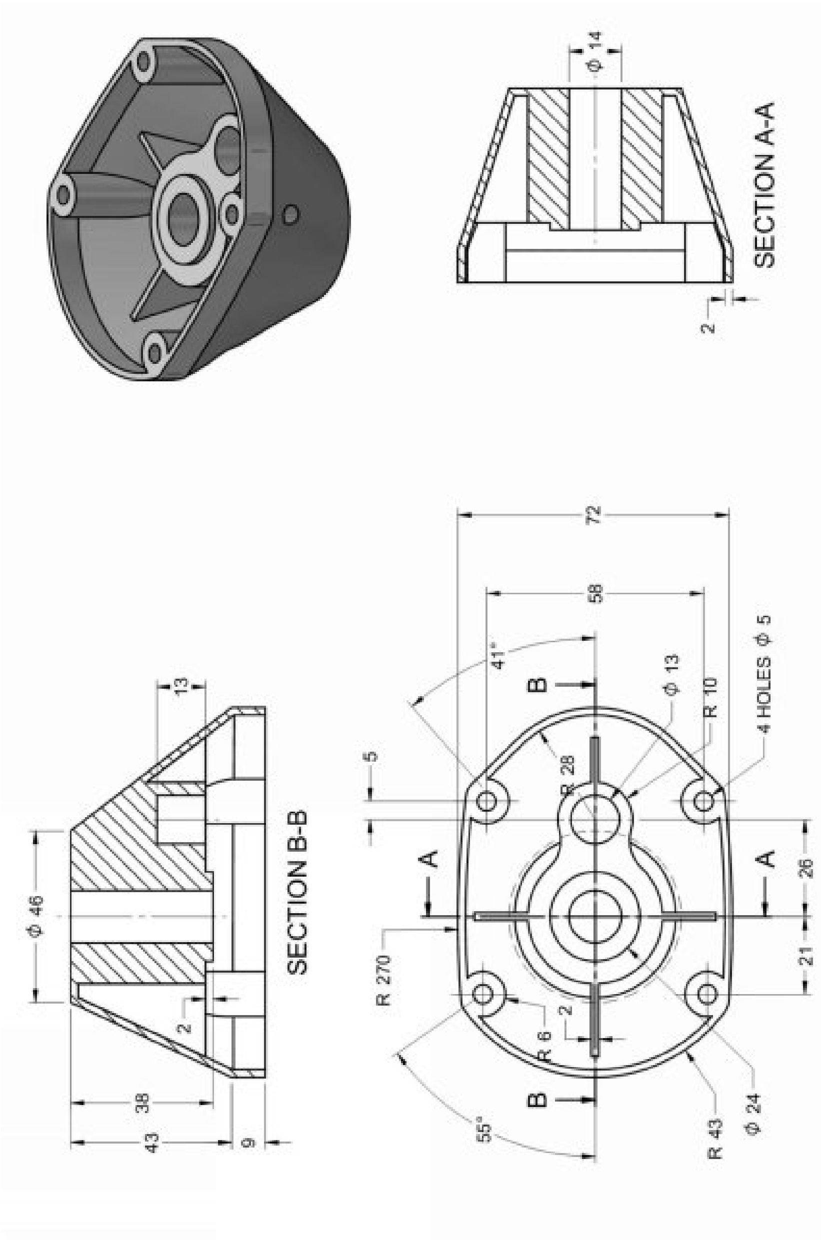
6. Save and close the file.

Questions

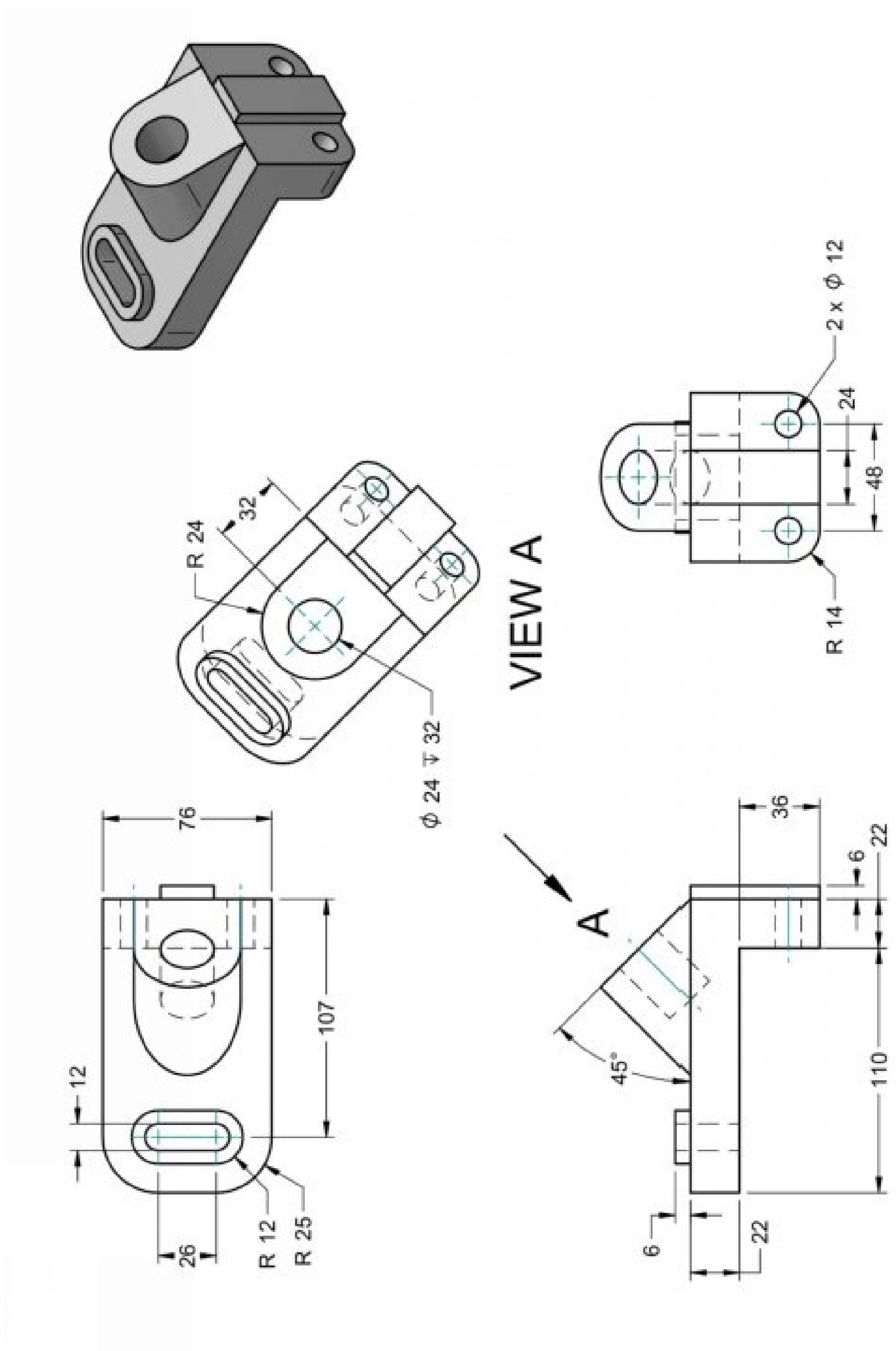
1. What is the use of the **Stiffener** command?
2. Why do we create multi body parts?
3. How do you split a single body into multiple bodies?

Exercises

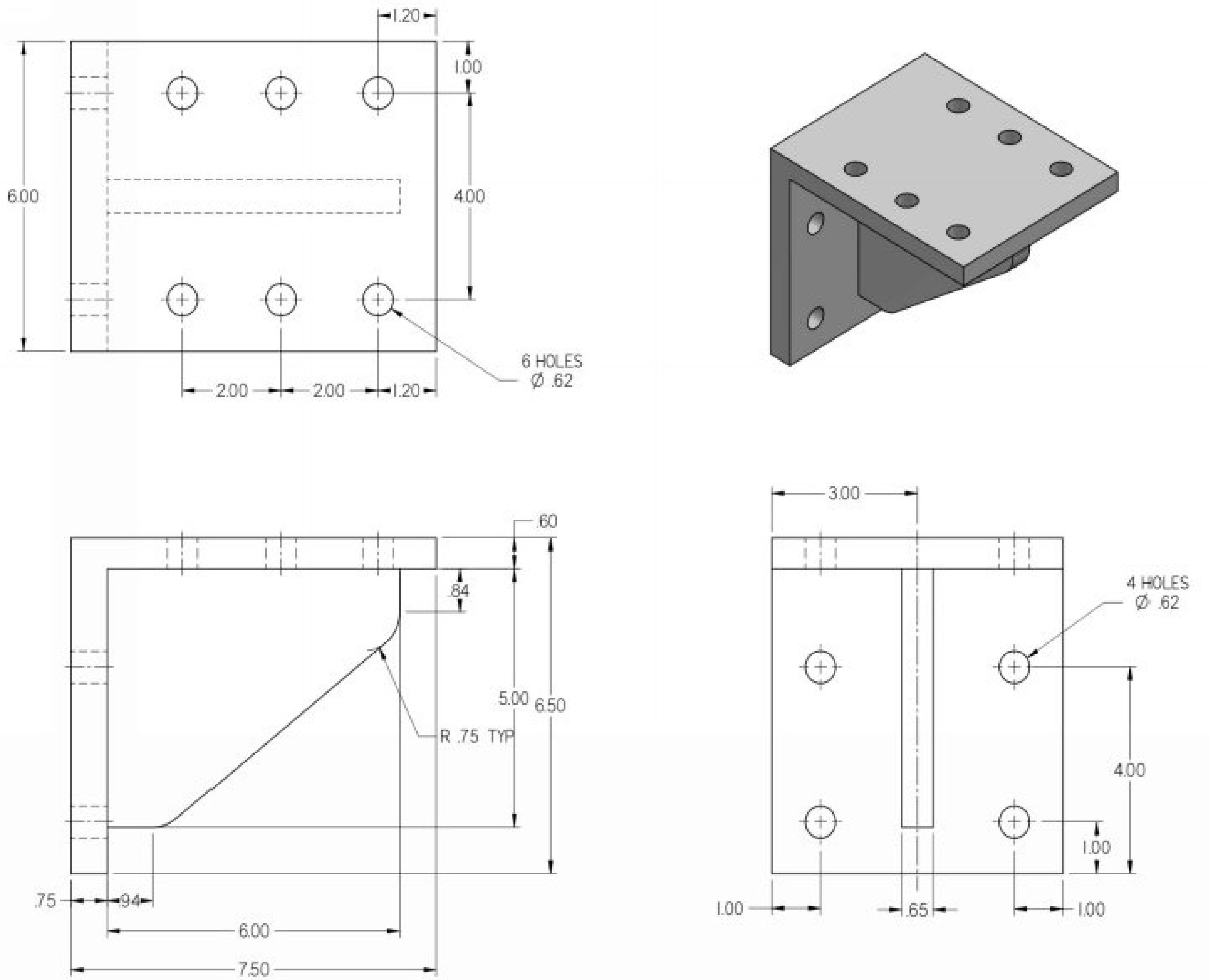
Exercise 1



Exercise 2



Exercise 3 (Inches)


1.20
1.00
6.00
4.00
2.00
2.00
1.20
6 HOLES
Ø .62
.60
.84
5.00
6.50
R .75 TYP
.75
.94
6.00
7.50
3.00
4 HOLES
Ø .62
4.00
1.00
1.00
.65
1.00

Chapter 9: Modifying Parts

In design process, it is not required to achieve the final model in the first attempt. There is always a need to modify the existing parts to get the desired part geometry. In this chapter, you will learn various commands and techniques to make changes to a part.

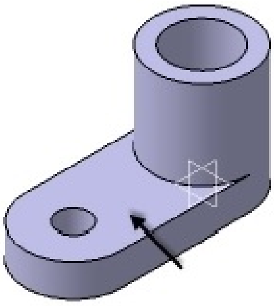
The topics covered in this chapter are:

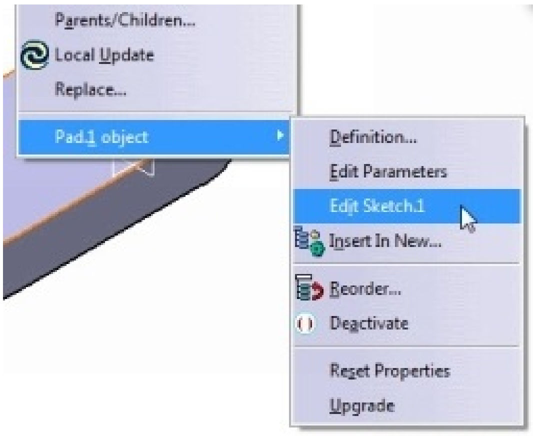
- *Edit Sketches*
- *Edit Feature Parameters*
- *Edit Feature definition*
- *Deactivate and activate features*
- *Change sketch support*

Edit Sketches

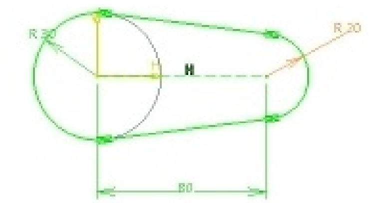
Sketches form the base of a 3D geometry. They control the size and shape of the geometry. If you want to modify the 3D geometry, most of the times, you are required to edit sketches.

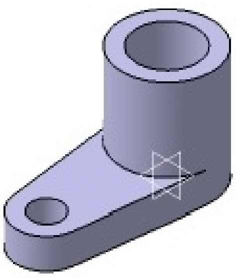
1. To do this, click the right-mouse button on the feature to edit and select **Featurename.object > Edit Sketch**.





2. Now, modify the sketch and click **Exit workbench**. You will notice that the part geometry updates immediately.

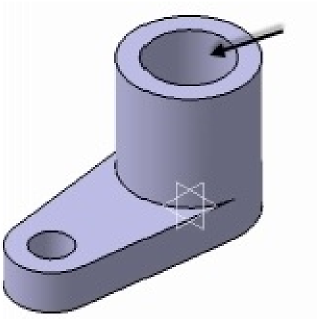


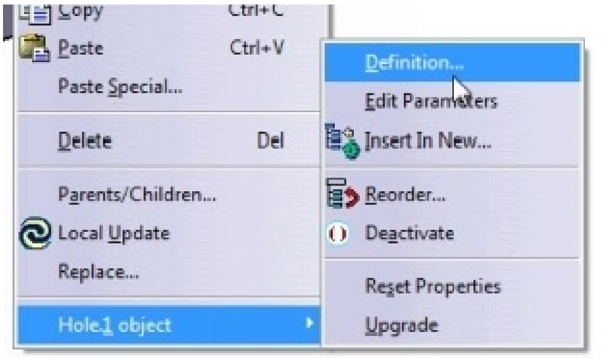


Edit Feature Definition

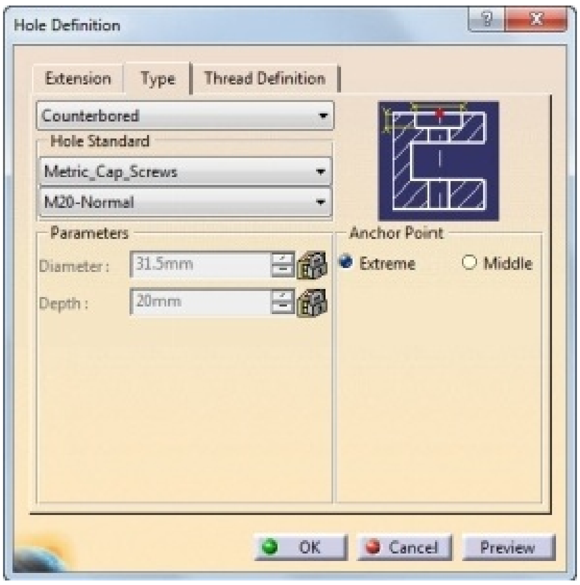
Features are the building blocks of model geometry.

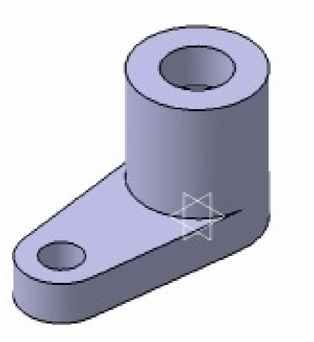
1. To modify a feature, click the right mouse button on it and select **Featurename.object > Definition**. The dialog related to the feature appears.





2. On this dialog, modify the parameters of the feature and click **OK**. The changes take place instantaneously.

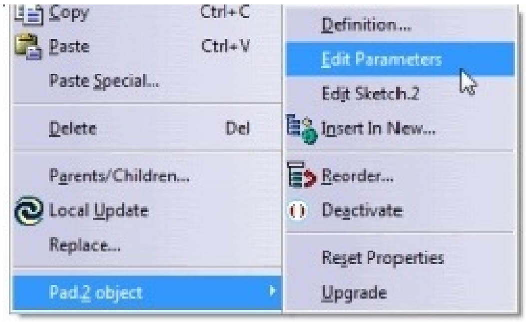


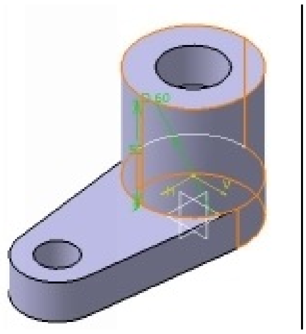


Edit Feature Parameters

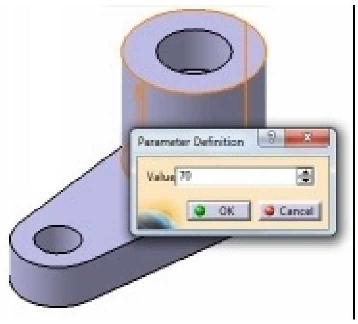
CATIA V5 allows you to modify a feature by editing its parametric dimensions.

1. Click the right mouse button on it and select **Featurename.object > Edit Parameters**. The parameters of the feature appear.

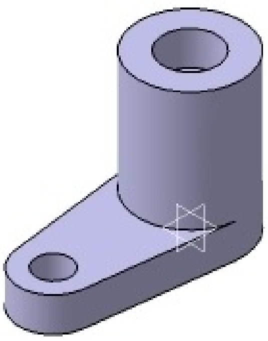




2. To edit a parameter, double-click on it and type-in a new value in the **Parameter Definition** box. Click **OK**.



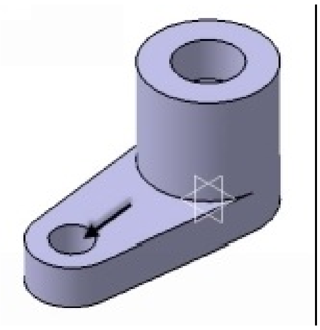
3. On the **Tools** toolbar, click the **Update All**  button to update the feature.

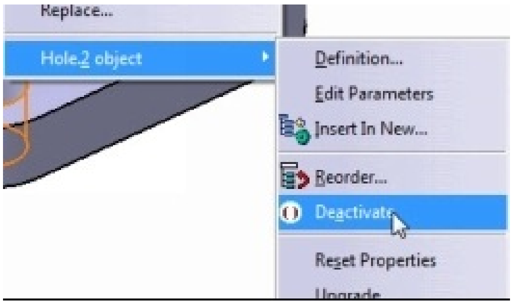


Deactivate Features

Sometimes you may need to deactivate some features of model geometry.

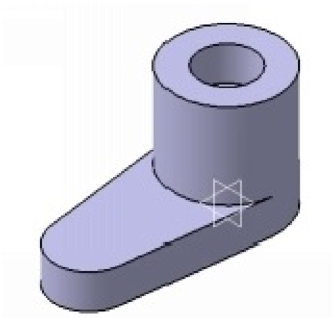
1. To do this, click the right mouse button on a feature and select **Deactivate**.





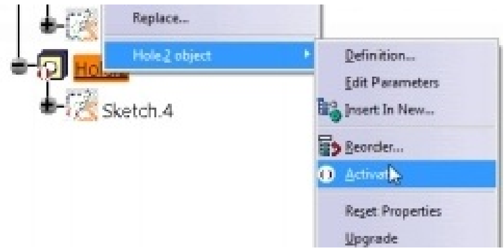
On the **Deactivate** dialog, check the **Deactivate aggregated elements** option if you want to deactivate all the sketches and references elements related to the feature. Otherwise, uncheck this option.

2. Click **OK** to deactivate the feature.

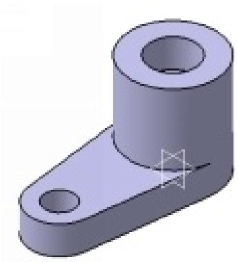


Activate Features

1. If you want to activate the deactivated features, then expand the Specification Tree.
2. Click the right mouse button on the deactivated feature and select **Featurename.object > Activate**.



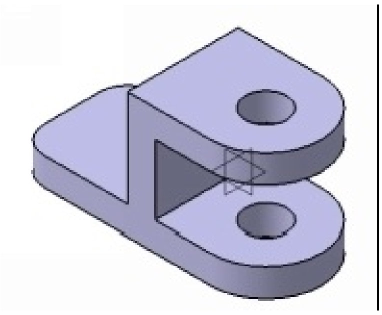
3. Click **OK**. The feature will become active again.

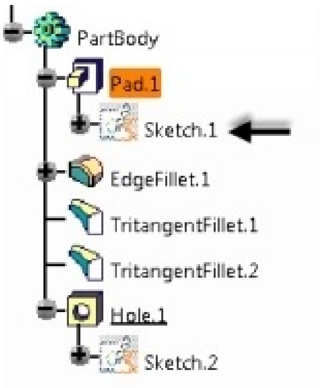


Changing the Sketch Support

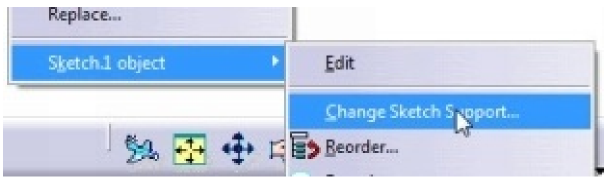
CATIA V5 allows you to change the sketch support of a feature.

1. Expand the **Specification Tree** and go to the feature to modify.

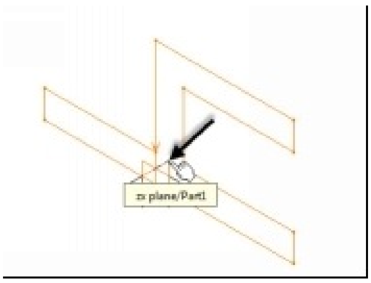




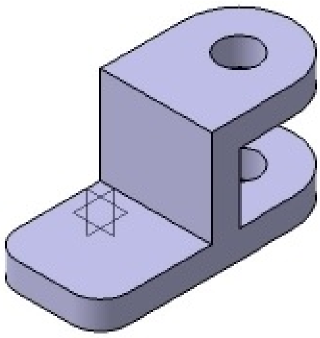
2. Click the right mouse button on the sketch of the feature and select **Sketchname.object > Change Sketch Support**.



3. Select a different plane to define the new sketch support. You can select it from the graphics window or Specification Tree.



4. On the **Sketch Positioning** dialog, check the **Move Geometry** option and click **OK**. The orientation of the geometry will change accordingly.

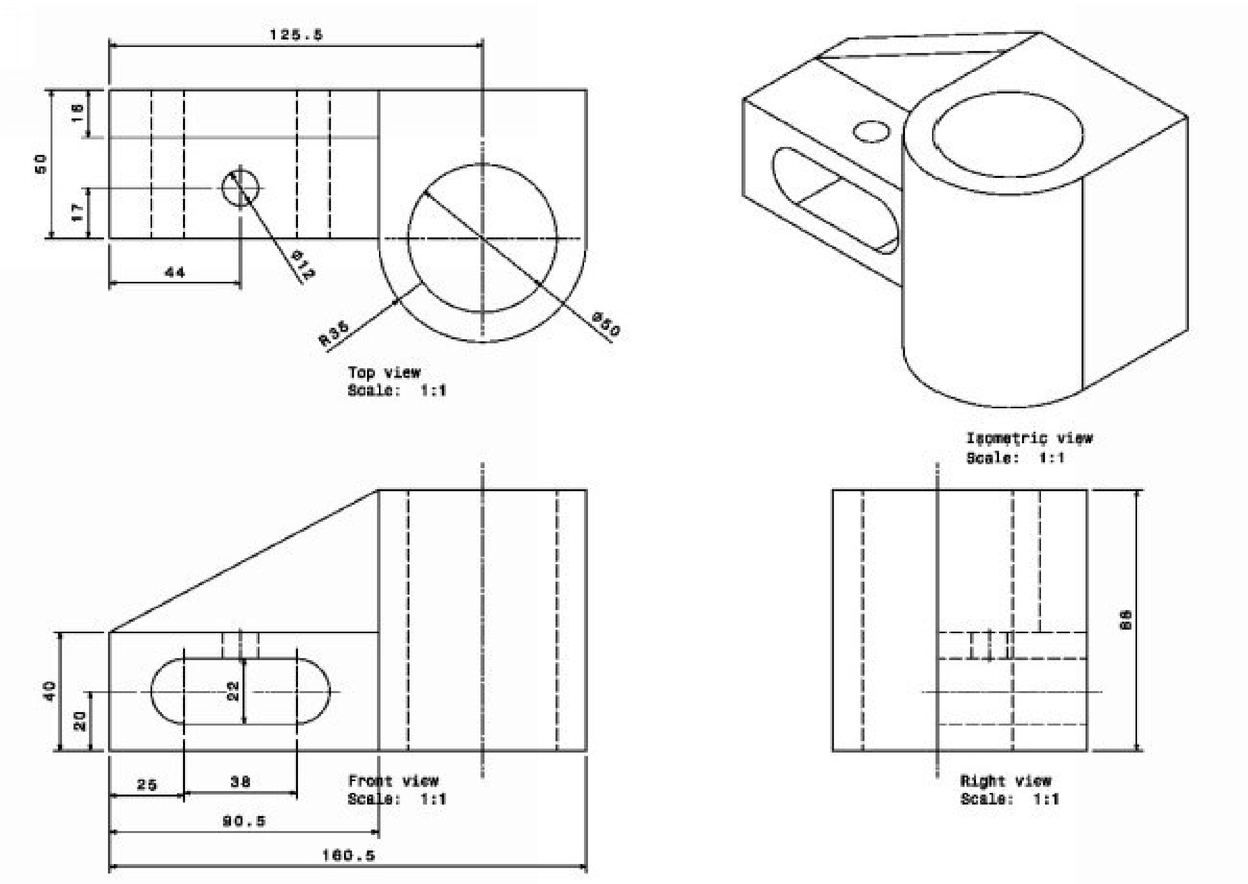


If there are any sketches or features on the geometry, it may show some warnings and error messages. You have to manually solve these problems or avoid changing the sketch support when the model becomes complex. It is recommended that you select correct plane initially based on model orientation.

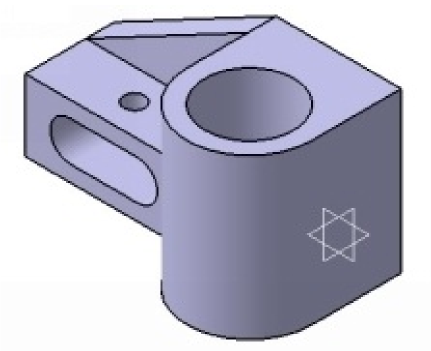
Examples

Example 1

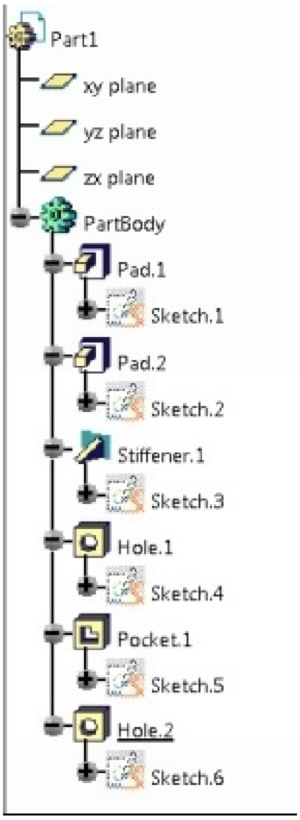
In this example, you will create the part shown below, and then modify it.



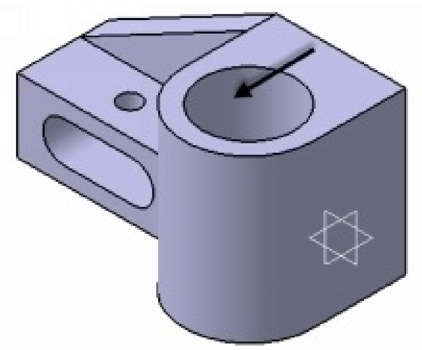
1. Start **CATIA V5-6R2015** and open a part file.
2. Create the part using the tools and commands in CATIA V5.



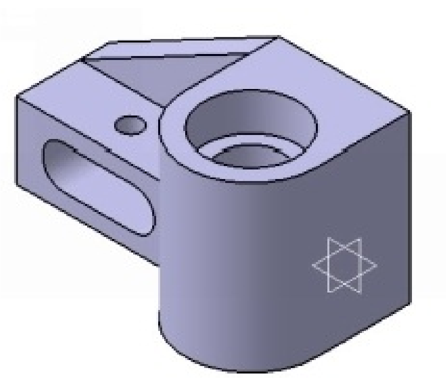
The Specification Tree of the part is given next.



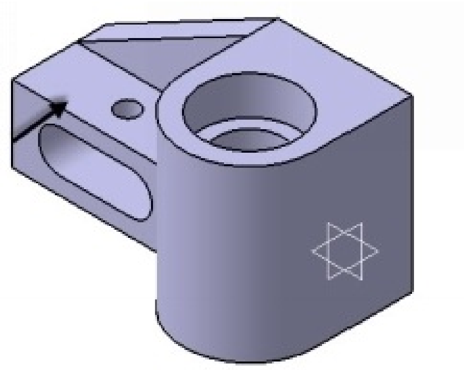
3. Double-click on the 50 mm diameter hole. The **Hole Definition** dialog appears.



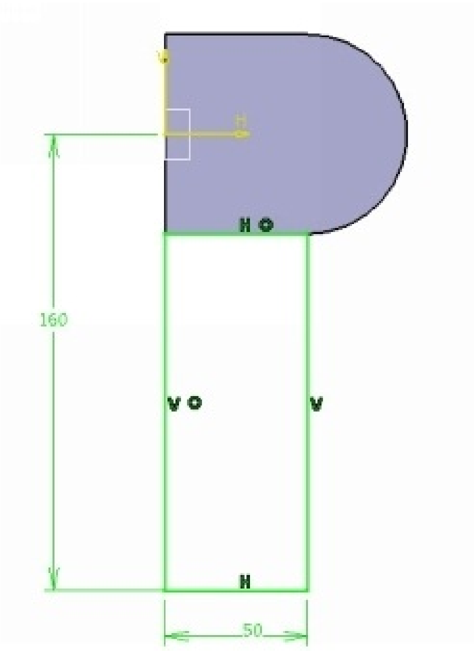
4. On the **Hole Definition** dialog, click the **Type** tab, and then select **Counterbored** from the drop-down.
5. Set the counterbore **Diameter** to **50** and **Depth** to **20**.
6. Click the **Extension** tab and set the **Diameter** to 35. Click **OK** to close the dialog.



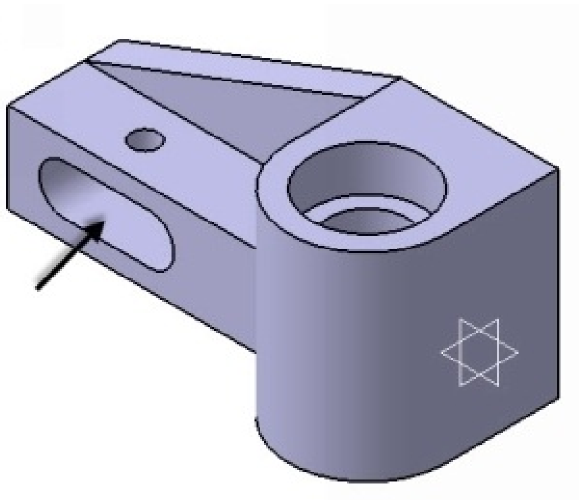
7. Click the right mouse button on the rectangular pad feature and select **Pad.object > Edit Sketch**.



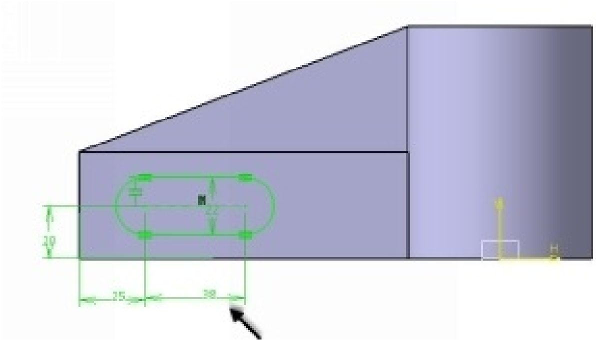
8. Modify the sketch, as shown below.

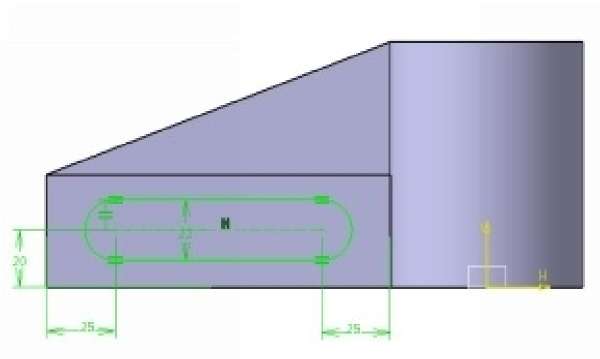


9. Exit the sketch.
10. Click the right mouse button on the slot feature and select **Pocket1.object > Edit Sketch**.

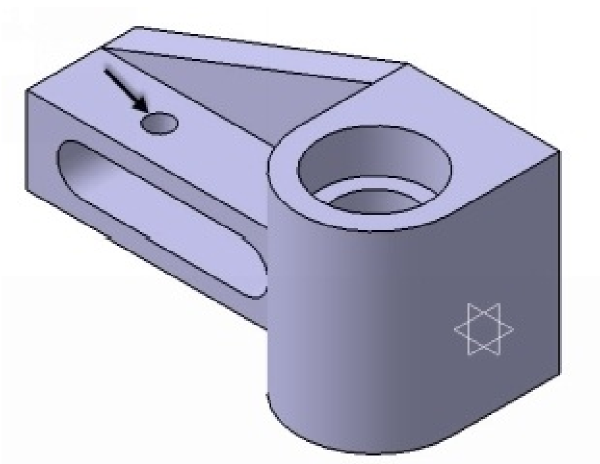


11. Delete the length dimension of the slot, and then add a new dimension between the right-side arc and right vertical edge.

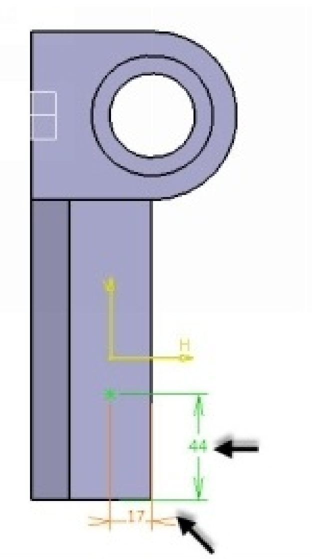




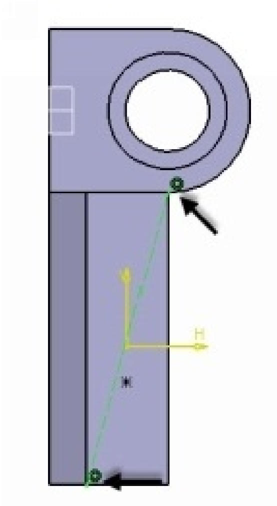
12. Exit the sketch.
13. Double-click on the small hole, and then click the **Positioned sketch** button on the dialog.



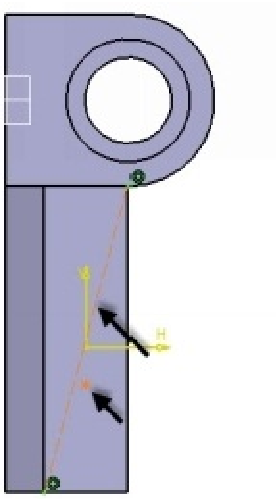
14. Delete the positioning dimensions.



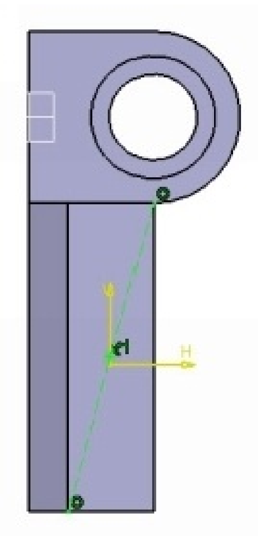
15. Create a construction line and make its ends coincident with the corners, as shown below.



16. Press the Ctrl key and select the hole point and the construction line.

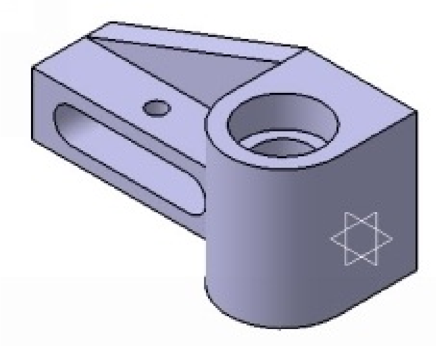


17. Activate the **Constraint Define in Dialog Box** command.
18. On the **Constraint Definition** dialog, check the **Midpoint** option, and then click **OK**. This positions the hole point on the midpoint of the construction line.

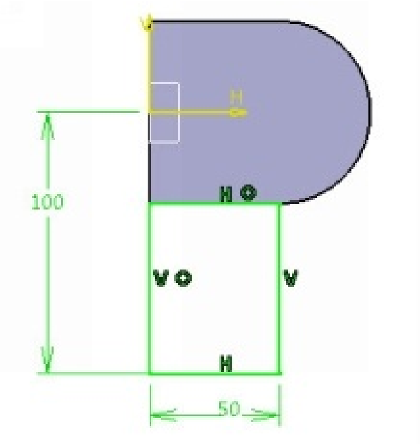


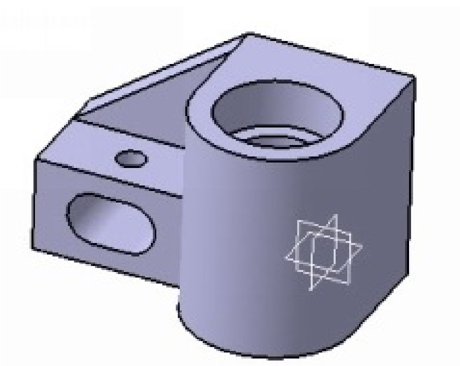
19. Exit the sketch.

20. Click **OK** on the **Hole Definition** dialog.



21. Now, change the size of the pad feature. You will notice that the slot and hole are adjusted automatically.





22. Save and close the file.

Questions

1. How do you modify the sketch of a feature?
2. How do you modify a feature directly?
3. How can you change the orientation of the model?

Exercises

Exercise 1