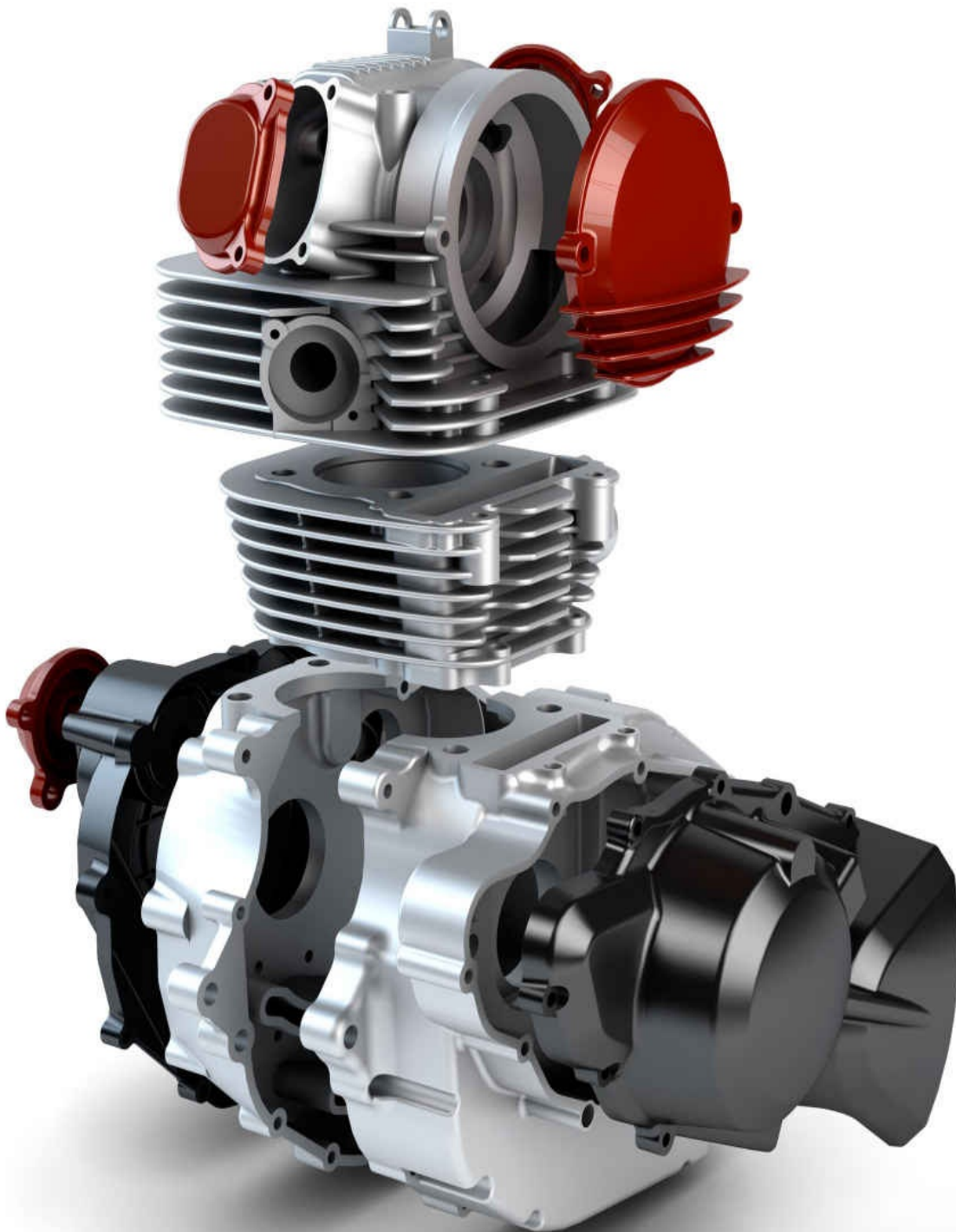


CATIA V5-6R2015 Basics - Part III

Assembly Design, Drafting, Sheetmetal Design, and
Surface Design



Tutorial Books

CATIA V5-6R2015
Basics-Part III
Tutorial Books

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Chapter 10: Assemblies

After creating individual components, you can bring them together into an assembly. By doing so, it is possible to identify incorrect design problems that may not have been noticeable at the part level. In this chapter, you will learn how to bring components into the Assembly Design Workbench and position them.

The topics covered in this chapter are:

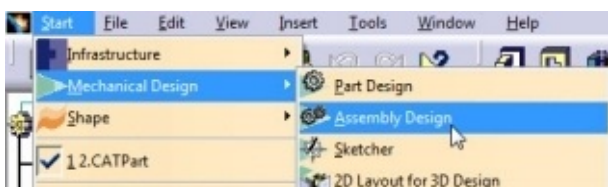
- *Starting an assembly*
- *Inserting Components*
- *Adding Constraints*
- *Moving components*
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Starting an Assembly

To begin an assembly file, you can use the **New** icon and select the **Product** template.




Another way to start an assembly is to click **Start > Mechanical Design > Assembly Design**. This opens the assembly environment. Now, add components to the assembly using the **Existing Components** command.



Inserting Components

There are two different methods to insert an existing part into an assembly. The first one is to insert using the **Existing Component** command.

1. On the **Product Structure Tools** toolbar, click the **Existing Component**  button (or) click **Insert > Existing Component** on the Menu bar.
2. In the Specification Tree, click **Product**.



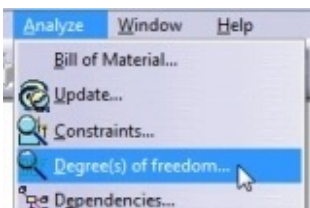
3. Browse to the Location of the component and select it.
4. Click **Open**. The component is inserted in the assembly.

Fixing the first Component

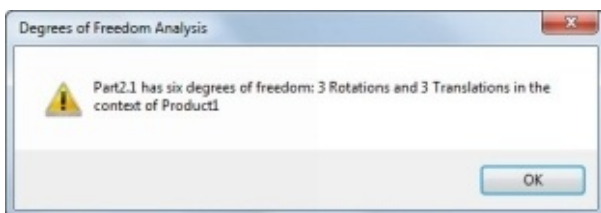
After inserting components into an assembly, you have to define constraints between them. By applying constraints, you can make components to flush with each other or two cylindrical faces concentric with each other, and so on. As you add constraints between components, the degrees of freedom will be removed from them. By default, there are six degrees of freedom for a part (three linear and three rotational). Eliminating degrees of freedom will make components attached and interact with each other as in real life. Now, you will learn to add constraints between components


After placing the component at the origin, it is free to move. You can check the degrees of freedom by using the **Degree(s) of freedom** command.

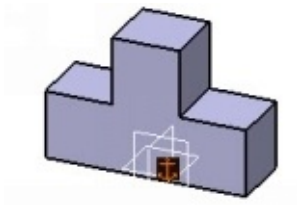
1. Double-click on the Part in the Specification Tree.
2. On the Menu bar, click **Analyze > Degree(s) of freedom**.



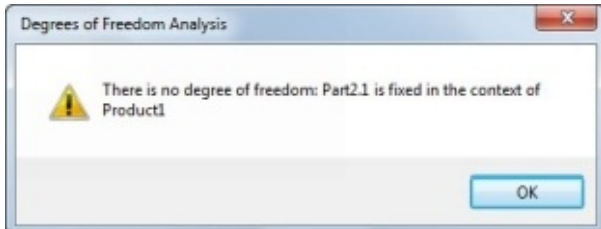
A warning message appears showing the degrees of freedom of the selected component.




3. In order to remove the degrees of freedom of the first component, double-click on **Product1** in the Specification Tree.
4. On the **Constraints** toolbar, click the **Fix Component**  button (or) click **Insert > Fix** on the Menu bar.
5. Select the first component to fix it at the origin. You will notice that the **Fix** glyph appears on the components.

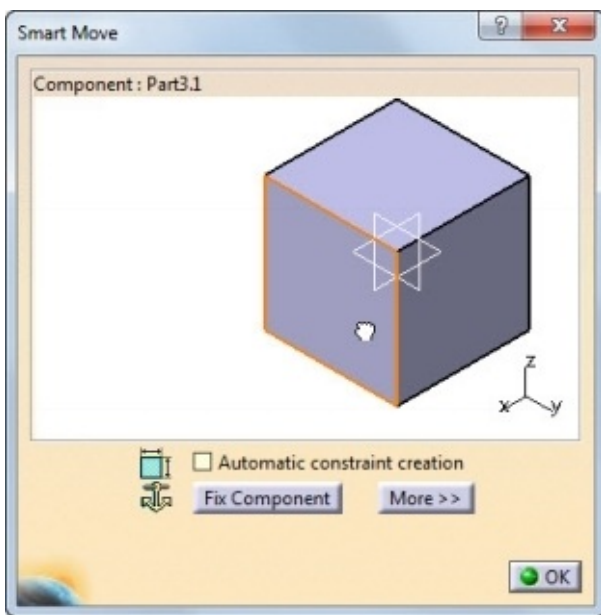


5. Now, analyze the degrees of freedom of the component. You can notice that the component is fully fixed at the origin.

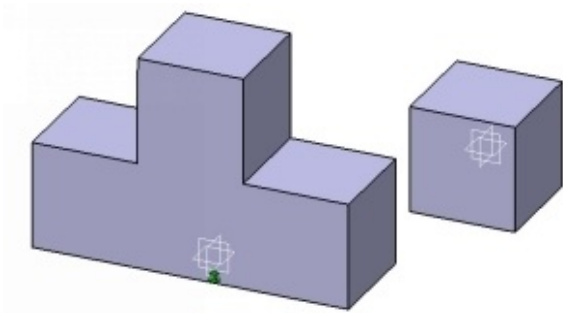


Inserting the Second Component

1. On the **Product Structure Tools** toolbar, click the **Existing Component With Positioning**  button (or) click **Insert > Existing Component with Positioning** on the Menu bar.
2. In the Specification Tree, click **Product1** and go to the location of the second component.
3. Select the component and click **Open**. The **Smart Move** dialog appears on the screen.
4. In the **Smart Move** dialog, you can click and drag the component to the required location.







5. Click **OK**.

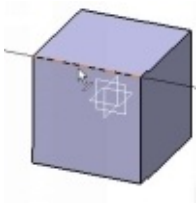








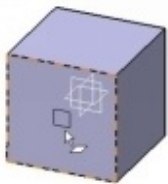
Manipulation





After inserting components into the assembly, you can move or rotate them.

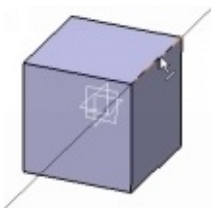
1. On the **Move** toolbar, click the **Manipulation** button (or) click **Edit > Move > Manipulate**.
2. If the component is constrained, then check the **With respect to constraints** option on the **Manipulation Parameters** dialog. This allows you to manipulate the component by considering the applied constraints.
3. On the **Manipulation Parameters** dialog, click the **Drag along X axis**  button, and then drag the component along the x-axis.
4. Likewise, use the **Drag along Y axis** , **Drag along Z axis**  buttons drag the component along Y and Z axes, respectively.
5. Click the **Drag along any axis**  button and click anyone of the edges of the component. Drag the pointer to move the component along the selected edge.



6. Use the **Drag along XY plane** , **Drag along YZ plane** , or **Drag along XZ plane**  buttons to move the component on three different planes.
7. Click the **Drag along any plane**  button and select a face or plane of the component. Drag the pointer to move the component on the selected plane.



3. Use the **Drag around X axis** , **Drag around Y axis** , or **Drag around Z axis**  buttons to rotate the component around three different axes .
4. Click the **Drag around any axis**  button and select an edge of the component. Drag the pointer to rotate the component around the selected edge.



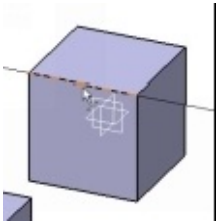


0. Click **OK**.

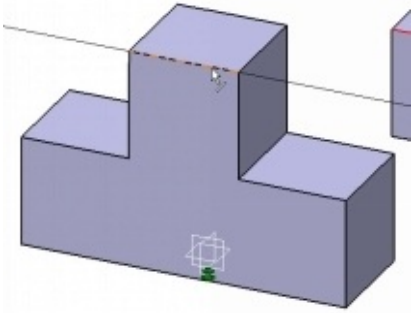
Snap

This command moves a component by snapping the elements (edges, planes) of two components.

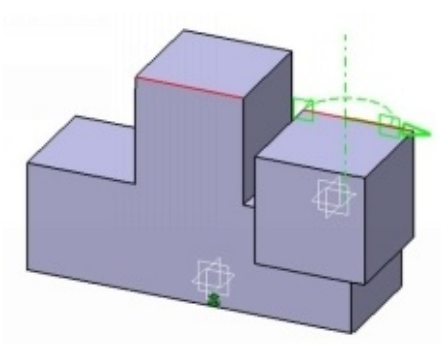
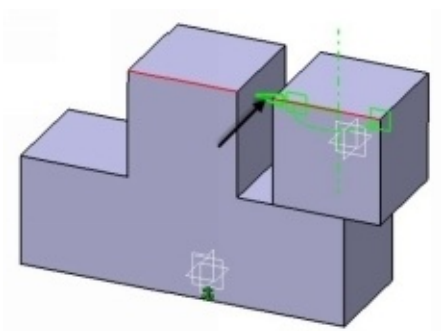
1. On the **Move** toolbar, click **Snap** drop-down > **Snap** (or) click **Edit** > **Move** > **Snap** on the Menu bar.
2. Click on the edge of the first component.



3. Click on the edge of the second component. The two edges will be aligned.



4. Click on the green arrow to reverse the alignment direction.

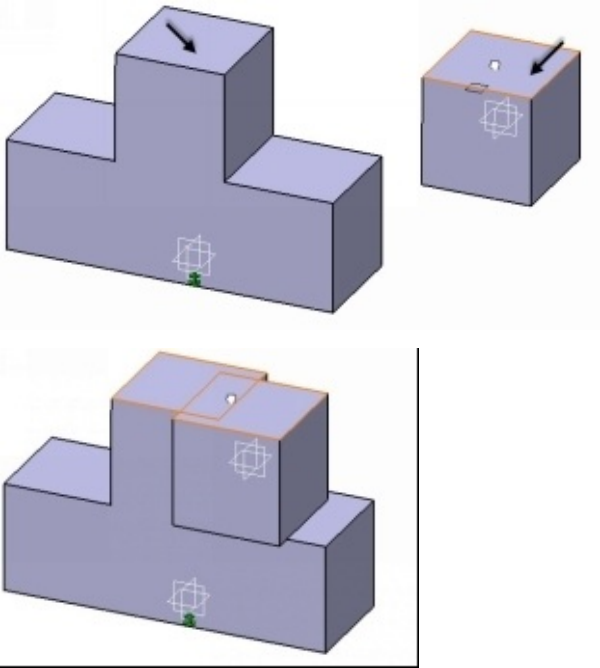




Smart Move

This command and the **Snap** command function in same way. The advantage of this command is that you can create constraints between two snapped objects.

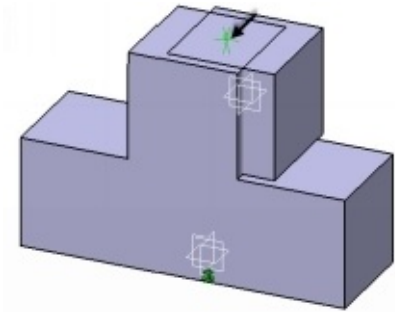
1. On the **Move** toolbar, click **Snap** drop-down > **Smart Move** (or) click **Edit > Move > Smart Move** on the Menu bar.
2. Click on an object (edge or face) of the first component, and then drag it onto the second component. The two objects will be aligned together.



3. Click the green arrow if you want to reverse the alignment direction.
4. On the **Smart Move** dialog, check the **Automatic constraint creation** option to apply constraints between the aligned objects.
5. On the **Smart Move** dialog, click the **More** button to view the type of constraints that can be applied.
6. If you want to apply only a particular type constraint between the two objects, then select the constraint from the list.
7. Move the constraint to the top of the list by clicking the upward arrow button.



3. Check the **Create verified constraints first** option, and then click **OK**. The constraint will be applied between the two objects.

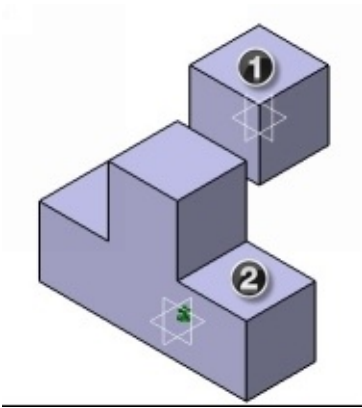




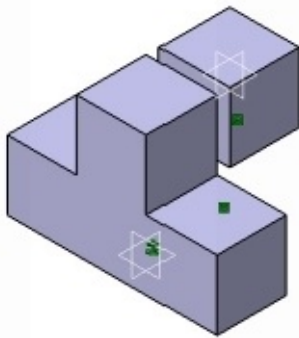
Contact Constraint

The **Contact Constraint** command makes two faces coincident and opposite to each other.

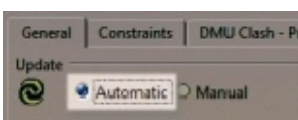
1. On the **Constraints** toolbar, click the **Contact Constraint** button (or) click **Insert > Contact** on the Menu bar.
2. Select a face of the first part.
3. Click on a face of the second part. This creates a contact constraint between the two faces.



4. On the **Update** toolbar, click the **Update All**  button to update the positions components.



If you want to update the assembly automatically, then open the **Options** dialog (click **Tools > Options** on the Menu bar). On the **Options** dialog, click **Mechanical Design > Assembly Design**. Select **Update > Automatic** option, and then click **OK**. However, this may slow down the speed due to instant updates.

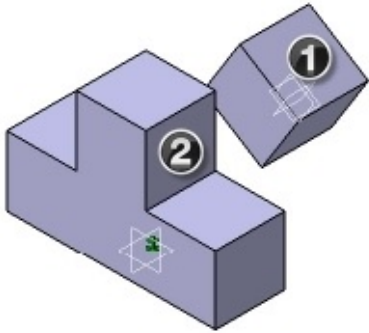




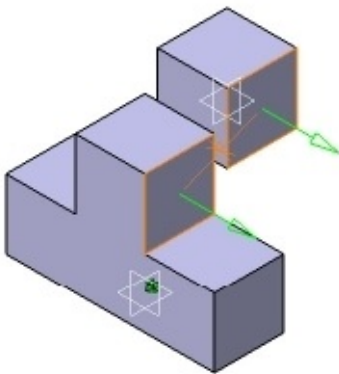
Offset Constraint

The **Offset Constraint** command creates a distance between two faces. In addition, the faces will be parallel to each other.

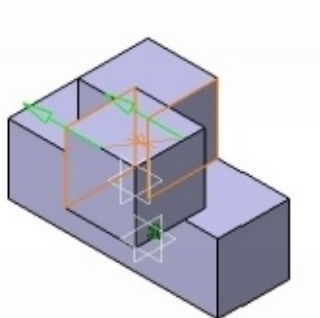
1. On the **Constraints** toolbar, click the **Offset Constraint** button (or) click **Insert > Offset** on the Menu bar.
2. Select a face of the first part.
3. Click on a face of the second part.



4. On the **Constraint Properties** dialog, select **Orientation > Same** to make the selected faces point in same direction.



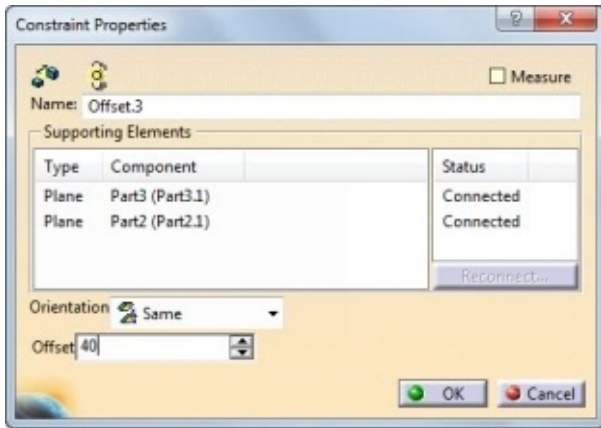
If you select **Orientation > Opposite**, the faces point in the direction opposite to each other.



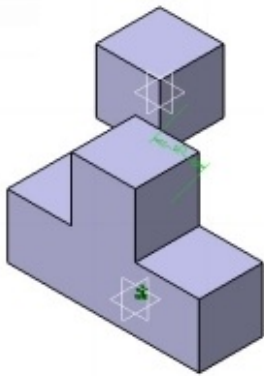
If you select **Orientation > Undefined**, the faces point in the current direction.

5. Type-in a value in the **Offset** box (or) if you check the **Measure** option, the current

distance between the selected faces will be used as offset distance.



5. Click **OK**.

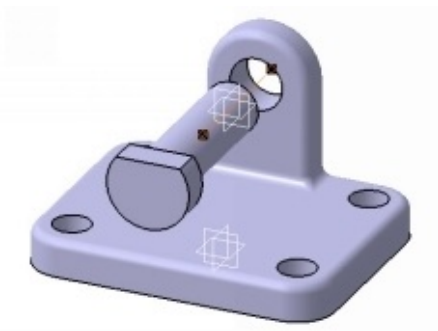
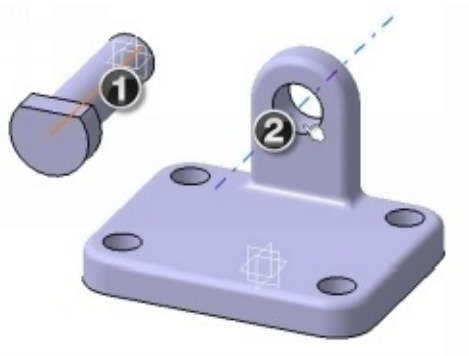




Coincidence Constraint

The **Coincidence Constraint** command makes the axes of two cylindrical faces coincide with each other.

1. On the **Constraints** toolbar, click the **Coincidence Constraint** button (or) click **Insert** > **Coincidence** on the Menu bar.
2. Click on a cylindrical face, linear edge, or axis of the first part.
3. Click on a cylindrical face, linear edge, or axis of the target part. The two cylindrical axes will be aligned together.





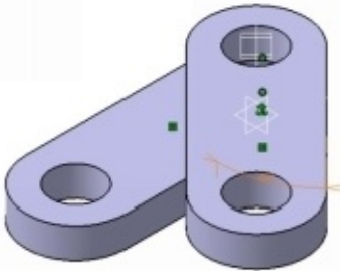
Angle Constraint

The **Angle Constraint** command is used to position faces at a specified angle.

1. On the **Constraints** toolbar, click the **Angle Constraint** button (or) click **Insert > Angle** on the Menu bar.
2. Click on a plane or linear element of the first part.
3. Click on a plane or linear element of the second part.



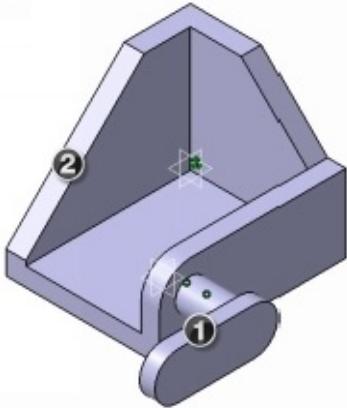
4. On the **Constraint Properties** dialog, select **Sector 1**, **Sector 2**, **Sector 3**, or **Sector 4** from the **Sector** drop-down menu. The angle value changes for each sector.
5. Type-in a value in the **Angle** box on the **Constraint Properties** dialog. Click **OK** to position the first part at the specified angle.



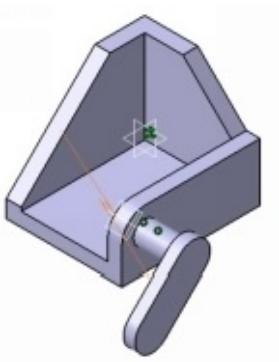
Parallelism and Perpendicularity Constraints

The **Angle Constraint** command can also be used to make an axis, face or edge of one part parallel to that of another part.

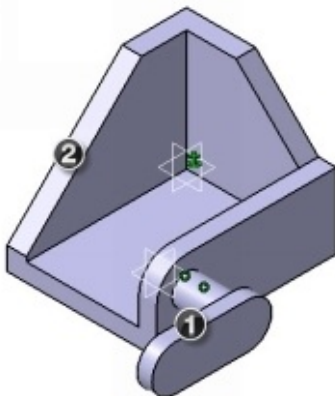
1. Activate the **Angle Constraint** command and select a planar face, cylindrical face, linear edge, or axis of the first part.
2. Next, click on an element of the second part.

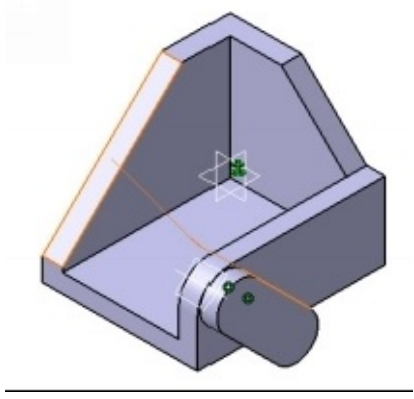


3. On the **Constraint Properties** dialog, select the **Parallelism** option.
4. On the dialog, set the **Orientation** and click **OK**. The selected elements will be parallel to each other.



Likewise, you can make two faces, edges, or axes perpendicular to each other using the **Perpendicularity** option on the **Constraint Properties** dialog.

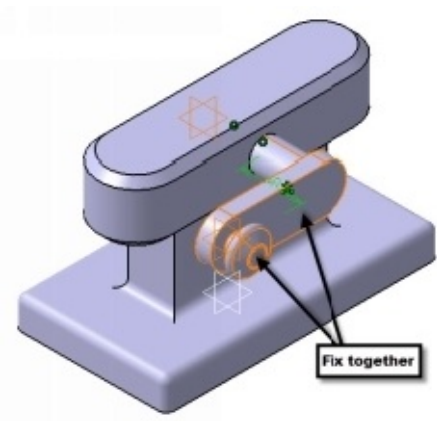





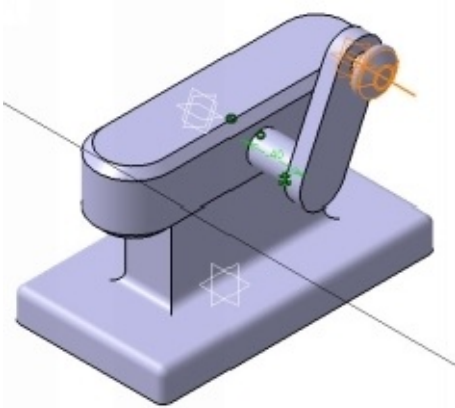
Fix Together

The **Fix Together** constraint makes components to form a rigid set. As you move a single part in a rigid set, all the other components will also be moved.

1. On the **Constraints** toolbar, click the **Fix Together** button (or) click **Insert > Fix Together** on the Menu bar.
2. Select components from the assembly window.

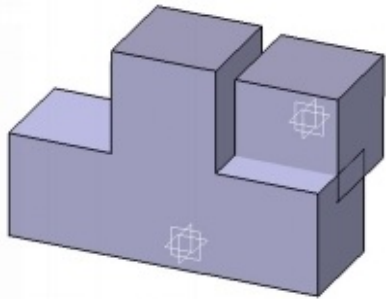


3. Click **OK** on the dialog. The selected components are fixed together.
4. Now, activate the **Manipulation**  command and check the **With respect to constraints** option.
5. Rotate or move anyone of the parts, which are fixed together. You will notice that the other parts are also manipulated.

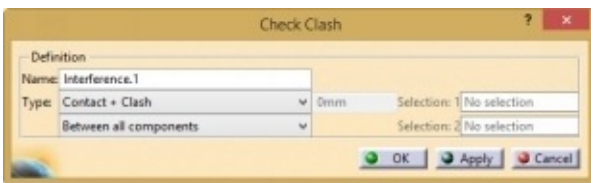


Clash

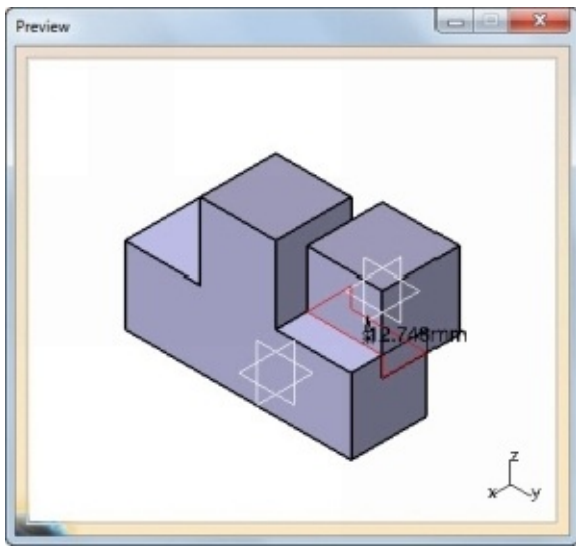
In an assembly, two or more components can overlap or occupy the same space. However, this would be physically impossible in the real world. When you add constraints between components, CATIA V5 develops real-world contacts and movements between them. However, sometimes clashes can occur. To check such errors, CATIA V5 provides you with a command called **Clash**.




1. On the **Space Analysis** toolbar, click the **Clash** button (or) click **Analyze > Clash** on the menu bar.
2. On the **Check Clash** dialog, select the type of clash analysis. You can select **Contact + Clash** or **Clearance + Contact + Clash**, **Authorized Penetration**, or **Clash Rule**.
3. Select **Between all components** from the lower drop-down menu. You can also select **Inside one selection**, **Selection against all**, or **Between two selections**. This defines components between which the clash analysis is performed.



4. Click **Apply** on the dialog. The preview window appears showing the interference value.

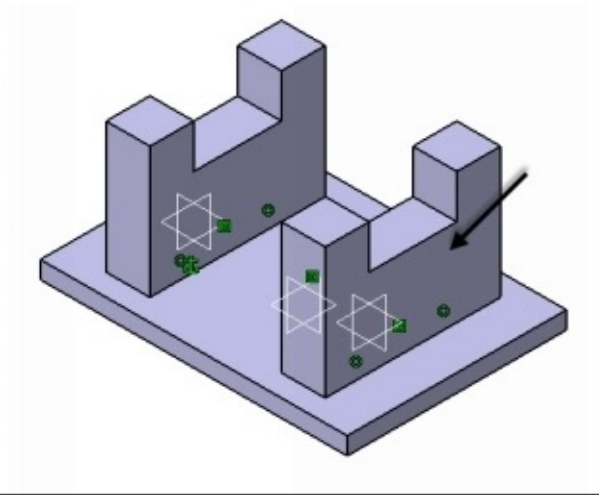


Now, you can export the clash report by using the **Export As**  button.

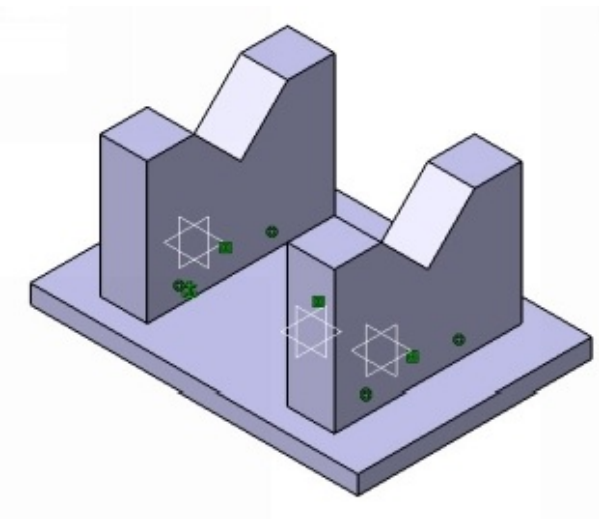
Editing and Updating Assemblies

During the design process, the correct design is not achieved on the first attempt. There is always a need to go back and make modifications. CATIA V5, allows you to accomplish this process very easily.

1. To modify a part in an assembly, double click on it. This activates the **Part Design** workbench.



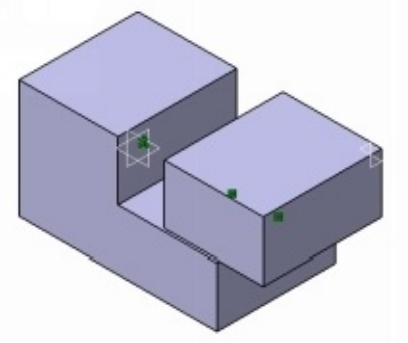
2. Now, make changes to the part.



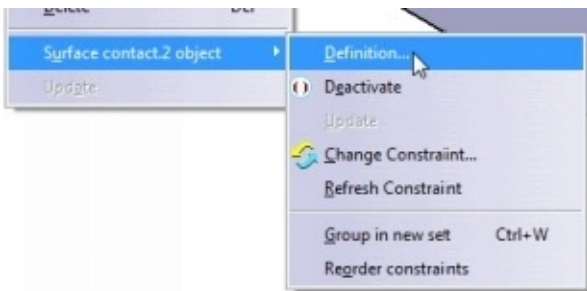
3. In the Specification Tree, double-click on **Product1** to return to the **Assembly Design** workbench.

Redefining Constraints

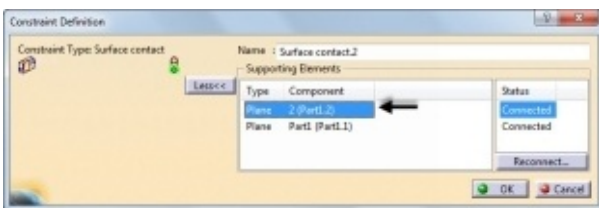
You can also redefine the existing constraints in an assembly. For example, if you want to change the faces that contact each other, then follow the steps given next.



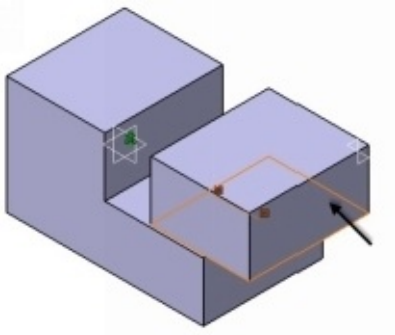
1. In the Specification Tree, expand the **Constraints** section.
2. Click the right mouse button on the **Surface contact** constraint, and then select **Surface contact.object > Definition**.



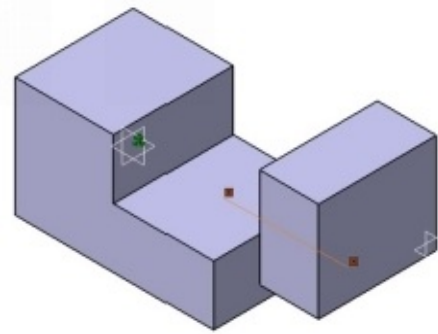
3. Click the **More** button on the **Constraint Definition** dialog.
4. On the expanded dialog, click the face to replace.



5. Click the **Reconnect** button.
6. Click on the new face.



7. Click **OK**. The contact constraint will be redefined.

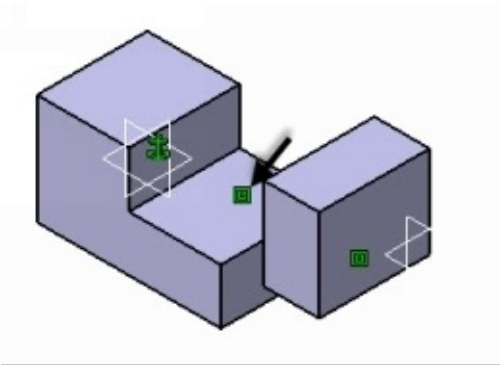




Change Constraint

You can also convert an existing constraint into another type of constraint. For example, if you want to convert the **Contact Constraint** into **Offset Constraint**, then follow the steps given next.

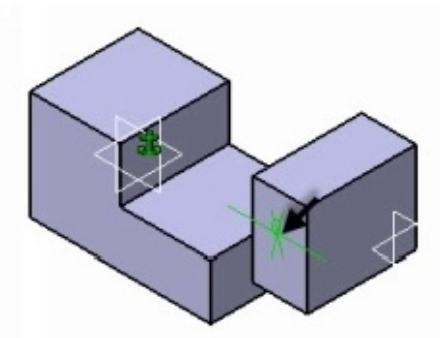
1. On the **Constraints** toolbar, click the **Change Constraint** button.
2. Click on the **Contact constraint** glyph that appears on the assembly. The **Possible Constraints** dialog appears.

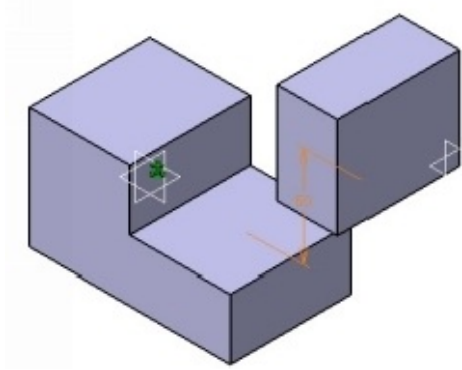
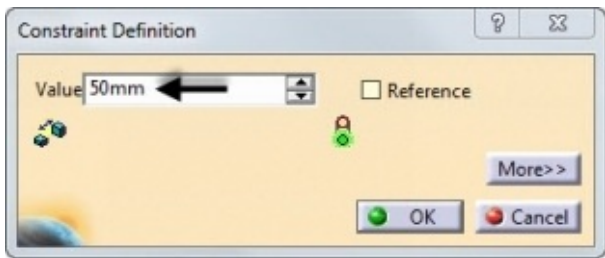


3. On the **Possible Constraints** dialog, select **Offset**, and then click **OK**. The **Contact** constraint is converted into the **Offset** constraint.



4. Now, double-click on the Offset distance value to change it.



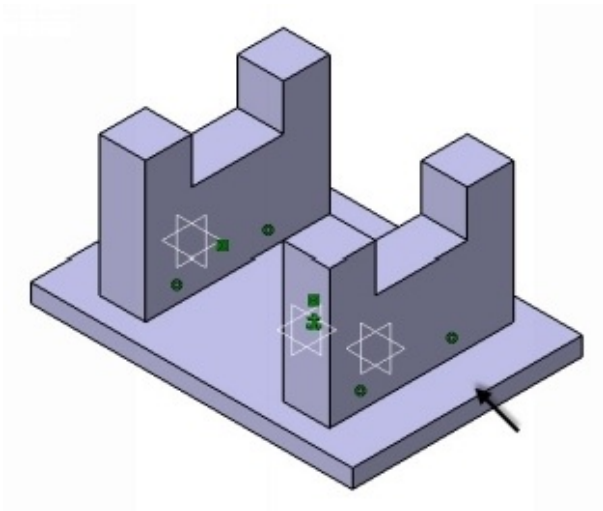




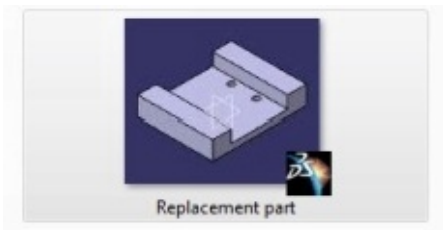
Replace Component

CATIA V5 allows you to replace any component in an assembly. To do this, follow the steps given next.

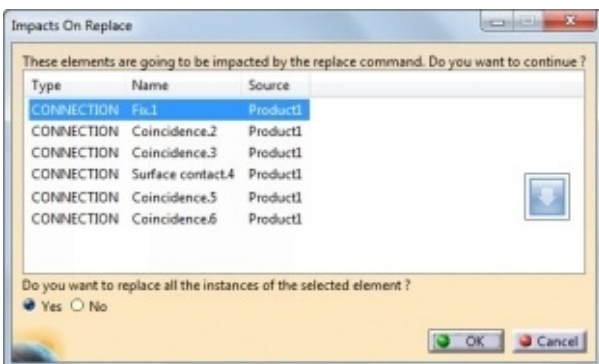
1. On the **Product Structure Tools** toolbar, click the **Replace Component** button (or) **Edit > Components > Replace Component** on the Menu bar.
2. Click on the component to replace.



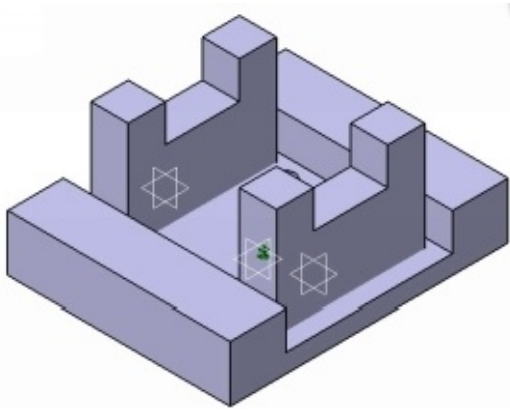
3. On the **File Selection** dialog, go to the location of the replacement part.



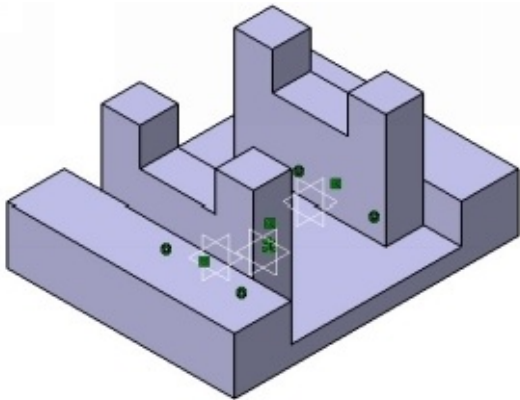
4. Select the component and click **Open**. If the new component is not similar to the old component, then the **Impacts On Replace** dialog appears. It shows the constraints that are affected.



5. Click **OK** to replace the component.

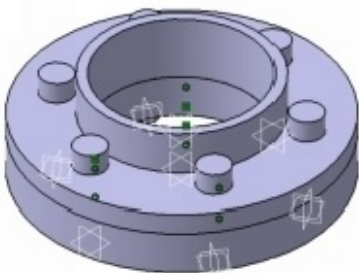


- Now, you can redefine the existing constraints or delete them and define new constraints. In this case, you can redefine the existing constraints.

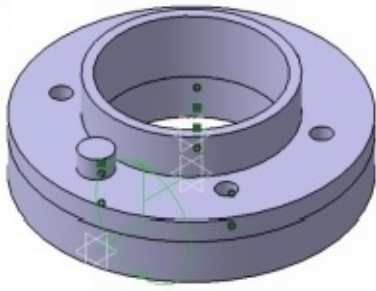


Reuse Pattern

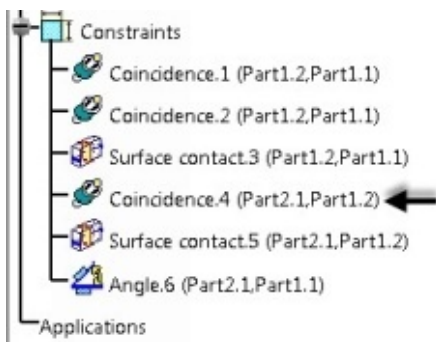
The **Reuse Pattern** command allows you to replicate individual components in an assembly. However, instead of defining layouts of rectangular or circular patterns, you can select an existing pattern as a reference. For example, in the assembly shown in figure, you can position one screw using constraints, and then use the **Reuse Pattern** command to place screws in the remaining holes.



- Position the screw in one hole using the **Coincidence Constraint**, **Contact Constraint**, and **Angle Constraint**.



2. On the **Constraints** toolbar, click the **Reuse Pattern** button (or) click **Insert > Reuse Pattern** on the Menu bar.
3. In the Specification Tree, expand the **Constraints** section and select the **Coincidence** constraint related to the screw. The pattern preview appears.

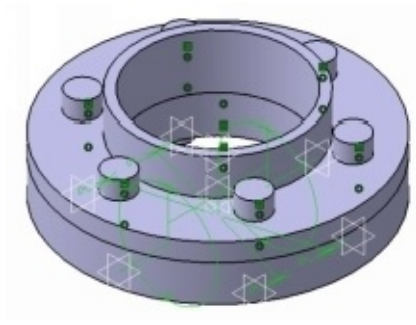


4. On the **Instantiation on a pattern** dialog, check the **Keep link with the pattern** option. This links the components with the pattern.
5. Select the **generated constraints** option to apply constraints to individual instances of the pattern.
5. Select **First instance on pattern > re-use the original component**. This creates instances at all the locations on the pattern except the original one.

If you select **create a new instance**, new instances of the component will be created at all the locations on the pattern.

If you select **cut & paste the original component**, the original component will be pasted at all the locations on the pattern.

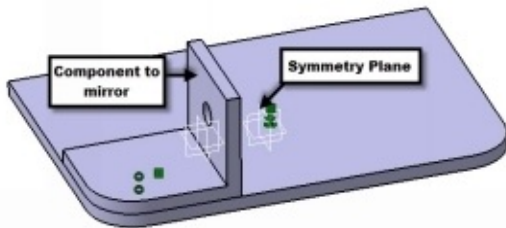
7. Click **OK**. The screw will be replicated using the existing pattern.



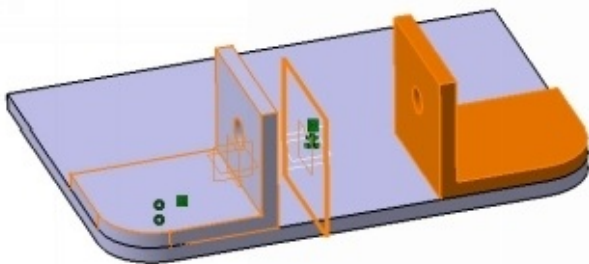
Symmetry

When designing symmetric assemblies, the **Symmetry** command will help you in saving time and capture design intent.

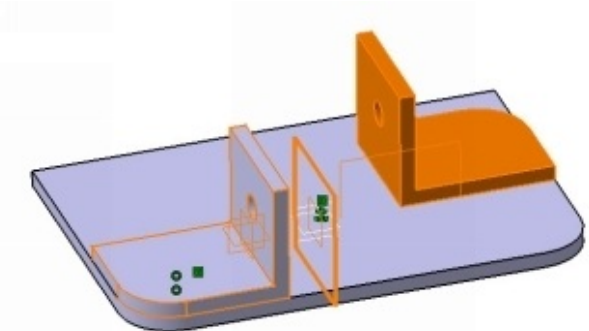
1. On the **Assembly Features** toolbar, click the **Symmetry** button (or) click **Insert > Symmetry** on the Menu bar.
2. Select the symmetry plane.
3. Click on the component to mirror.



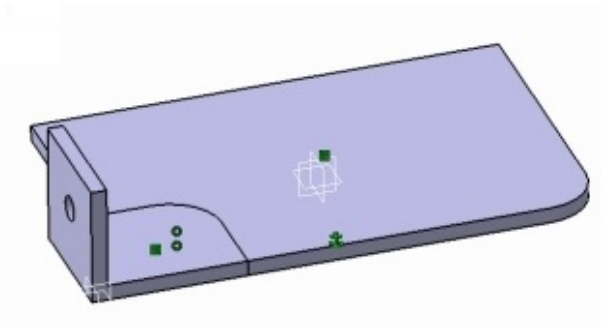
4. On the **Assembly Symmetry Wizard** dialog, select **Mirror, new component** to mirror the component about the symmetry plane.



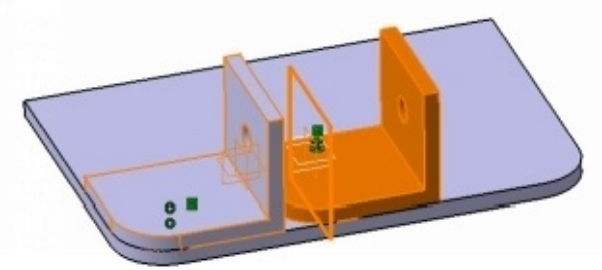
Select **Rotation, new instance** to create a new instance of the selected component and rotate it about the symmetry plane.



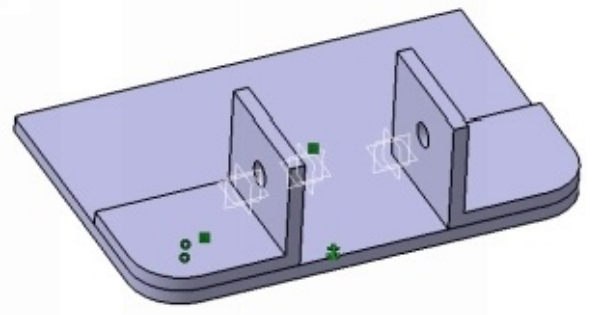
Select **Rotation, same instance** to rotate the selected instance about the symmetry plane.



Select **Translation, new instance** to create a new instance and translate it.



5. Select the required options under the **Geometry to be mirrored in new part**.
6. If you want to associate the mirrored component with original, then check the **Keep link with position** and **Keep link with geometry** options.
7. Click **Finish**, and then **Close**.



Sub-assemblies

The use of sub-assemblies has many advantages in CATIA V5. Sub-assemblies make large assemblies easier to manage. They make it easy for multiple users to collaborate on a single large assembly design. They can also affect the way you document a large assembly design in 2D drawings. For these reasons, it is important for you to create sub-assemblies in a variety of ways. The easiest way to create a sub-assembly is to insert an existing assembly into another assembly. Next, apply constraints to constrain the assembly. The process of applying constraints is also simplified. You are required to apply constraints between only one part of a sub-assembly and a part of the main assembly. In addition, you can easily hide a group of components with the help of sub-assemblies. To do this, right-click on a sub-assembly and select **Hide/Show**.

Sub-assembly



Main-assembly



Top Down Assembly Design

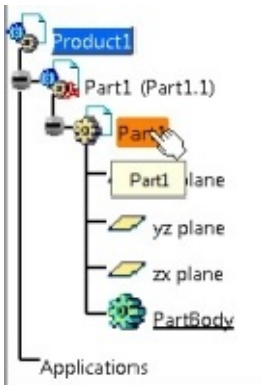
In CATIA V5, there are two methods to create an assembly. The method you are probably familiar with is to create individual components, and then insert them into an assembly. This method is known as Bottom-Up Assembly Design. The second method is called Top Down Assembly Design. In this method, you will create individual components within the Assembly Design Workbench. This allows you to design an individual part while taking into account how it will interact with other components in an assembly. There are several advantages in Top-Down Assembly Design. As you design a part within the assembly, you can be sure that it will fit properly. You can also use reference geometry from the other components.



Creating a New Part

Top-down assembly design can be used to add new parts to an already existing assembly.

1. To create a part at the assembly level using the Top down Design, activate the **Part** command (click the **Part** button on the **Product Structure Tools** toolbar).
2. In the Specification Tree, click **Product1**. A part file is created.
3. In the Specification Tree, expand the Part1 section, and then double-click on the part name. This activates the **Part Design** workbench.



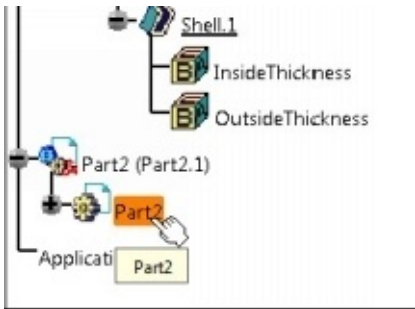
4. Now, use the part modeling commands and create the part geometry.



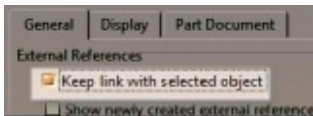
5. In the Specification Tree, double-click on **Product1** to switch to the Assembly Design workbench.
6. To create the second component, activate the **Part** command and select **Product1** in the Specification Tree. The **New Part: Origin Point** message appears.
7. Click **Yes** to define a new origin for the component (or) click **No** to use the assembly origin for the component.
8. If you click **Yes**, then you have to select a point or component to define the origin.



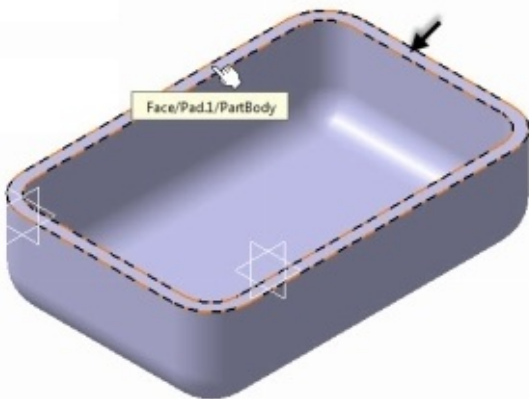
9). In the Specification Tree, expand Part2 and double-click on **Part2**.



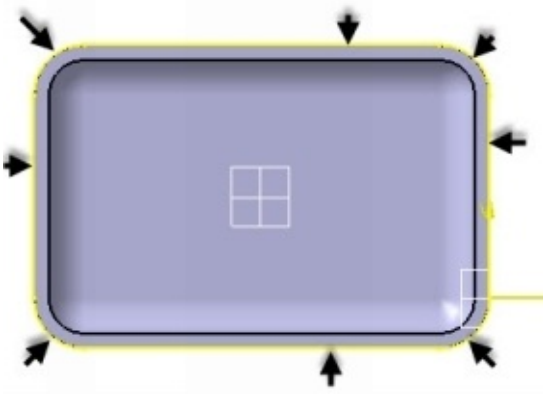
0. On the Menu bar, click **Tools > Options** to open the **Options** dialog.
1. On the **Options** dialog, click **Infrastructure > Part Infrastructure**.
2. On the **General** tab, check the **Keep Link with selected object**. Click **OK**.



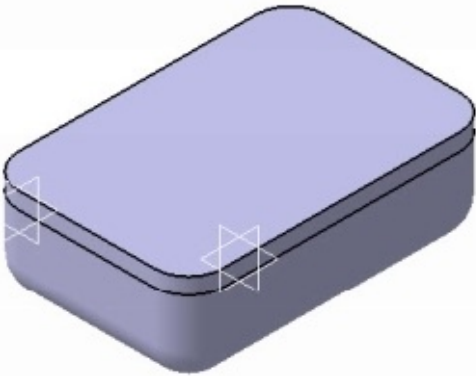
3. Now, you can create the part by using the faces and edges of the first part as reference. For example, activate the **Sketch** command and select the top face of the first part.



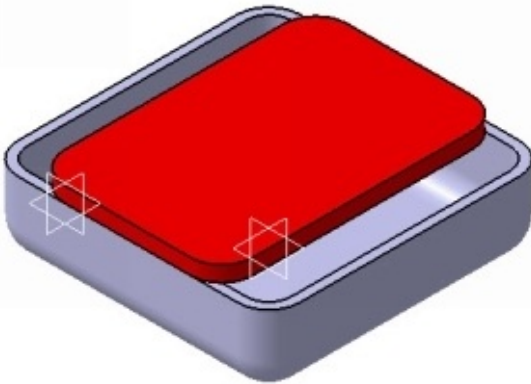
4. Activate the **Project 3D Elements** command and project the outer edges of the first component.



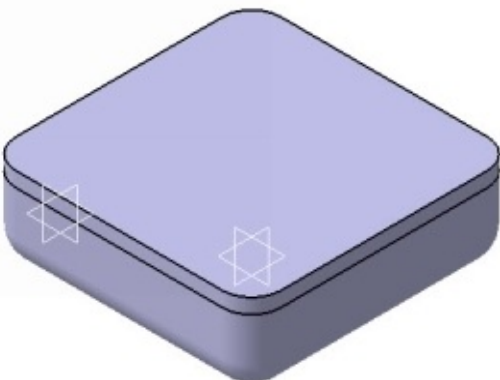
5. Use the sketch to create a *Pad* feature.



6. Activate the first component and modify the model.



7. Return to the **Assembly Design** workbench to see that the second component is update, automatically.



8. To save the assembly and its components, click **File > Save Management** on the Menu

bar.

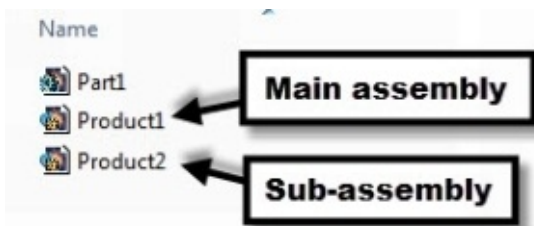
9. On the **Save Management** dialog, click on the Product, and then click the **Save as** button.
0. Define the location and file name of the product, and then click **Save**.
1. Likewise, save the components of the assembly.
2. Click **OK**.



Creating a Product

The **Product** command creates a new assembly inside the main assembly. The assembly and its parts will be saved as separate files.

1. On the **Product Structure Tools** toolbar, click the **Product** button (or) click **Insert > New Product** on the Menu bar.
2. In the Specification tree, click **Product1**. A new assembly will be listed inside the main assembly.
3. Double-click on **Product2** to activate it.
4. Create individual parts of the assembly.
5. Save the assembly. You will notice that separate files are created for the main assembly, sub-assembly, and parts.

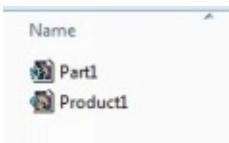




Creating a Component

The **Component** command creates a new assembly inside the main assembly. The sub-assembly will be an integral part of the main assembly.

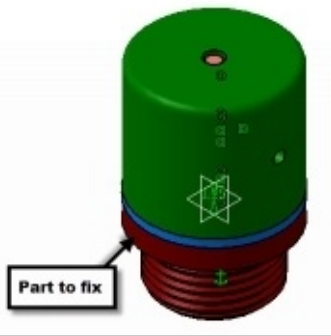
1. On the **Product Structure Tools** toolbar, click the **Component** button (or) click **Insert > New Component** on the Menu bar.
2. Click **Product1** in the Specification Tree.
3. Create individual parts inside the sub-assembly.
4. Save the assembly. You will notice that only the files of the main assembly and parts are created. The sub-assembly file is not created.



Explode

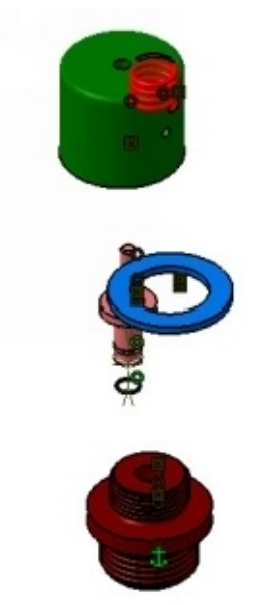
To document an assembly design properly, it is very common to create an exploded view. In an exploded view, the components of an assembly are pulled apart to show how they were assembled.

1. To create an exploded view, activate the **Explode** command (click **Explode** on the **Move** toolbar).
2. On the **Explode** dialog, click in the **Fixed product** box and select the part to be fixed at its location.

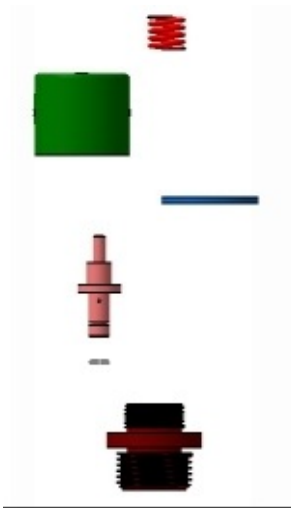


3. Select **Depth > All Levels** on the **Explode** dialog. This explodes all the parts including the one in subassemblies. If you select **Depth > First Level**, the parts in subassemblies will not be exploded.
4. Select **Type > Constrained**. This explodes the parts with respect to the constraints existing between parts.

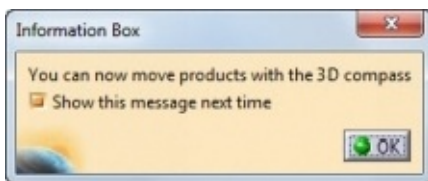
If you select **Type > 3D**, the parts will be exploded randomly in the 3D space.



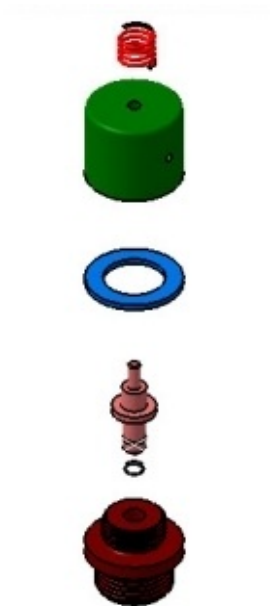
If you select **Type > 2D**, the parts will be exploded in a 2D plane parallel to the viewpoint. For example, if you set the viewpoint to front plane, the parts will be exploded in the front plane.



5. Click **Apply**. The **Information Box** appears. Click **OK** to close the box.



5. Drag the scroller on **Explode** dialog to change the explode distance.
7. Click **OK**, and then **Yes**.

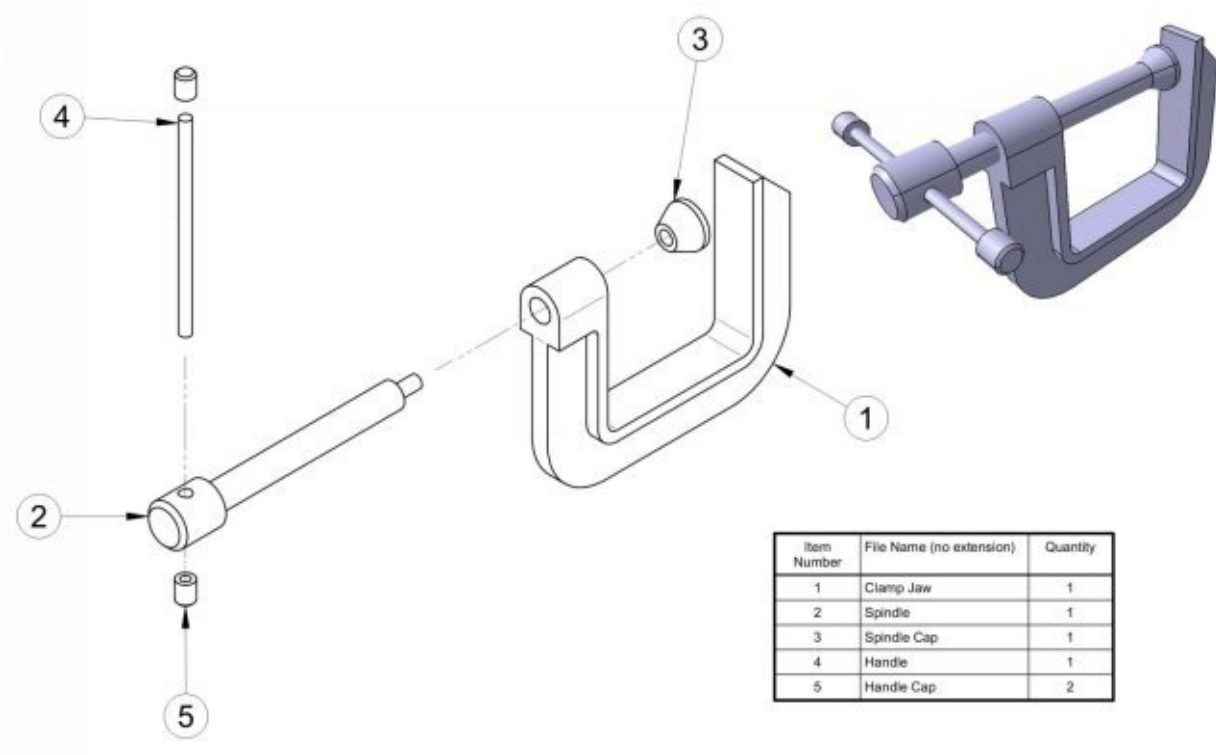


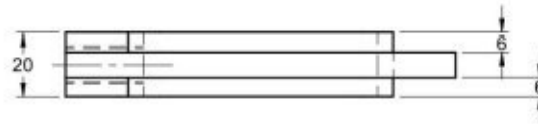
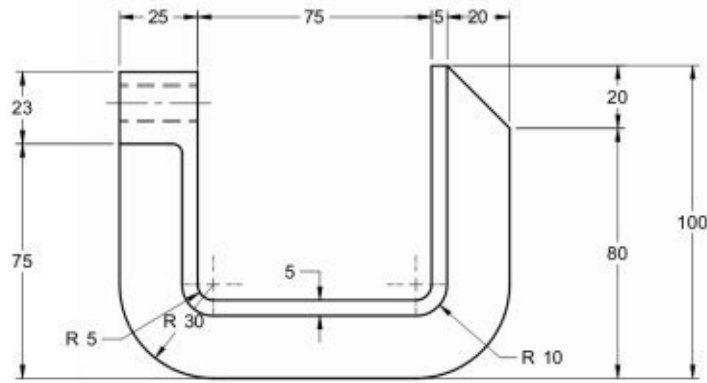
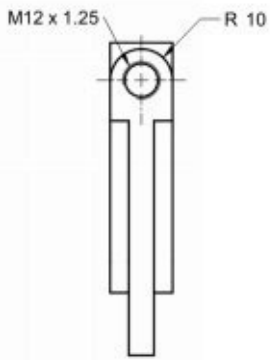
3. Click the **Update All** button to switch back to the assembled view.

Examples

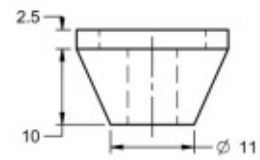
Example 1 (Bottom Up Assembly)

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



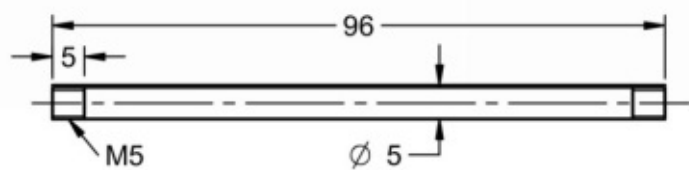
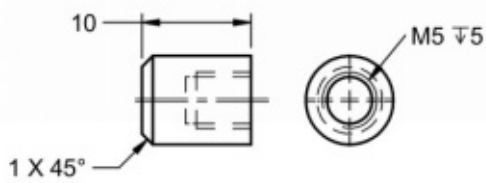


CLAMP JAW





SPINDLE

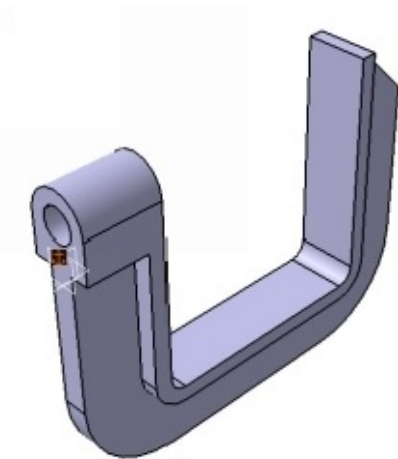
SPINDLE CAP




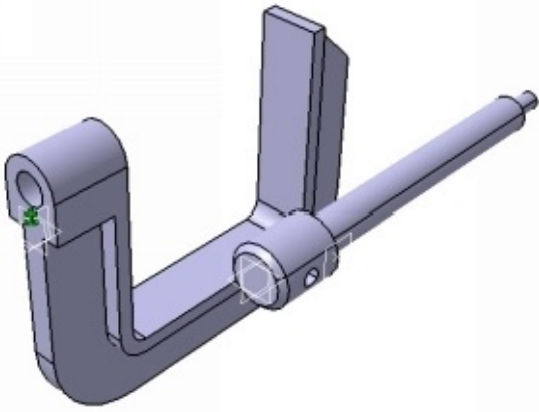
HANDLE CAP


HANDLE

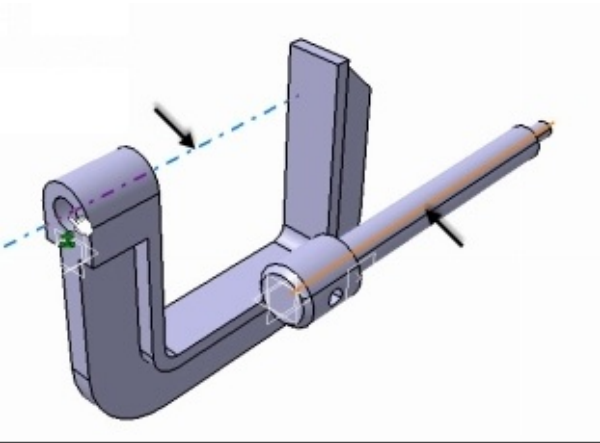
1. Start **CATIA V5-6R2015**.
2. Create and save all the components of the assembly in a single folder. Name this folder as *G-Clamp*. Close all the files.
3. On the Menu, click **Start > Mechanical Design > Assembly Design**.
4. On the **Product Structure Tools** toolbar, click the **Existing Component**  icon (or) select **Insert > Existing Component** on the menu.
5. Click **Product1** in the Specification tree.
6. On the **File Selection** dialog, go to the *G-Clamp* folder. Select *Clamp Jaw* and click **Open**.
7. On the **Constraints** toolbar, click the **Fix Component**  icon and select the Clamp Jaw. This fixes the component at the origin.




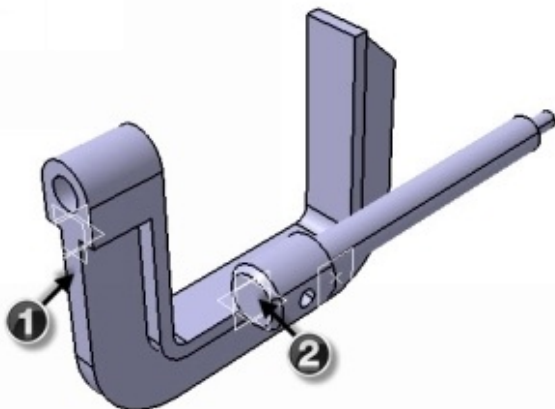
3. On the **Product Structure Tools** toolbar, click the **Existing Component With Positioning**  icon (or) select **Insert > Existing Component with Positioning** on the menu.
4. Click **Product1** in the Specification tree.
0. On the **File Selection** dialog, select *Spindle*, and then click **Open**.
 1. On the **Part number conflicts** dialog, click **Automatic rename**, and then click **OK**.
 2. On the **Smart Move** dialog, click and drag the component so that it is positioned, as shown in figure.




3. Click **OK** on the **Smart Move** dialog.
4. On the **Constraints** toolbar, click the **Coincidence Constraint**  icon (or) select **Insert > Coincidence** on the Menu.
5. On the **Assistant** message, check **Do not prompt in future** option, and then click **Close**.
6. Select the axes of the spindle and clamp jaw.

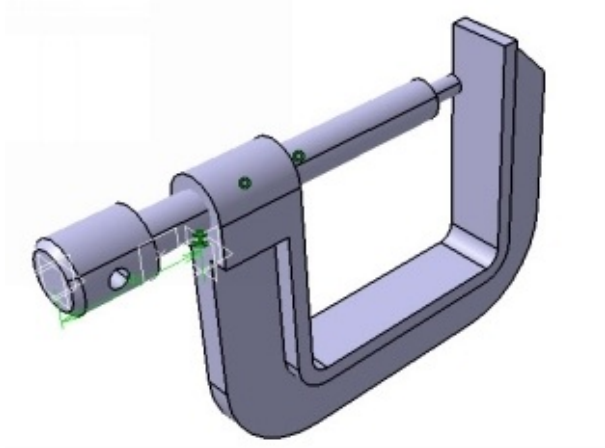


7. On the **Constraints** toolbar, click the **Offset Constraint**  icon (or) click **Inset > Offset** on the Menu.
8. Click on the front faces of the clamp jaw and that of the spindle.



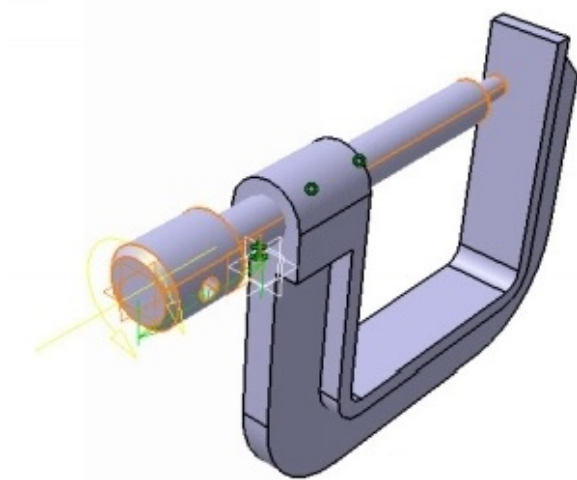
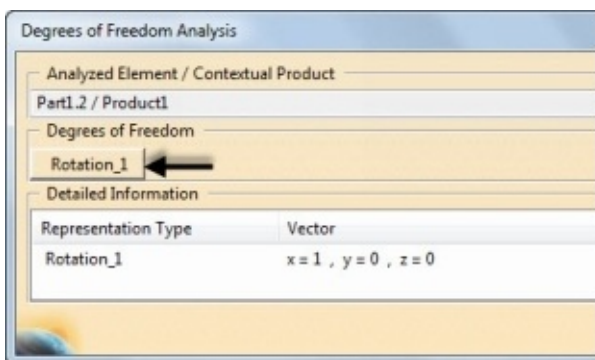
9. On the **Constraint Properties** dialog, set the **Orientation** to **Same** and type-in **40** in the **Offset** box.
0. Click **OK**.


1. On the **Update** toolbar, click the **Update All**  icon.

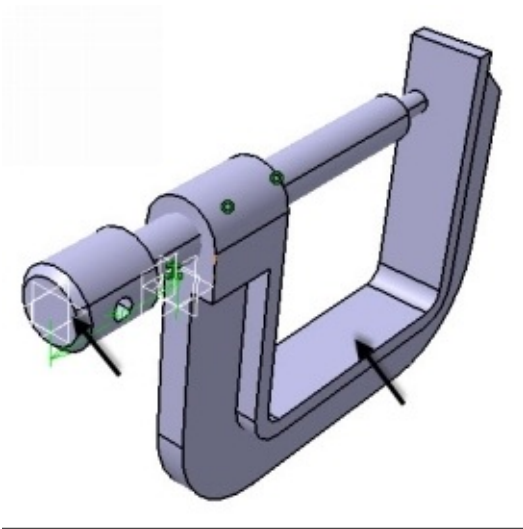


2. In the Specification tree, click the right mouse button on **Part 1.1** and select **Part1.1.1 object > Component Degrees of Freedom**.

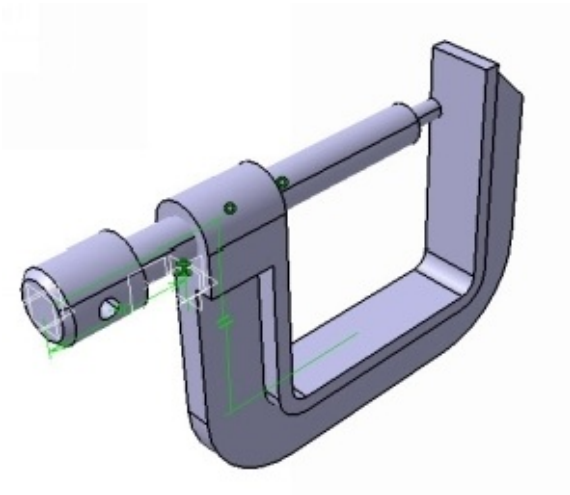
The **Degrees of Freedom Analysis** dialog appears on the screen. It shows **Rotation** in Degrees of Freedom section. In addition, a rotation symbol appears on the spindle.



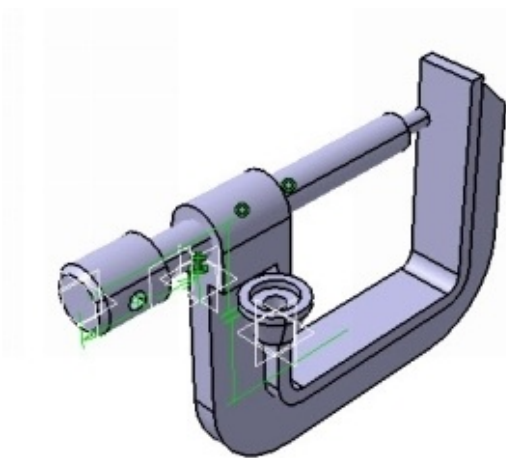
3. Close the **Degrees of Freedom Analysis** dialog.
4. On the **Constraints** toolbar, click the **Angle Constraint**  icon (or) select **Insert > Angle** on the Menu.
5. Click on the XY plane of the *Spindle* and bottom flat face of the *Clamp Jaw*.



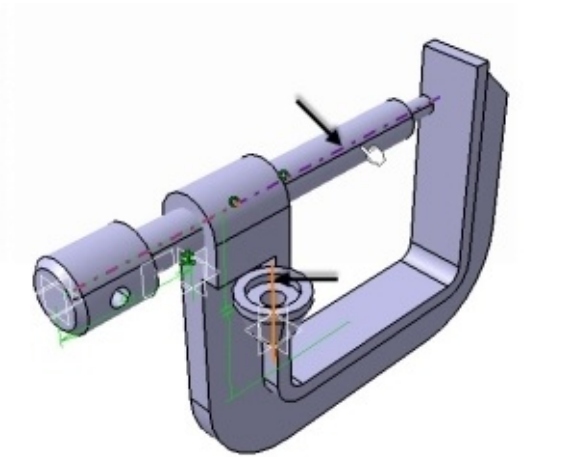
6. On the **Constraint Properties** dialog, select the **Parallelism** option and set the **Orientation** to **Same**.
7. Click **OK** to apply the parallel constraint.




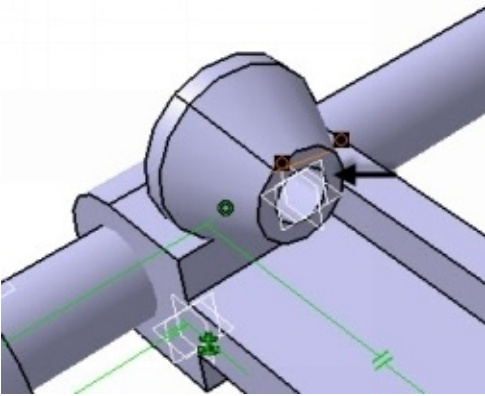
8. Now, check the degrees of freedom of the spindle. You will notice that it is fully constrained.
9. Insert the *Spindle Cap* into the assembly window.



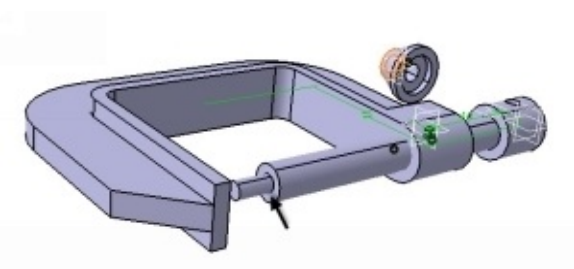
0. Activate the **Coincidence Constraint** command and click on the axes of the spindle and spindle cap.



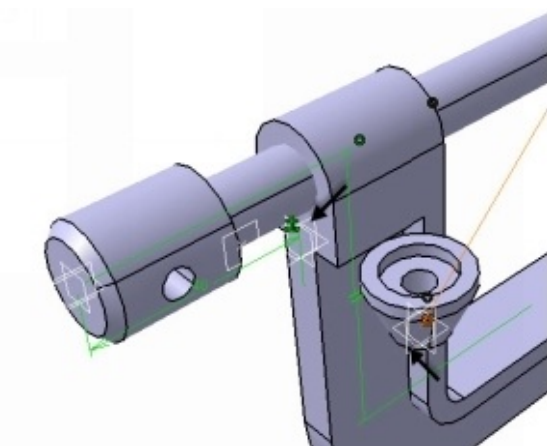
1. On the **Constraints** toolbar, click the **Contact Constraint**  icon (or) select **Insert > Contact** on the Menu.
2. Click on the bottom face of the spindle cap.




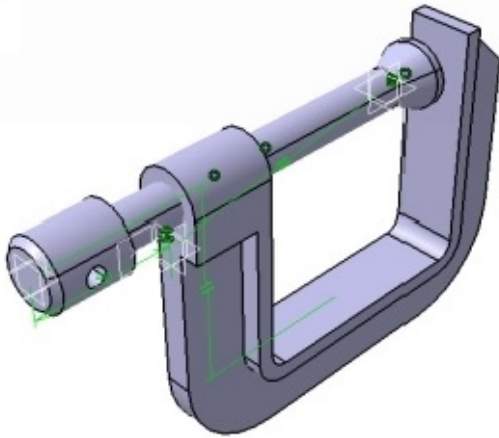
3. Rotate the assembly and click on the face, as shown below.



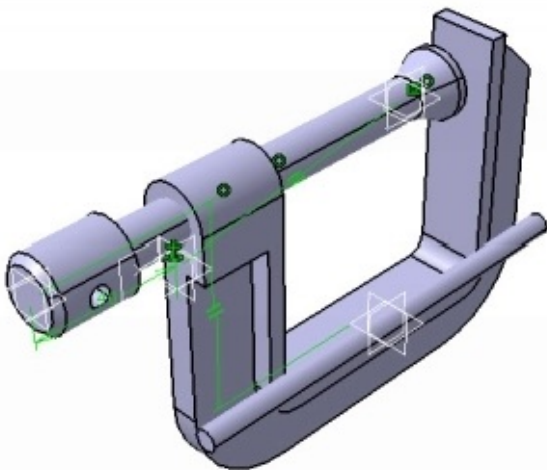
4. Activate the **Angle Constraint** command.
5. Select the zx plane of the spindle cap and that of the spindle.



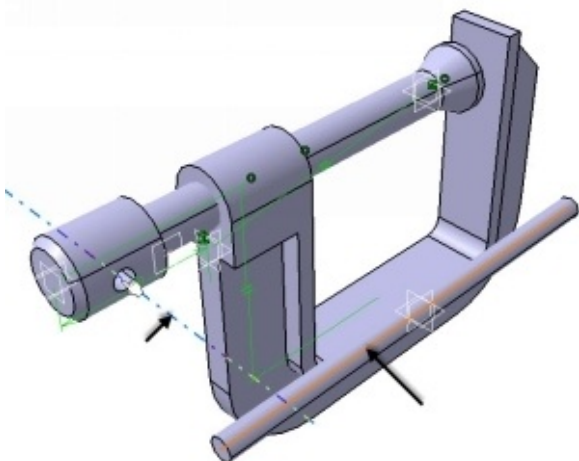
6. On the **Constraint Properties** dialog, select the **Parallelism** option and set the **Orientation** to **Same**.
7. Click **OK** to apply the parallel constraint.
8. On the **Update** toolbar, click the **Update All**  icon.



9. Insert the *Handle* into the assembly.

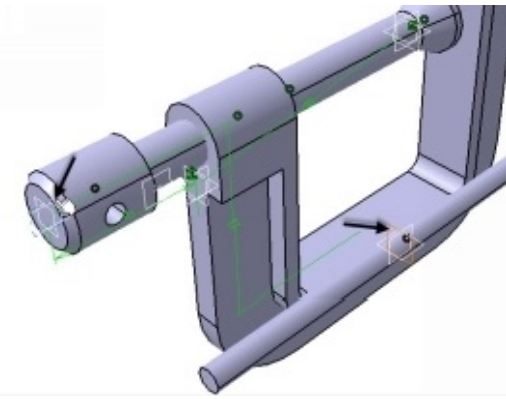


0. Activate the **Coincidence Constraint** command and select the axes of the handle and the hole on the spindle.




1. Activate the **Offset Constraint** command.

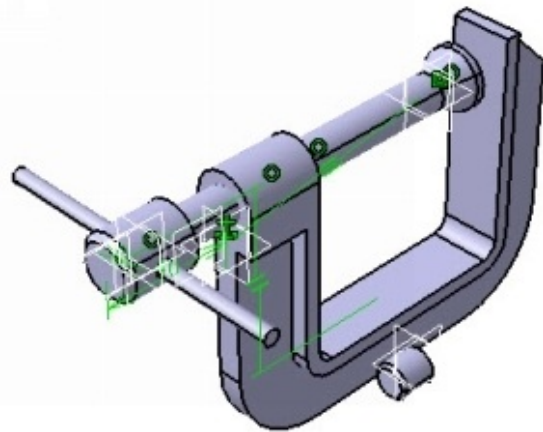
2. Click on the yz plane of the handle and zx plane of the spindle.



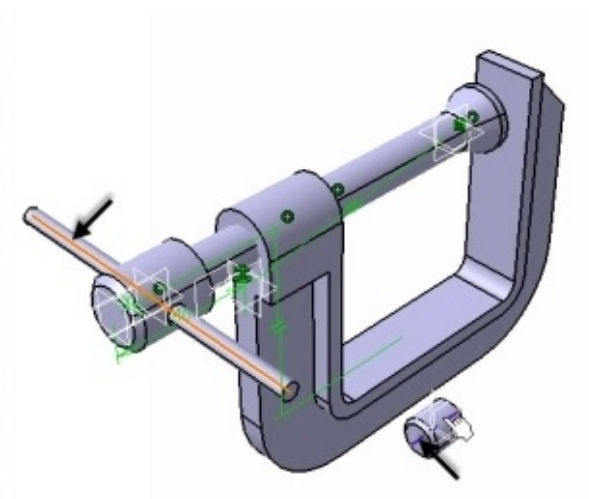
3. On the **Constraint Properties** dialog, type-in **0** in the **Offset** box and click **OK**.

4. On the **Update** toolbar, click the **Update All**  icon.

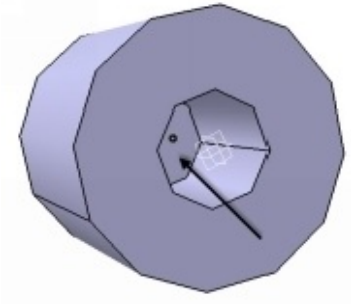
5. Insert the *Handle cap* into the Assembly window.



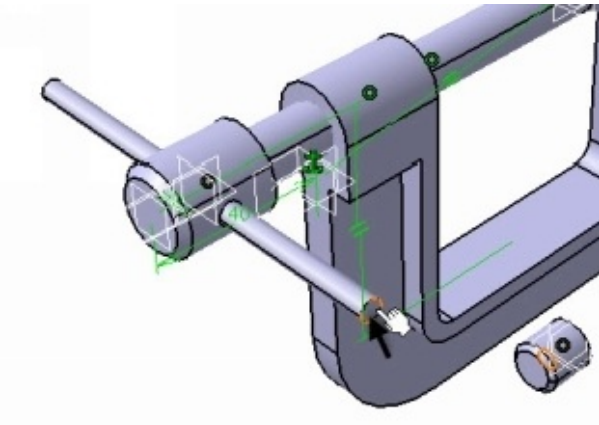
6. Activate the **Coincidence Constraint** command, and then select the axes of the handle and handle cap.



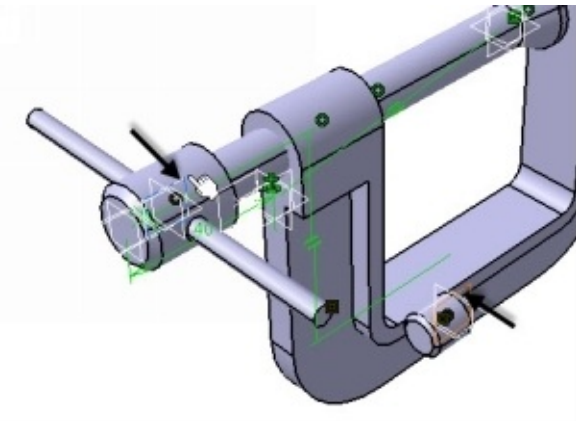
7. Activate the **Contact Constraint** command and click on the innermost flat face of the handle cap.



8. Click on the end face of the handle.




9. Activate the **Angle Constraint** command, and then select the xy planes of the handle and handle cap.

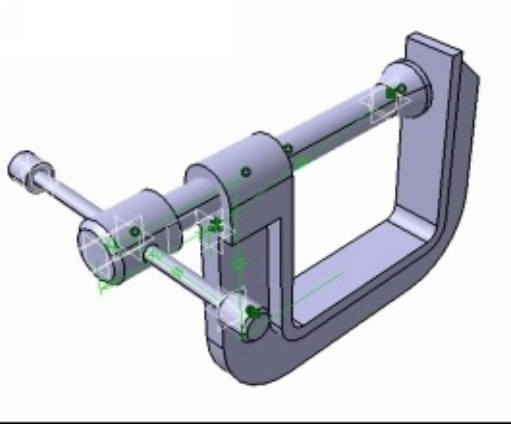
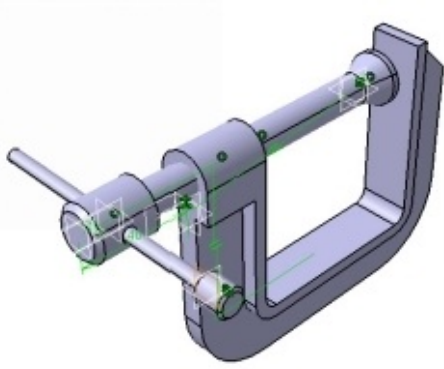


0. On the **Constraint Properties** dialog, select the **Parallelism** option and set the **Orientation** to **Same**.

1. Click **OK** to apply the parallel constraint.

2. On the **Update** toolbar, click the **Update All**  icon.

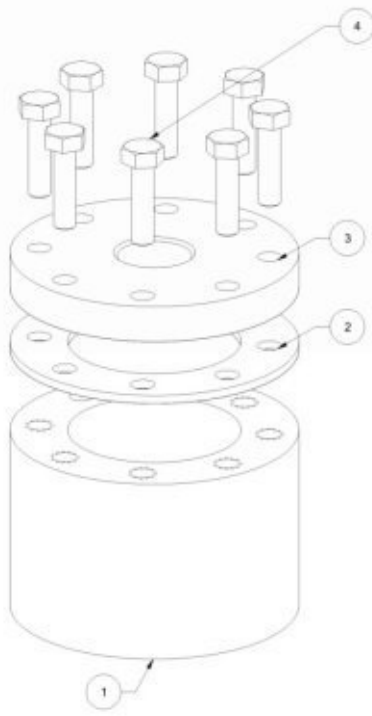
3. Likewise, assemble another instance of the handle cap.



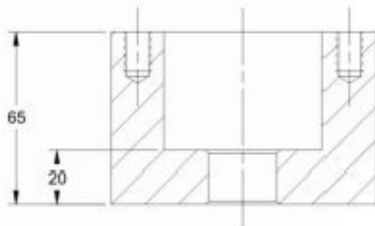
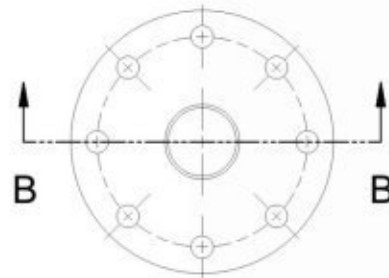
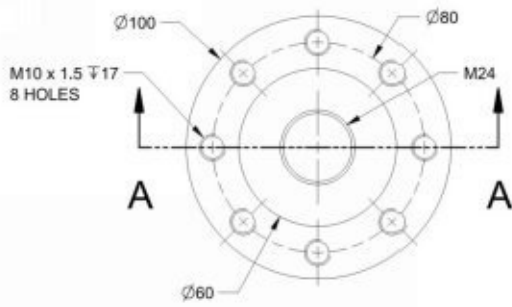
4. Save and close the assembly.

Example 2 (Top Down Assembly)

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

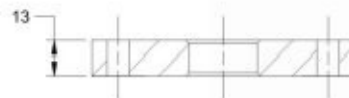


4	HEX BOLT M10 x 1.5 x 30	8
3	COVER PLATE	1
2	GASKET	1
1	CYLINDER BASE	1
PC NO	PART NAME	QTY



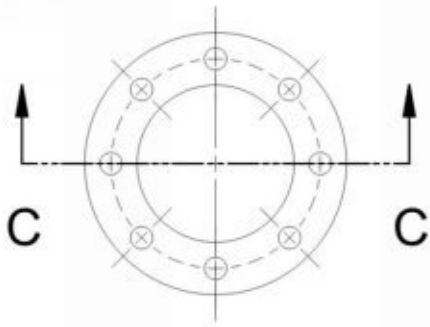
SECTION A-A

Cylinder Base



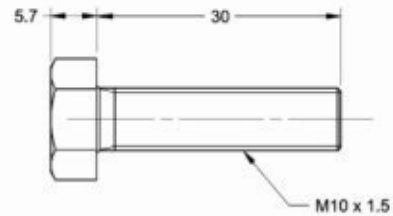
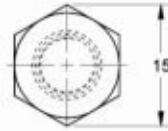
SECTION B-B

Cover Plate




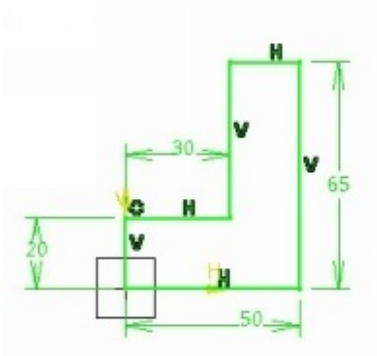
SECTION C-C

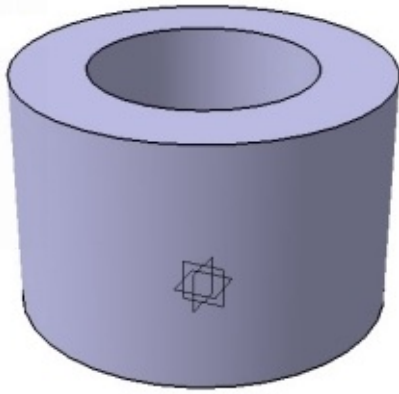
Gasket



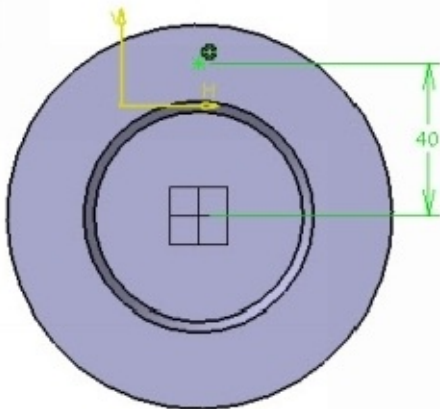
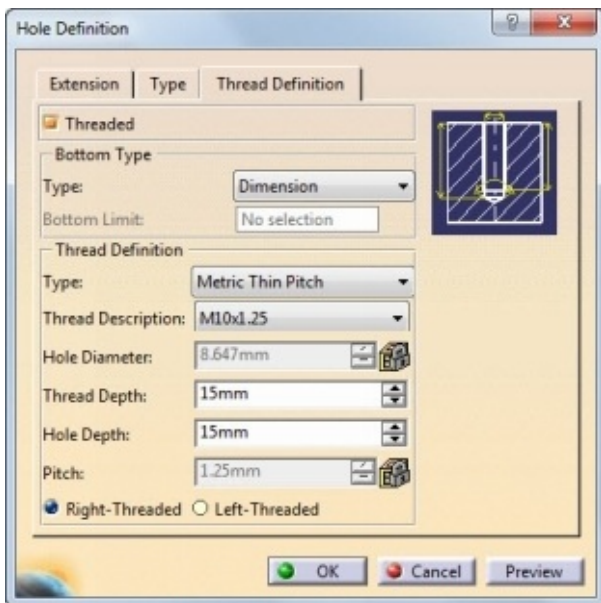
Screw

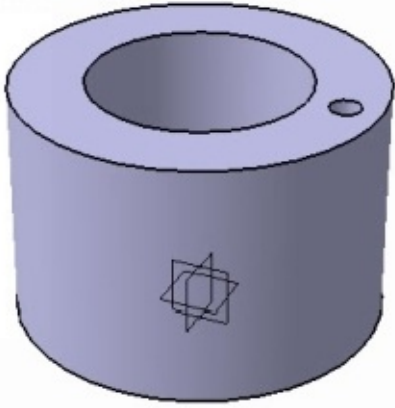
1. Start **CATIA V5-6R2015**.
2. On the **Standard** toolbar, click the **New** button to open the **New** dialog.
3. On the **New** dialog, select **List of Types > Product** and click **OK**.
4. On the **Product Structure Tools** toolbar, click the **Part**  button.
5. Select **Product 1** from the Specification Tree. This creates a new part file inside the assembly.
6. In the Specification Tree, expand **Product 1 > Part 1(Part 1.1) > Part 1**.
7. Double-click on Part 1 to activate the part mode.
8. Draw a sketch on the zx plane and revolve it.



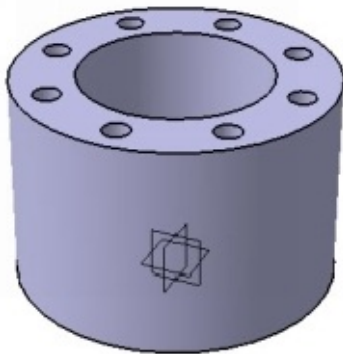



3. Create a threaded hole on the top face of the model.

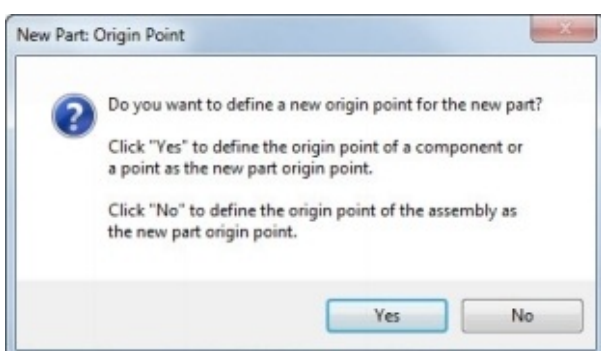





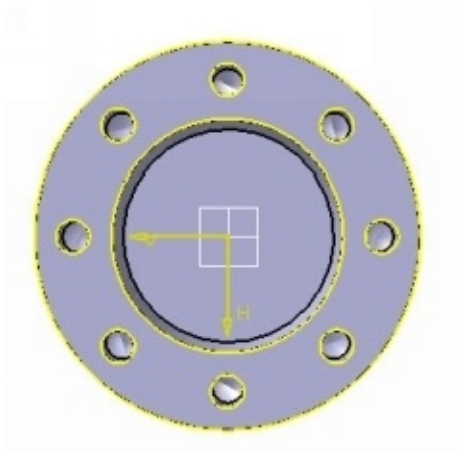
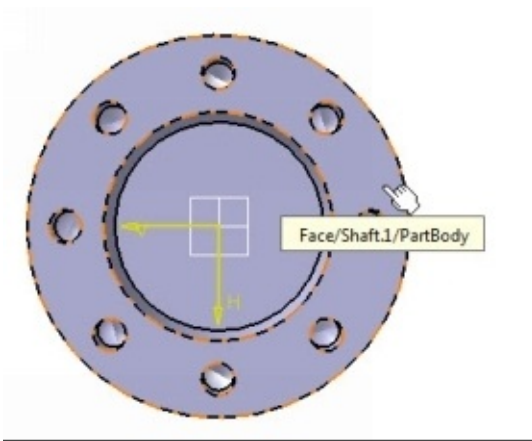
0. Create a circular pattern of the hole.



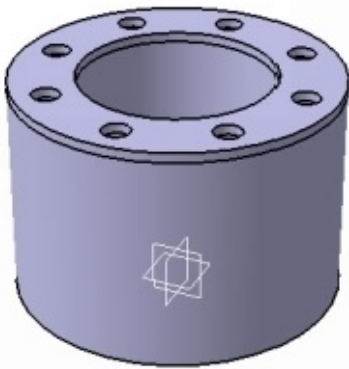
1. In the Specification Tree, double-click on **Product 1** to switch back to the Assembly mode.
2. Activate the **Part**  command (On the **Product Structure Tools** toolbar, click the **Part** button) and select **Product1** from the Specification Tree. The **New Part: Origin Point** alert message pops up on the screen.



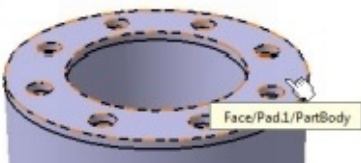
3. Click **Yes**.
4. In the Specification Tree, expand the **Part 2 (Part 2.1)** item and double-click on **Part 2**. This activates the Part mode.
5. Activate the **Sketch** command and click on the top face of the model.
6. On the **3D Geometry** toolbar, click the **Project 3D Elements**  button.
7. Click on the top face of the model, and then click **OK** on the **Projection** dialog. This projects all the edges of the top face onto the sketch plane.



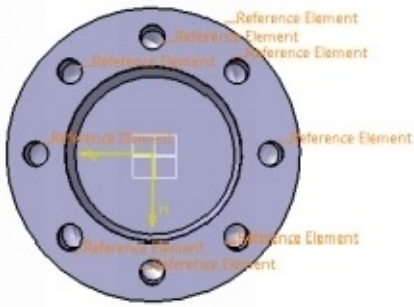
8. Exit the Sketcher workbench.
9. Extrude the sketch up to 3 mm length.



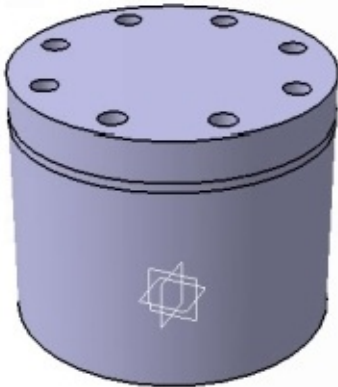
0. In the Specification Tree, double-click on **Product1** to activate the assembly mode.
1. Activate the **Part** command and create a new part file inside the assembly.
2. In the Specification Tree, double-click on Part3 to activate the part mode.
3. Start a sketch on the top face of the gasket.



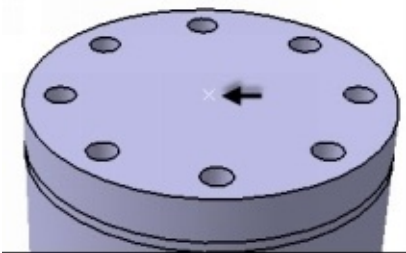
4. Project the outer circular edge and the edges of the holes.



5. Exit the sketch and extrude it up to 13 mm length.



6. Place a point at the center of the top face of the model.



7. Activate the Assembly mode by clicking **Product1** in the Specification Tree.

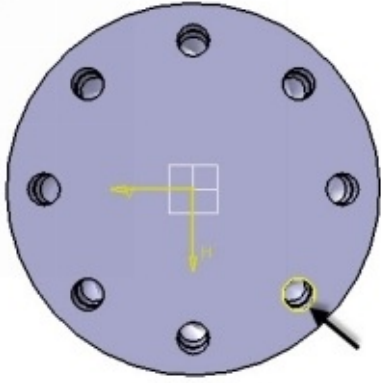
8. Activate the **Part** command create a new part inside the assembly.

9. Expand the Specification Tree and double-click on Part4.

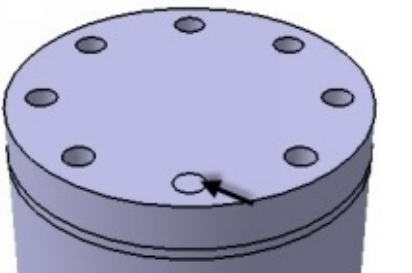
10. Start a sketch on the top face of the assembly.

11. Activate the **Project 3D Elements** command and select anyone of the circular edges of the holes.

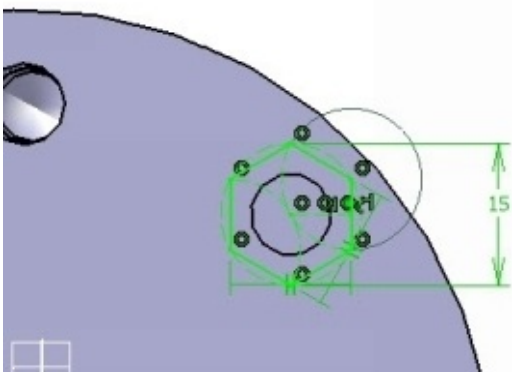
12. Click **OK** to project the selected edge.



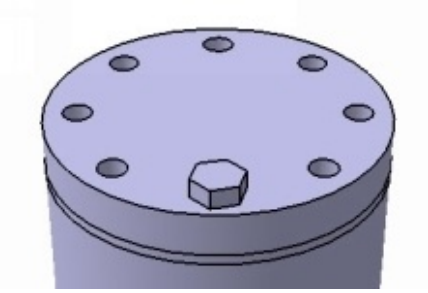
33. Exit the sketch and activate the **Pad** command.
34. Extrude the sketch up to 30 mm length in the downward direction.



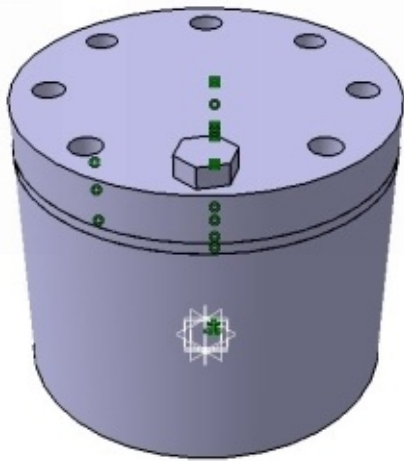
35. Start a sketch on the top face of the extrude feature.
36. On the **Profile** toolbar, click **Predefined Profile drop-down > Hexagon**, and then draw a hexagon.
37. Press and hold the Ctrl key, and then select the circle edge of the extrude feature and center point of the hexagon.
38. Click the **Constraints Defined in Dialog** button.
39. On the **Constraint Definition** dialog, check the **Concentricity** option and click **OK**.
40. Add dimensional constraint to the hexagon.




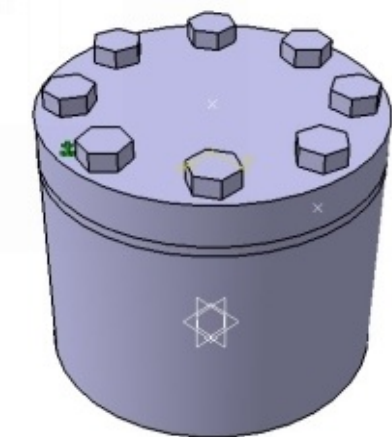
41. Exit the sketch and extrude it up to 5.7 mm length.



12. Activate the Assembly mode by double-clicking on **Product1**.
13. Fix the cylindrical base by using the **Fix Component** constraint.



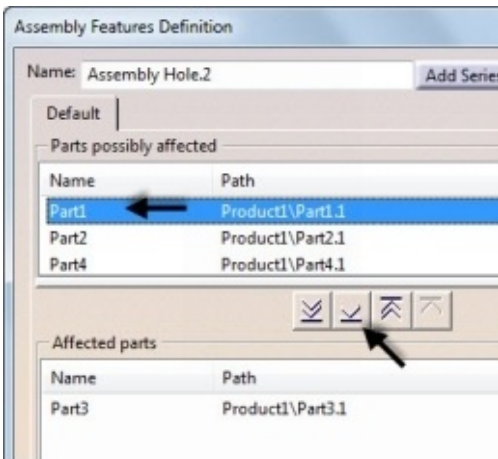
14. On the **Constraints** toolbar, click the **Reuse Pattern**  button. This brings up the **Instantiation on a pattern** dialog.
15. Select the bolt to define the component to instantiate.
16. In the Specification Tree, expand Part1 and select **CircPattern1**. The pattern and the component that has the pattern are selected.
17. Click **OK** to pattern the bolts.



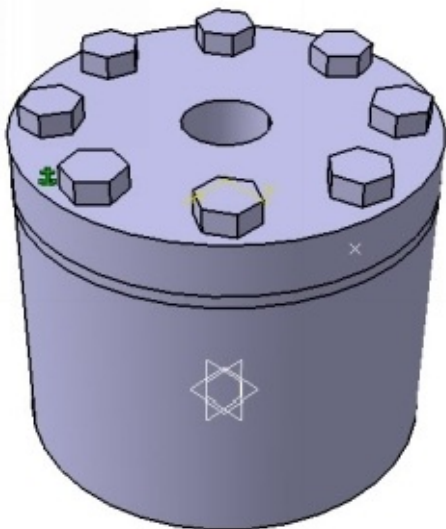
18. On the **Assembly Features** toolbar, click the **Hole** button (or) on the Menu bar, click **Insert > Assembly Features > Hole**.

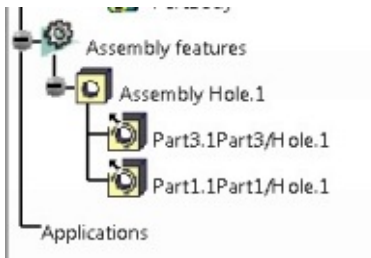


19. Select the point located on the top face of the model.
20. Click on the top face of the assembly.
21. On the **Assembly Features Definition** dialog, select the **Part1** from the **Parts possibly affected** section and click the down-arrow button.

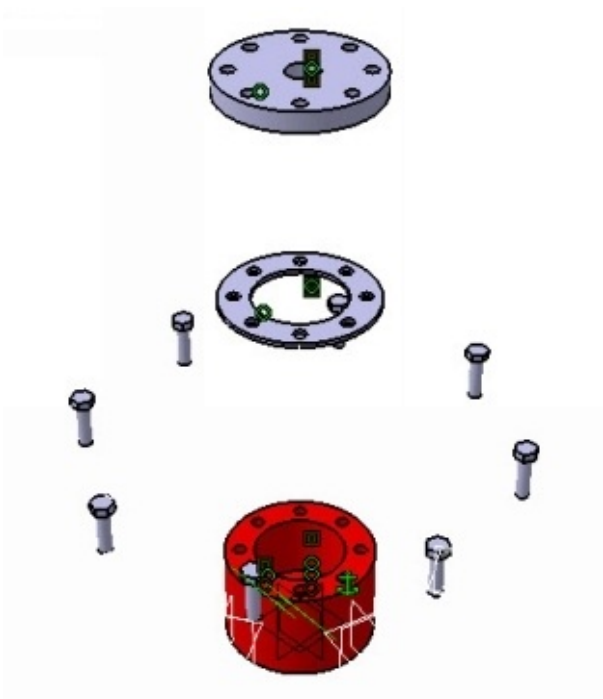


22. On the **Hole Definition** dialog, select **Extension > Blind**.
23. Type-in **81** in the **Depth** box.
24. Click the **Thread Definition** tab and check the **Threaded** option.
25. Select **Type > Metric Thick Pitch**.
26. Select **Thread Description > M24**.
27. Type-in **81** in the **Thread Depth** box.
28. Click **OK** to create the hole. You will notice a new item in the Specification Tree.





59. On the Menu bar, click **File > Save Management**.
50. Click **Save** and **OK** on the **Save Management** dialog.
51. On the **Move** toolbar, click the **Explode** button.
52. On the **Explode** dialog, click in the **Fixed product** selection box, and then select the base.
53. Click **OK** to explode the assembly. A warning message pops up showing that you are about to modify product positions. Click **Yes**.
54. Click the **Fit** button on the **View** toolbar. This fits the exploded state inside the graphics window.



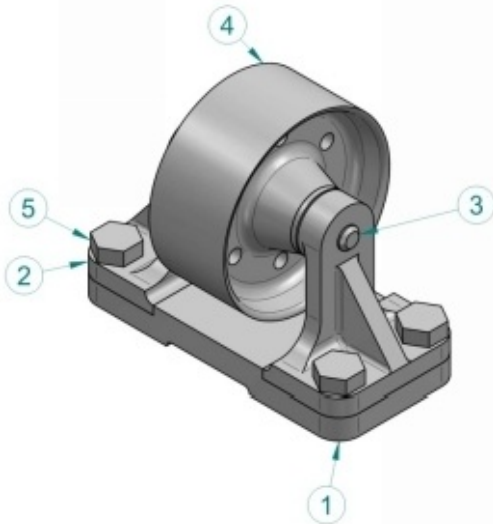
5. Click **Update All** on the **Update** toolbar.
6. Save and close the assembly.

Questions

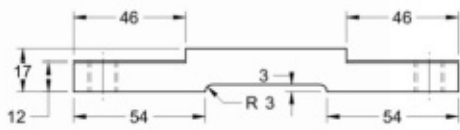
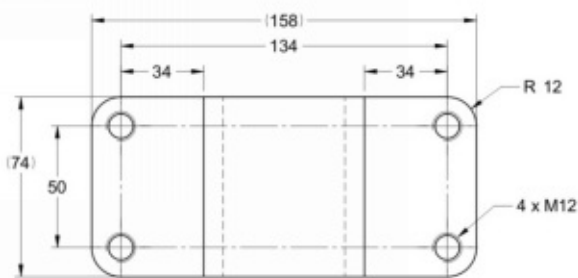
1. How do you start an assembly from an already opened part?
2. What is the use of the **Reuse Pattern** command?
3. List the advantages of Top-down assembly approach.
4. How do you create a sub-assembly in the Assembly Design Workbench?
5. Briefly explain how to edit components in an assembly.

5. What are the results that can be achieved using the **Symmetry** command?
7. How do you redefine constraints in CATIA V5?
3. What are the uses of **Angle** constraint?

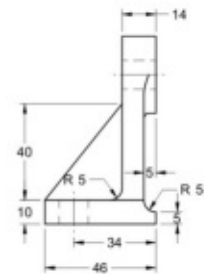
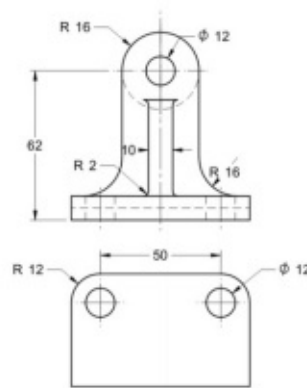
Exercise 1



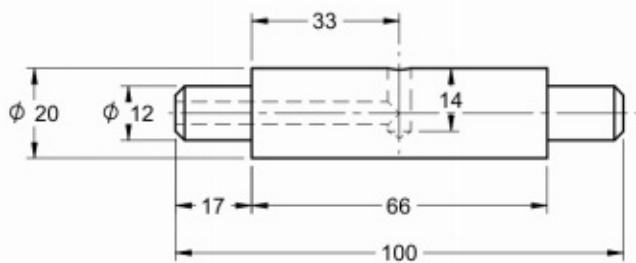
Part Number	Part Name to external	Quantity
1	Base	1
2	Bracket	2
3	Spindle	1
4	Roller-Bush assembly	1
5	Bolt	4



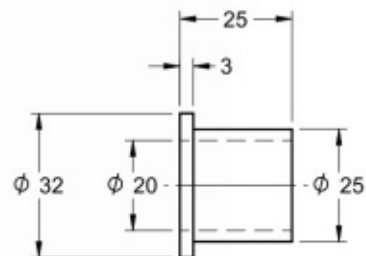
Base



Bracket



SPINDLE



BUSH

Chapter 11: Drawings

Drawings are used to document 3D models in the traditional 2D format including dimensions and other instructions useful for the manufacturing purpose. In CATIA V5, you first create 3D models and assemblies, and then use them to generate the drawing. There is a direct association between the 3D model and the drawing. When changes are made to the model, every view in the drawing will be updated. This relationship between the 3D model and the drawing makes the drawing process fast and accurate. Because of the mainstream adoption of 2D drawings of the mechanical industry, drawings are one of the three main file types you can create in CATIA V5.

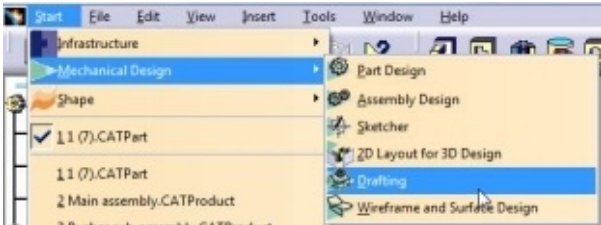
The topics covered in this chapter are:

- *Create front view*
- *Projected views*
- *Auxiliary views*
- *Sections views*
- *Detail views*
- *Break-out Section views*
- *View Breaks*
- *Parts List and Balloons*
- *Generate Dimensions*
- *Dimensions*
- *Centerlines*
- *Axis*
- *Notes*

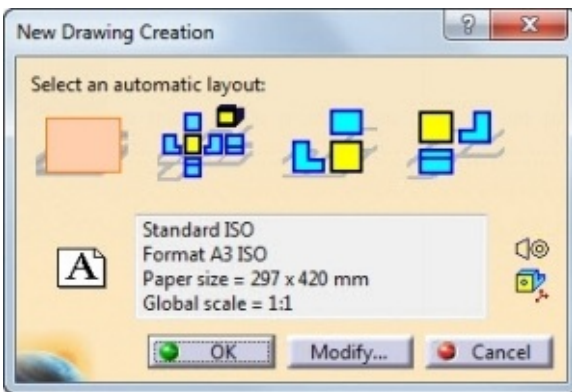
Starting a Drawing

To start a new drawing, open the part or product document, which you want to document, and then follow the steps given next.

1. On the Menu bar, click **Start > Mechanical Design > Drafting**.



2. On the **New Drawing Creation** dialog, click the **Empty Sheet** icon to the start a drawing with an empty sheet.



If you select the **All Views** icon, the drawing with all views of the part file will be created.

Likewise, use the **Front, Bottom and Right** and **Front, Top and Left** icons to start the drawing with the respective view layouts.

3. Click the **Modify** button to open the **New Drawing** dialog.
4. On the **New Drawing** dialog, set the **Standard** of the drawing.
5. Select the sheet size from the **Sheet Style** drop-down.
6. Set the drawing orientation to **Portrait** or **Landscape**.

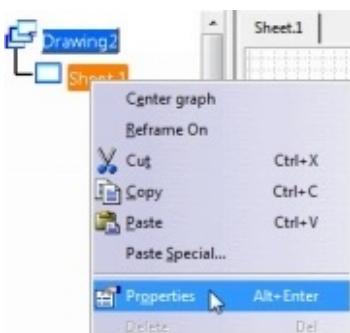


7. Click **OK** twice to start the drawing.

Modifying the Sheet Properties

Before creating the drawing, you have to check the sheet properties, and modify them as per your requirement.

1. In the Specification tree, click the right mouse button on **Sheet1** and select **Properties**.



2. On the **Properties** dialog, set the **Scale**, **Format**, and **Orientation**.
3. Set the **Projection Method** to **Third angle standard**.
4. Click **OK**.

Frame and Title Block

After modifying the sheet properties, you have to add the frame and Title Block to the sheet.

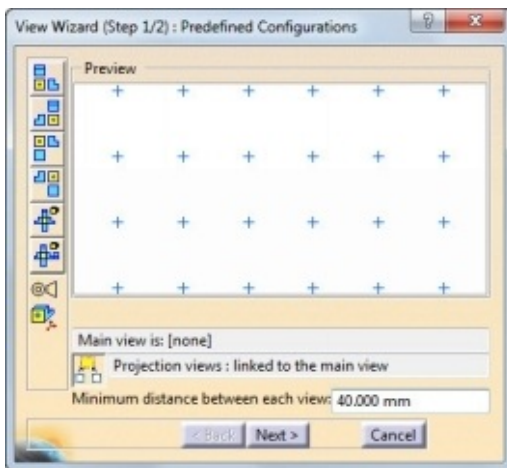
1. On the Menu bar, click **Edit > Sheet Background**. The sheet turns grey.
2. On the **Drawing** toolbar, click the **Frame and Title Block** button.


3. On the **Manage Frame and Title Block** dialog, select the required style from the **Style of Title Block** drop-down.
4. Select **Create** from the **Action** section, and then click **OK**. This adds a frame and title block to the sheet.
5. To switch back the drawing sheet, click **Edit > Working Views** on the Menu bar.

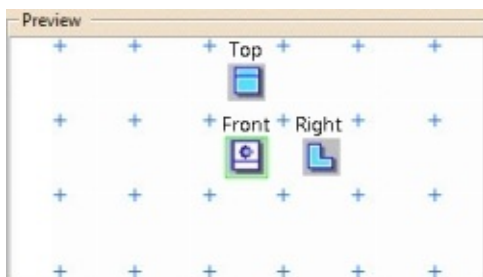
View Creation Wizard

There are different standard views of a 3D part such as front, right, top, and isometric. In CATIA V5, you can create these views using the **View Creation Wizard** command.


1. On the **Views** toolbar, click **Wizard** drop-down > **View Creation Wizard** (or) click **Insert > Wizard > Wizard** on the Menu bar. This displays the **View Wizard** dialog.

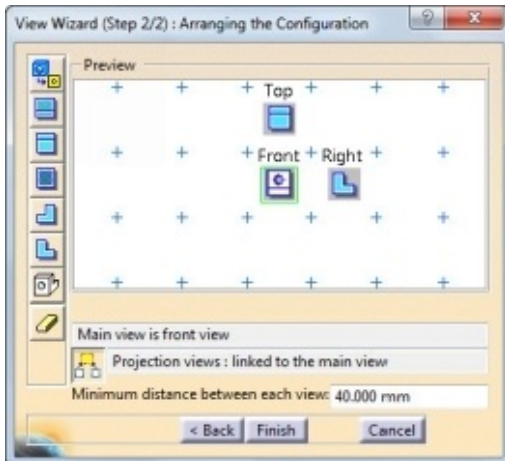


2. On the **View Wizard** dialog, click the **Configuration 1 using the 3rd angle projection method**  button to create the Top, front and right views.

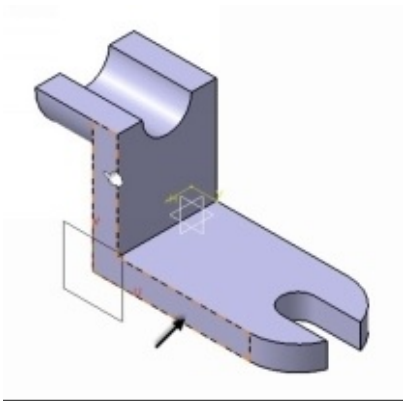


Likewise, use the other configuration buttons available at the left side on the **View Wizard** dialog to generate views as per your requirement.

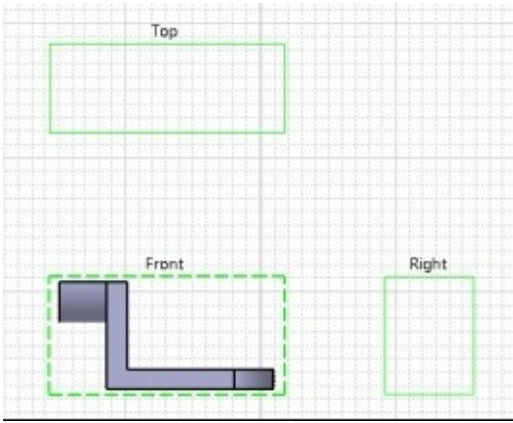
3. Ensure that the **Views Link**  button is pressed. This maintains a link between the front view and all other views. When you move the front view, the other views will also be moved.
4. Type-in the minimum distance between the views.
5. Click the **Next** button.
6. Click and drag the views in the Preview section to arrange them, if required.
7. Use the buttons available on the left side to add more views to the drawing, if required.



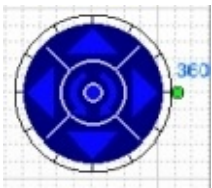
3. Click **Finish**. Now, you have to select a plane or face of the 3D model to define the front view.
4. On the Menu bar, click the **Windows > Part/Product name**. The 3D model will be opened.
5. Click on a face or plane of the model.



The previews of the views appear on the drawing sheet.

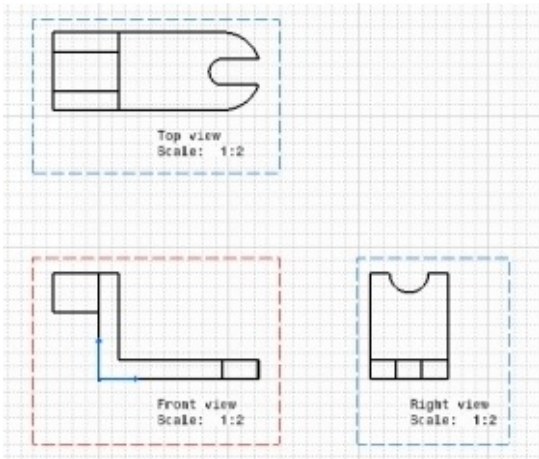


If you want to rotate the views, then click on the arrows located at the top right corner of the drawing sheet.



If you want to move the views, then press and hold the left mouse button and drag them.

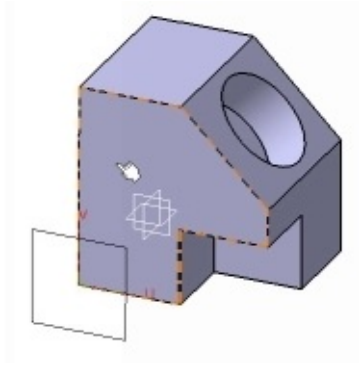
1. Click on the drawing sheet to generate the drawing views.



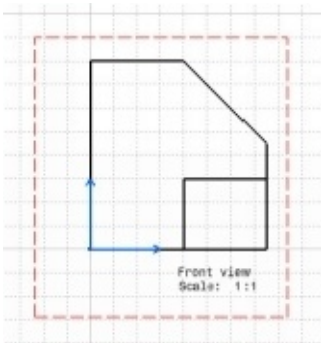
Front View

The **Front View** command allows you to create the front view of the drawing. You can later project this view to create other views.

1. On the **View** toolbar, click **Projections** drop-down > **Front View** (or) click **Insert** > **Views** > **Projections** > **Front View** on the Menu bar.
2. On the Menu bar, click **Windows** > **Part/Product name**. This takes you to the 3D model.
3. Click on a face or plane of the model to define the orientation of the front view.



4. Click on the drawing sheet to generate the drawing view.

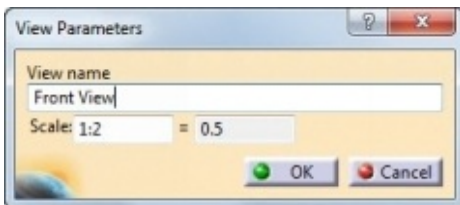




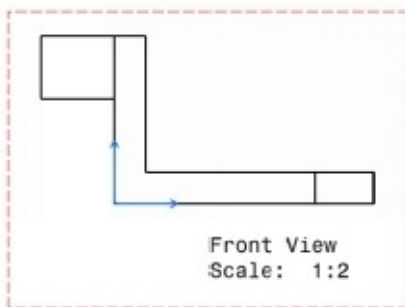
Advanced Front View

The **Advanced Front View** command creates a front view with a name and scale factor.

1. On the **View** toolbar, click **Projections** drop-down > **Advanced Front View** (or) click **Insert** > **Views** > **Projections** > **Advanced Front View** on the Menu bar.
2. On the **View Parameters** dialog, type-in values in the **View name** and **Scale** boxes.



3. Switch to the 3D model and select a face or plane to define the orientation of the front view.
4. Click on the drawing sheet to generate the view.

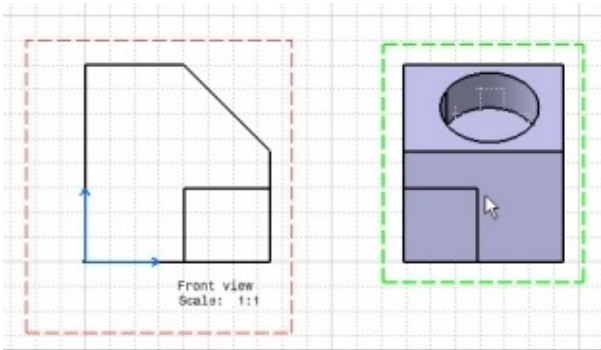




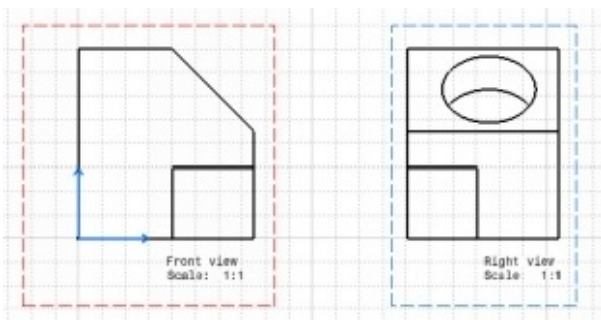
Projection View

After you have created the first view in your drawing, a projection view is one of the simplest views to create.

1. On the **Views** toolbar, click **Projections** drop-down > **Projection View** (or) click **Insert > Views > Projections > Projection** on the Menu bar.
2. After activating this command, move the pointer in the direction you wish to have the view projected.



3. Next, click on the sheet to specify the location.

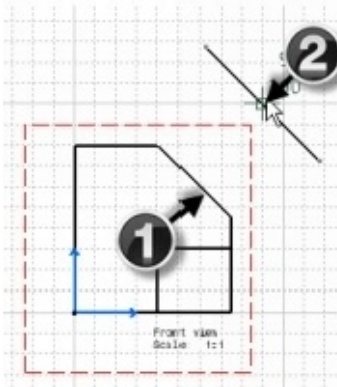




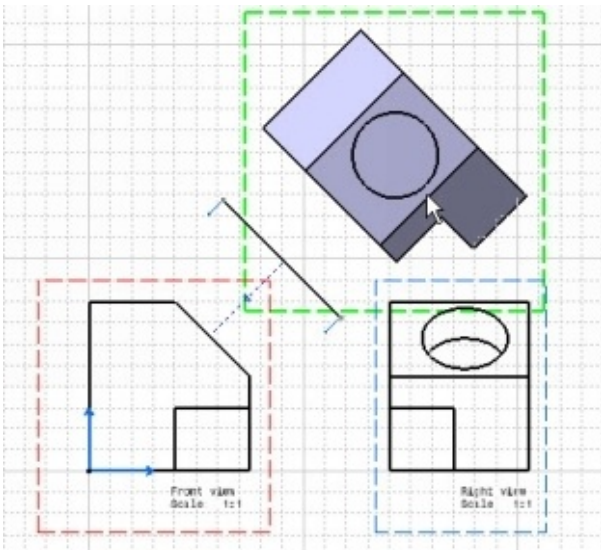
Auxiliary View

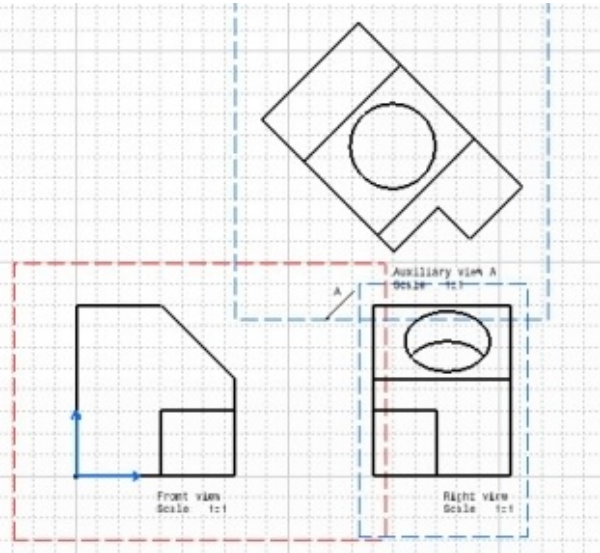
Most of the parts are represented by using orthographic views (front, top and/or side views). However, many parts have features located on inclined faces. You cannot get the true shape and size for these features by using the orthographic views. To see an accurate size and shape of the inclined features, you need to create an auxiliary view. You create an auxiliary view by projecting the part onto a plane other than horizontal, front or side planes.

1. On the **Views** toolbar, click **Projections** drop-down > **Auxiliary View** (or) click **Insert** > **Views** > **Projections** > **Auxiliary** on the Menu bar.
2. Now, click the angled edge of the model to establish the direction of the auxiliary view.
3. Move the pointer and click.



4. Drag the mouse to the desired location. Click to locate the view.

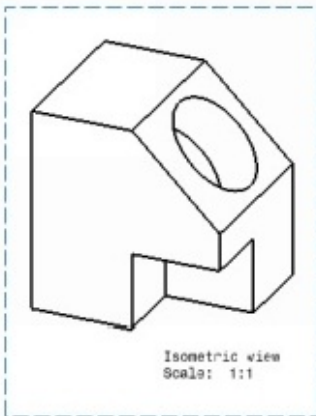




Isometric View

This command creates the Isometric View of the part/product.

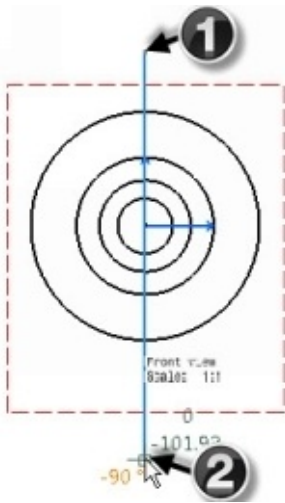
1. On the **Views** toolbar, click **Projections** drop-down > **Isometric View** (or) click **Insert** > **Views** > **Projections** > **Isometric** on the Menu bar.
2. Switch to the 3D model and select a face or plane.
3. Click on the drawing sheet.



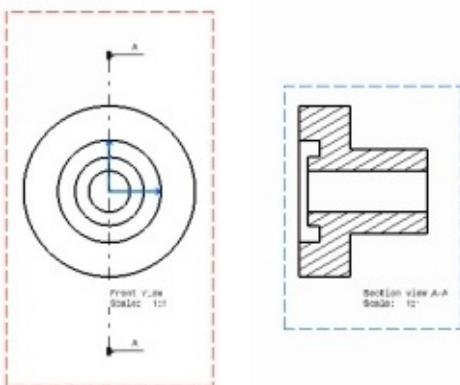
Section View

One of the common views used in 2D drawings is the section view. Creating a section view in CATIA V5 is very simple.

1. Activate the view to be sectioned. To do this, click the right mouse button on the view and select **Activate View**.
2. On the **Views** toolbar, click **Sections** drop-down > **Offset Section View** (or) click **Insert** > **Views** > **Sections** > **Offset Section View** on the Menu bar.
3. Draw a section line on the active view by selecting two points.
4. Double-click on the second point of the section line. This ends the section line.



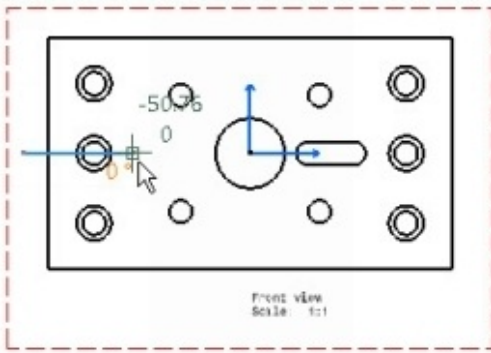
5. Move the pointer and click to locate the section view.



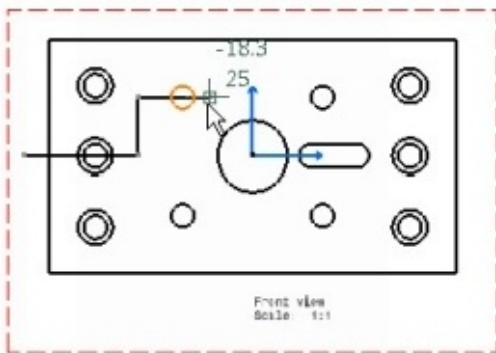
Offset Section View

If you want to create a section view by using a multi-segment section line, then activate the **Offset Section View** command.

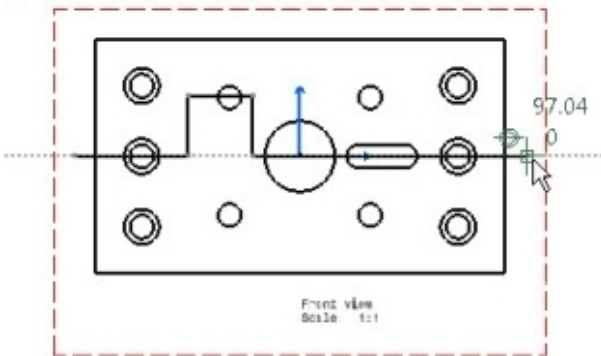
1. Click to define the first point of the section line.
2. Move the pointer and click to define the second point.



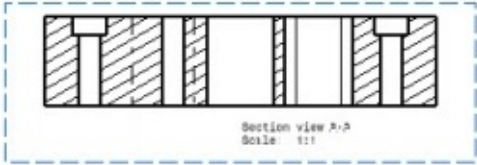
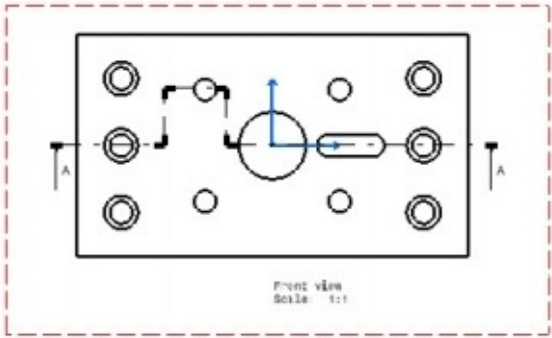
3. Move the pointer in the direction perpendicular to the section line and click. You will notice that a multi-segment line is created.



4. Likewise, create other multiple segments of the section line.
5. Double-click to end the section line.



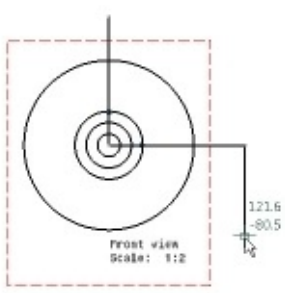
5. Move the pointer and click on either side of the section line.



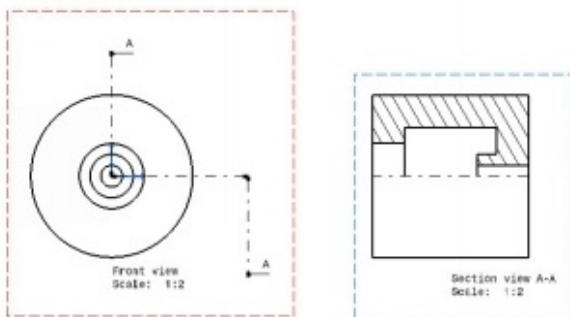
Half Section View

If you want to create a half section view, activate the **Offset Section View** command

1. Create a multi-segment section line, as shown below.



2. Double-click to end the section line.
3. Move the pointer and click to position the half section view.

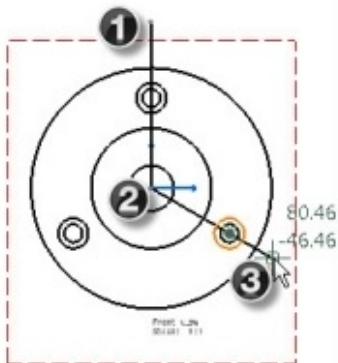




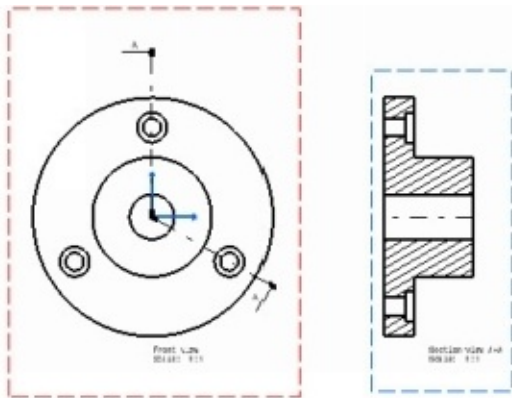
Aligned Section View

Use the **Aligned Section View** command to create a revolved section view. To create this type of section view, you need to create two section lines at an angle to each other.



1. On the **Views** toolbar, click **Sections** drop-down > **Aligned Section View** (or) click **Insert > Views > Sections > Aligned Section View** on the Menu bar.
2. Click to define the start point of the first line.
3. Move the pointer and click to define the end point of the first line.
4. Move the pointer and click to define the length and orientation of the second line.

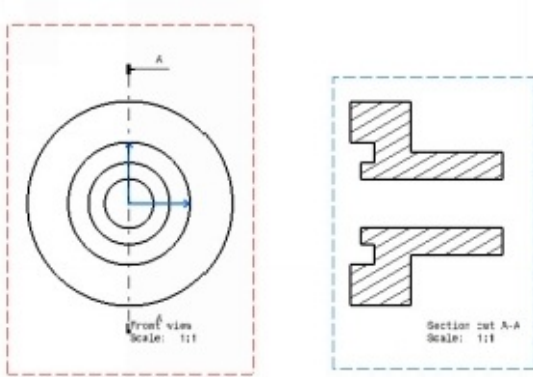


5. Double-click to end the section line.
6. Move the pointer and click to position the revolved section view.



Creating Section Cuts

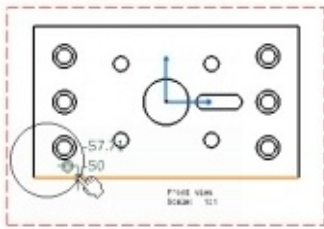
A section cut displays the surface that is exposed after sectioning. It hides the other surfaces. You can create section cuts using the **Offset Section Cut**  and **Aligned Section Cut**  commands. The procedure to create section cuts is similar to that of section views.



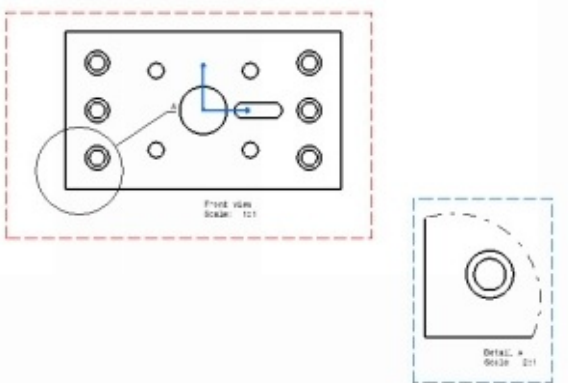
Detail View

If a drawing view contains small features that are difficult to see, a detailed view can be used to zoom in and make things clear. To create a detail view, follow the steps given next.

1. On the **Views** toolbar, click **Details** drop-down > **Detail View** (or) click **Insert** > **Views** > **Details** > **Detail** on the Menu bar. This automatically activates the circle command.
2. Draw a circle to identify the area that you wish to zoom in.



3. Once the circle is drawn, move the pointer and click to locate the view. The detail view will appear with a label.



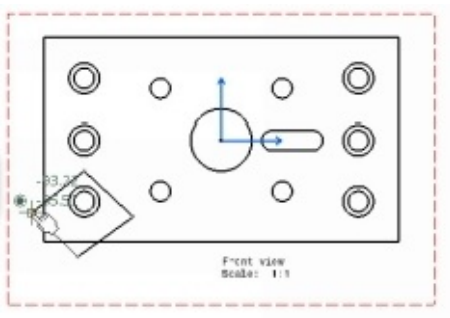
If you want to change the scale value, then click the right mouse button on the detail view and select **Properties**. On the **Properties** dialog, type-in a new value in the **Scale** box and click **OK**.



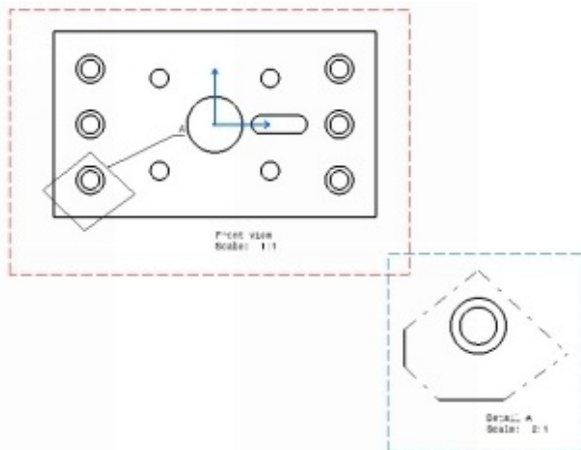
Detail View Profiles

This command creates a detail view using a user-defined profile.

1. On the **Views** toolbar, click **Details** drop-down > **Detail View Profile** (or) click **Insert > Views > Details > Sketched Detail Profile** on the Menu bar. This activates the **Profile** command.
2. Draw a closed profile enclosing the area to be detailed.



3. Move the pointer and click to locate the view.

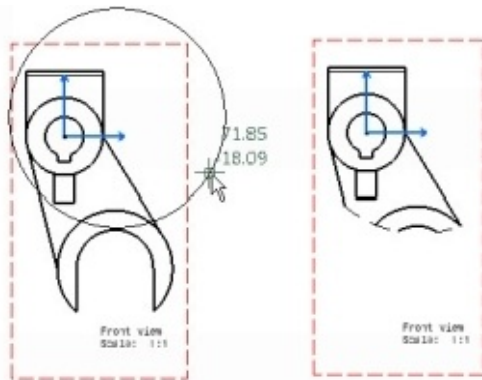




Clipping View

This command crops a view by the area defined by a circle.

1. On the **Views** toolbar, click **Clippings** drop-down > **Clipping View** (or) click **Insert** > **Views** > **Clippings** > **Clipping** on the Menu bar. This automatically activates the circle command.
2. Draw a circle on the view to be cropped.

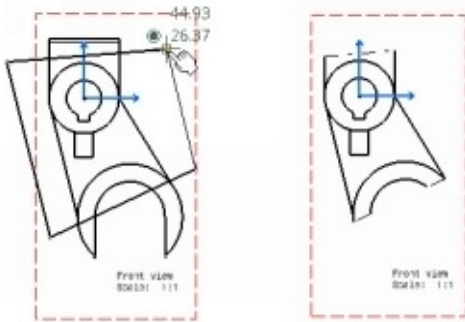




Clipping View Profile

This command crops a view by the area defined by a user-defined profile.

1. On the **Views** toolbar, click **Clippings** drop-down > **Clipping View Profile** (or) click **Insert > Views > Clippings > Sketched Clipping Profile** on the Menu bar.
2. Create a closed-profile on the view.

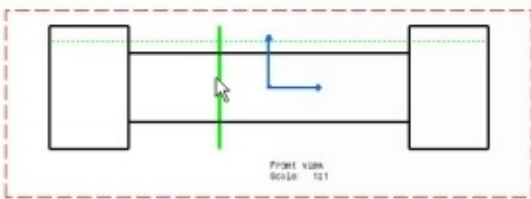




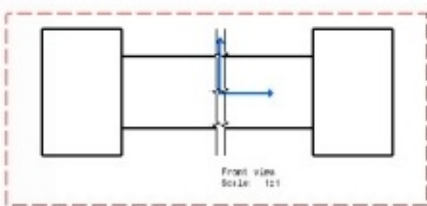
Broken View

You can add break lines to a drawing view, which is too large to fit on the drawing sheet. They break the view so that only important details are shown.

1. On the **Views** toolbar, click **Break view** drop-down > **Broken View** (or) click **Insert** > **Views** > **Break view** > **Broken View** on the Menu bar.
2. Click on the view to locate the beginning of the break. Next, you have to choose whether you want to create a vertical or horizontal break lines.
3. Select the vertical or horizontal dotted line to define the vertical or horizontal break lines.



4. Move the pointer and click again to locate the end of the break.
5. Click on the sheet to generate break lines.

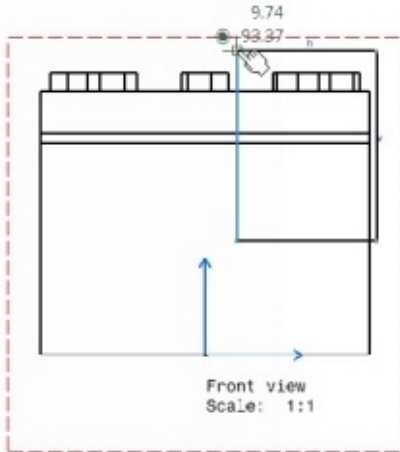


If you want to change the linetype of the break lines, then click the right mouse button on them and select **Properties**. On the **Properties** dialog, select a new linetype to be applied from the **Linetype** drop-down menu. Click **OK** to close the dialog.

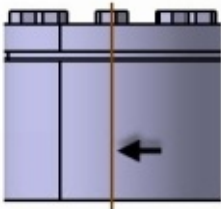
Breakout View

The **Breakout View** command alters an existing view to show the hidden portion of a part or assembly. This command is very useful to show the parts, which are hidden in an assembly view. You need to create a closed profile to breakout a view.

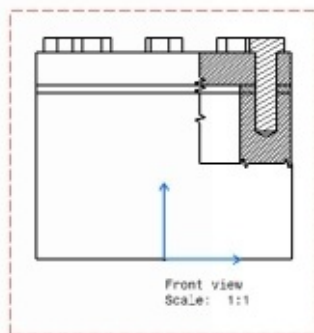
1. On the **Views** toolbar, click **Break view** drop-down > **Breakout View** (or) click **Insert** > **Views** > **Break view** > **Breakout View** on the Menu bar.
2. Draw a closed profile on the view to be broken.



3. On the 3D Viewer window, click and drag the vertical line located at the center of the view. This defines the depth of the breakout.






4. Click **OK** to close the **3D Viewer** window. This creates a breakout view.




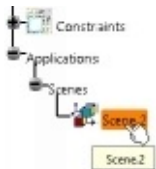
Exploded View

You can display an assembly in an exploded state as long as the assembly already has an exploded scene defined. Follow the steps given next to define the exploded scene in an assembly.

1. Open the assembly file.
2. On the **Scenes** toolbar, click the **Enhanced Scene**  button (or) click **Insert > Create Enhanced Scene** on the Menu bar.
3. Click **OK** on the **Enhanced Scene** dialog. This activates the enhanced scene mode.
4. On the **Enhanced Scene** toolbar, click the **Explode**  button, and then explode the assembly.
5. On the **Enhanced Scene** toolbar, click the **Exit Scene**  button.
5. Save the assembly file

Follow the steps given next to insert the exploded scene in a drawing.

1. Open a drawing file.
2. On the **Views** toolbar, click **Projections** drop-down > **Isometric View** .
3. Switch to assembly file.
4. In the Specification Tree, go to **Application > Scene** and select the Exploded Scene.



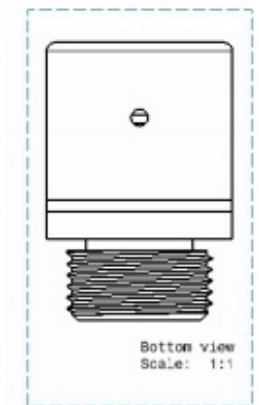
5. Click on a plane or face of the assembly model.
5. Click on the drawing sheet to generate the exploded view of the assembly.



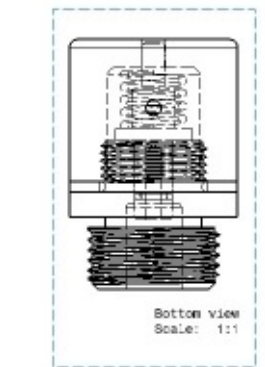
Isometric view
Scale: 2:3

View Properties

When working with CATIA V5 drawings, you can control the way a model view appears by using the view properties. For example, if you want to show or hide the hidden lines, then follow the steps given next.

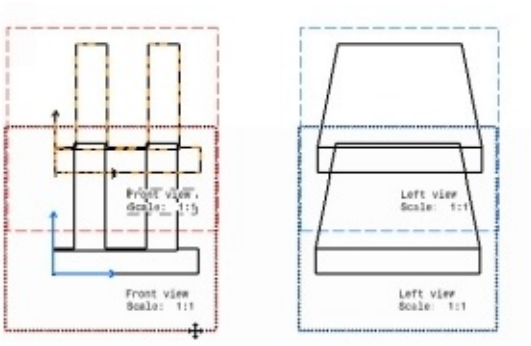


1. Click the right mouse button on the view and select **Properties**.
2. On the **Properties** dialog, under the **Dress-up** section, check the **Hidden Lines** option.
3. Click **OK** to apply the changes.

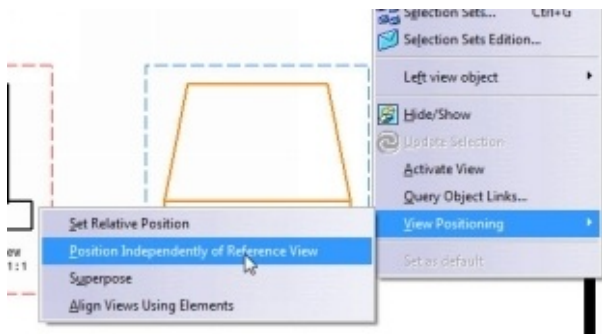


View Alignment

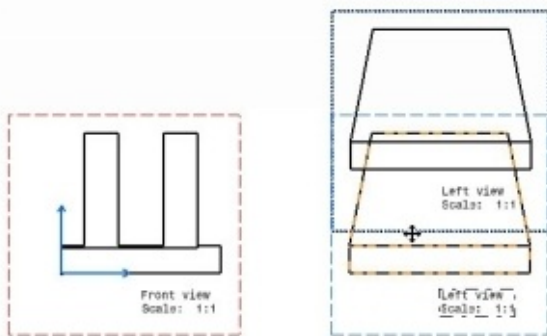
There are several types of views that are automatically aligned to a parent view. These include projected views, auxiliary views, and section views. For example, if you want to move any view, you need to move its parent view.



You can make the view independent of its parent view by breaking the link between them. To do this, click the right mouse button on the view. Select **View Positioning > Position Independently of Reference view**.



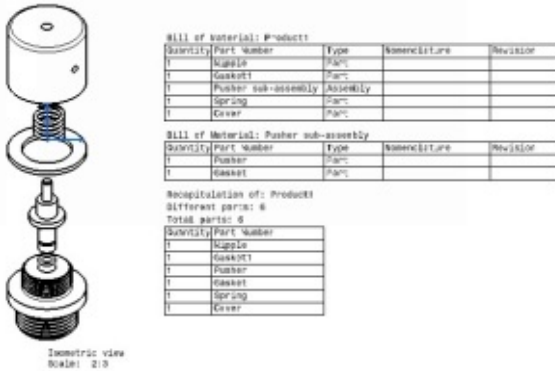
Now, you can move the view independently.




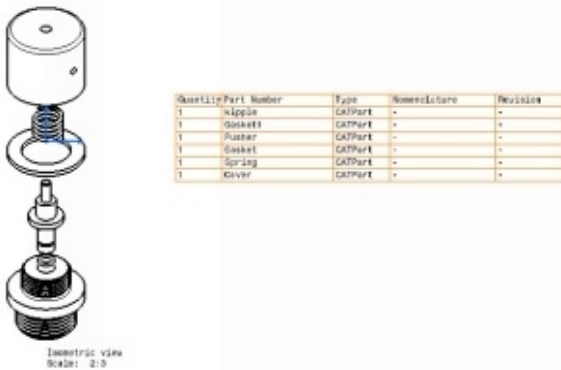
Bill of Material

Creating an assembly drawing is very similar to creating a part drawing. However, there are few things unique in an assembly drawing. One of them is creating parts list. A parts list identifies the different components in an assembly. Generating a parts list is very easy in CATIA V5. First, you need to have a view of the assembly.

1. On the Menu bar, click **Insert > Generation > Bill of Material > Bill of Material**.
2. Click on the drawing sheet to position the bill of materials.



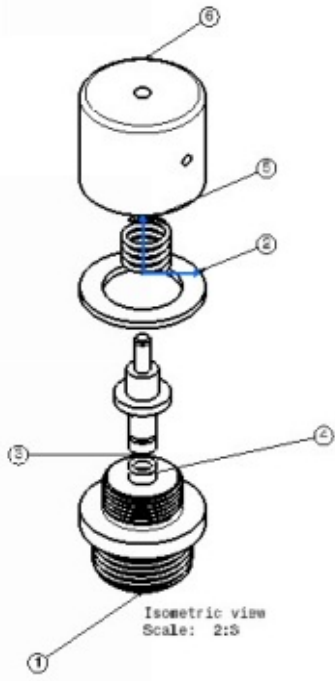
If you want only one set of bill of materials, then click **Insert > Generation > Bill of Material > Advanced Bill of Material**  on the Menu bar. Click **OK** on the **Bill of Material Creation** dialog. Select the assembly view and click on the drawing sheet to position the bill of material.



If you want to modify the bill of material, then click the right mouse button on the bill of material. Select **Properties** to open the **Properties** dialog. On the **Properties** dialog, click on the tabs and modify the properties. Click **OK** to apply the changes.

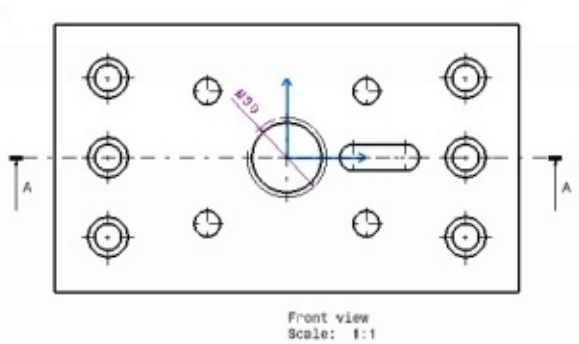
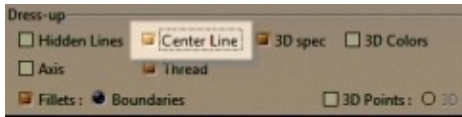
Balloons

To add balloons to the assembly drawing, on the **Generation** toolbar, click **Dimension Generation** drop-down > **Generate Balloons** (or) click **Insert > Generation > Balloon Generation** on the Menu bar.



Centerlines

Centerlines are used in engineering drawings to denote hole centers and lines. If you want to display the centerlines of a drawing view, then open the **Properties** dialog of the view and check the **Centerline** option. Click **OK** to close the dialog.



However, if you want to create centerlines manually, then use the commands available on the **Dress-up** toolbar.



Center Line

Use the **Center Line** command to add centerlines manually.

1.

On the **Dress-up** toolbar, click **Axis and Threads** drop-down > **Center Line** (or) click **Insert** > **Dress-up** > **Axis and Threads** > **Center Line** on the Menu bar.

2. Click on a circle or arc to add centerline to it.



Center Line with Reference

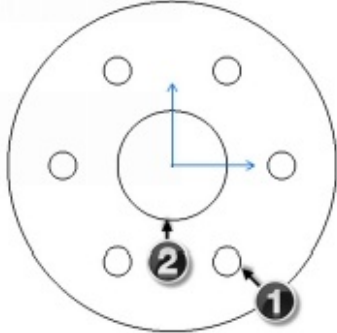
The **Center Line with Reference** command allows you to add centerlines that are at an angle to some reference. This command will be useful to add centerlines to holes that are arranged in a circular fashion.

1.

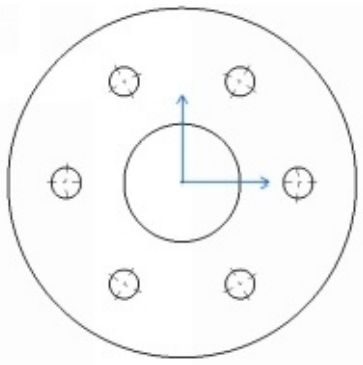
On the **Dress-up** toolbar, click **Axis and Threads** drop-down > **Center Line with**

Reference (or) click **Insert > Dress-up > Axis and Threads > Center Line with Reference** on the Menu bar.

2. Click on the hole to add a center line.
3. Click on a point or edge to define the reference. A centerline will be created on the hole.



4. Likewise, create centerlines on other holes.



Dimensions

CATIA V5 provides you with different types of commands to add dimensions to the drawing.



Generate Dimensions

One of the methods to add dimensions to the drawing is to retrieve the dimensions that are already contained in the 3D part file. The **Generate Dimensions** command helps you to do this.

1.

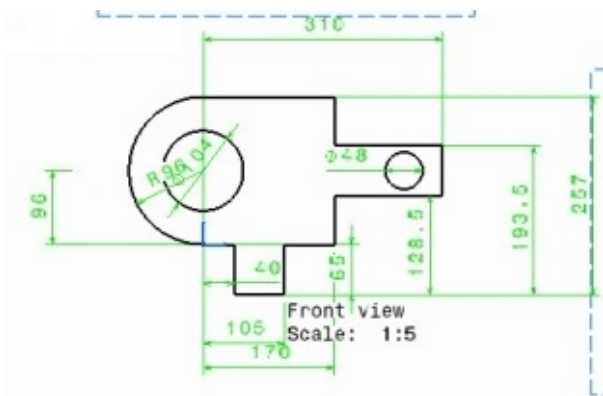
On the **Generation** toolbar, click **Dimension Generation** drop-down > **Generate Dimensions** (or) click **Insert** > **Generation** > **Generate Dimensions** on the Menu bar.

2.

Select the type of constraints and dimensions to be retrieved from the 3D model.

3.

Click **OK** on the **Generated Dimension Analysis** dialog. This generates dimensions of the model.



Generate Dimensions Step by Step

This command helps you to generate dimensions in a step-by-step manner.


1.

On the **Generation** toolbar, click **Dimension Generation** drop-down > **Generate Dimensions Step by Step** (or) click **Insert** > **Generation** > **Generate Dimensions Step by Step** on the Menu bar.

2.

On the **Step by Step Generation** dialog, type-in a value in the **Timeout** box. For example, if you type-in 1 in this box, the time taken to generate each dimension will be one second.


3.

Click the **Next Dimension Generation**  icon on the dialog. You will notice that the dimensions are generated one by one. Also, other options on the dialog become selectable. The functions of these options are given next.

 Click this button to generate all the dimensions of the model.

 Click this button to stop the dimension generation and close the dialog.

 Click this button to pause the dimension generation.


 Click this button to exclude a dimension.

 Click this button to transfer a dimension to another view.

4.

Click **OK** to complete the dimension generation.

Adding Dimensions

If you want to add some more dimensions, which are necessary to manufacture a part, use the **Dimensions**  command (activate it from the **Dimensioning** toolbar) to create linear, radial or any type of dimension. As you activate this command, the **Tools Palette** toolbar appears.



You can use the options on this toolbar to define the orientation of the dimension.



Chained Dimensions

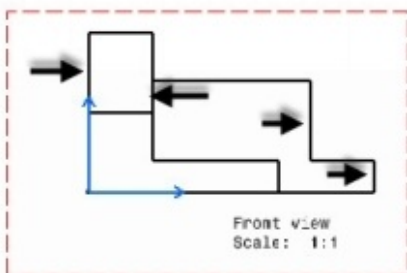
This command creates chained dimensions.

1.

On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Chained Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Chained Dimensions** on the Menu bar.

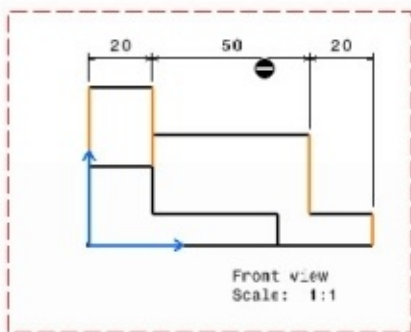
2.

Select two or more parallel edges from the drawing view.



3.

Move the pointer and click to position the chained dimension.





Cumulated Dimensions

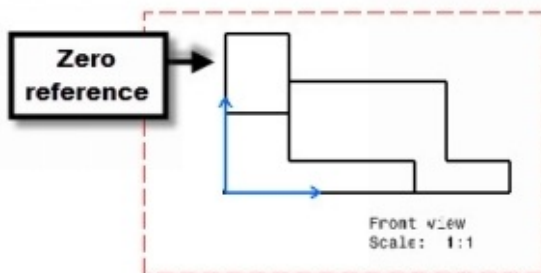
Cumulated dimensions are another type of dimensions that you can add to a drawing.

1.

On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Cumulated Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Cumulated Dimensions** on the Menu bar.

2.

Click on any edge of the drawing view to define the zero reference.



3.

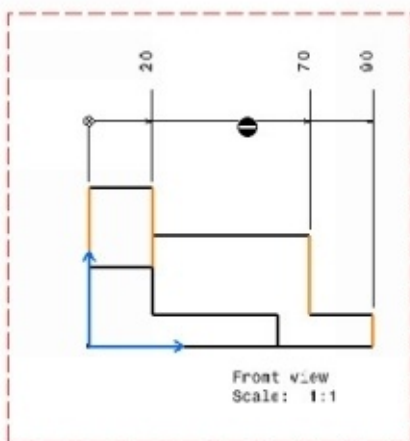
Now, click on an edge of the drawing view.

4.

Likewise, click on other edges parallel to the zero reference.

5.

Move the pointer and click to place the cumulated dimension.





Stacked Dimensions

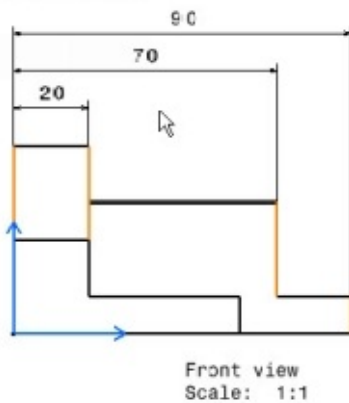
This command creates stacked dimensions.

1. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Stacked Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Stacked Dimensions** on the Menu bar.
- 2.

Select two or more parallel edges from the drawing view.

3.

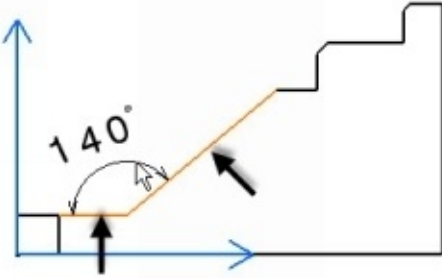
Move the pointer and click to position the stacked dimension.



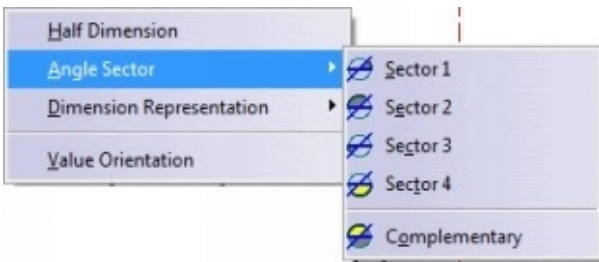
Angle Dimensions

This command creates angle dimensions.

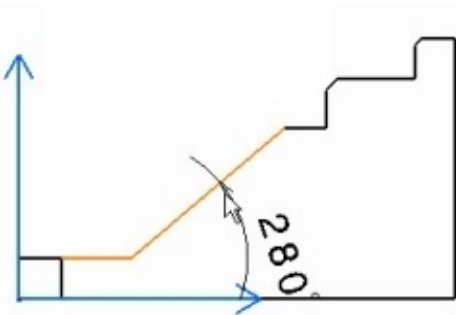
1. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Angle Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Angle Dimensions** on the Menu bar.
2. Select two lines, which are positioned at angle to each other.



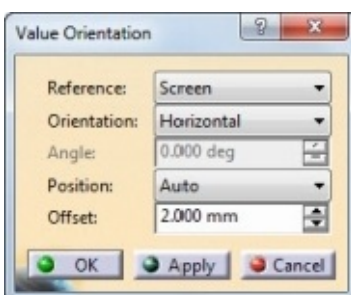
3. Click the right mouse and select any angle sector.

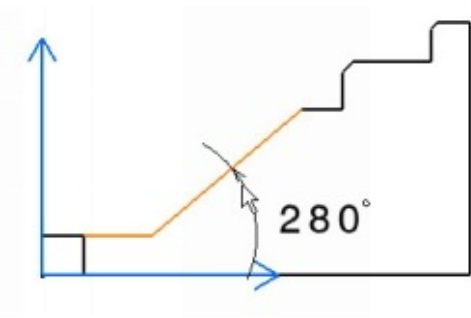


4. Click the right mouse button and select **Half Dimension**, if you want the half dimension of the angle.



5. Click the right mouse button and select **Value Orientation**. This displays the **Value Orientation** dialog. On this dialog, you can define the orientation of the angle value. For example, if you set the **Reference** to **Screen** and **Orientation** to **Horizontal**, the value will be orientated horizontal to the screen.





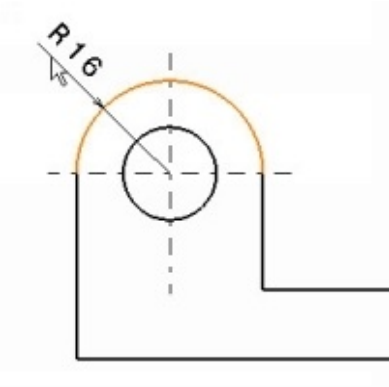
Also, you can use the **Position** drop-down to position the value outside or inside the dimension line. Type-in a value in the **Offset** box to define the gap between the value and the dimension line.

5. Click **OK** to close the dialog.
7. Click to position the dimension.

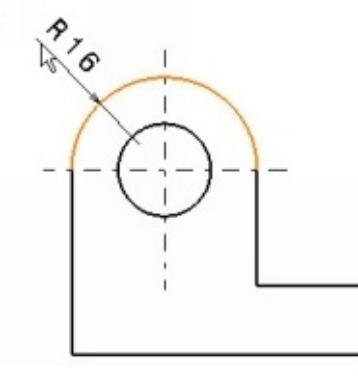
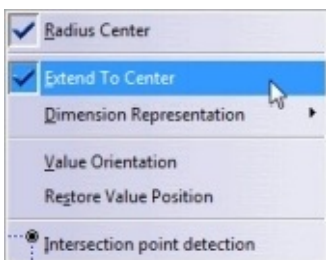
Radius Dimensions

This command creates radius dimensions.

1. On the **DIMENSIONING** toolbar, click **Dimensions** drop-down > **Radius Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Radius Dimensions** on the Menu bar.
2. Select an arc to be dimensioned.



3. Click the right mouse button and deselect the **Extend to Center** option. This creates a radius dimension without extending it to the center of the arc.



4. Click to place the dimension.

Diameter Dimensions

This command is used to create a diameter dimension for a shafted geometry.

1.

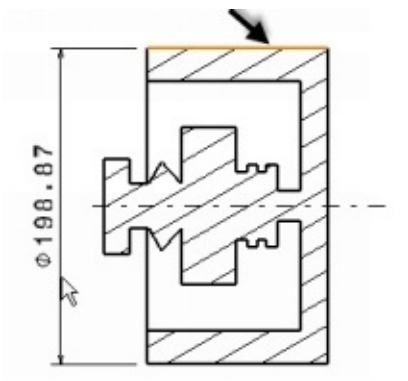
On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Diameter Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Diameter Dimensions** on the Menu bar.

2.

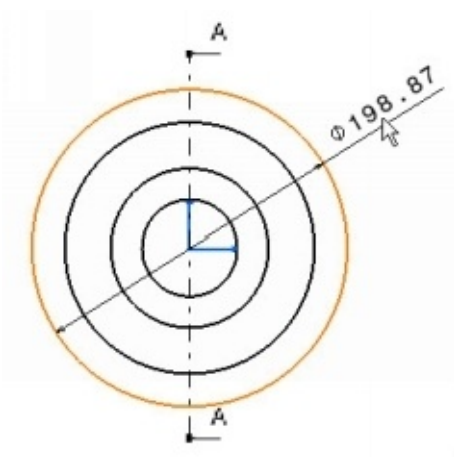
Click on the silhouette edge of the shaft feature. You will notice that the diameter dimension of the shaft feature appears.

3.

Move the pointer and click to position the dimension.



You can also select circular edges to add diameter dimensions.



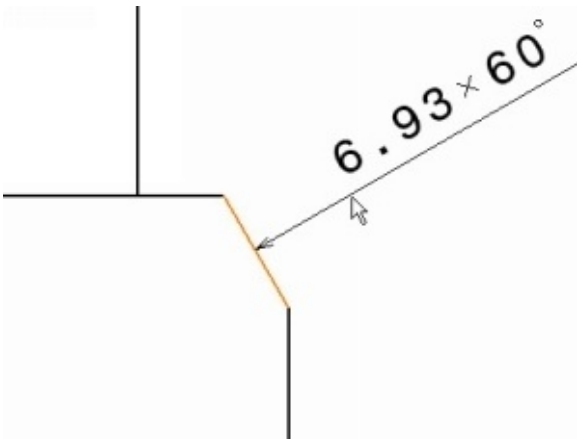
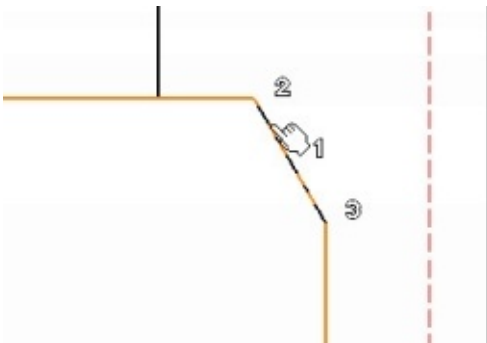


Chamfer Dimensions

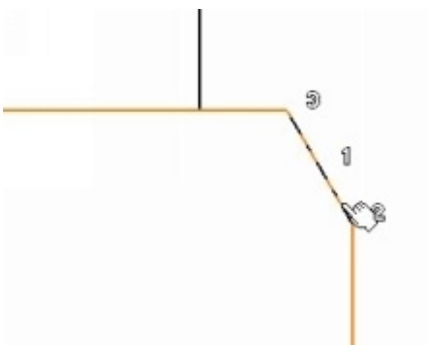
This command creates a chamfer dimension.

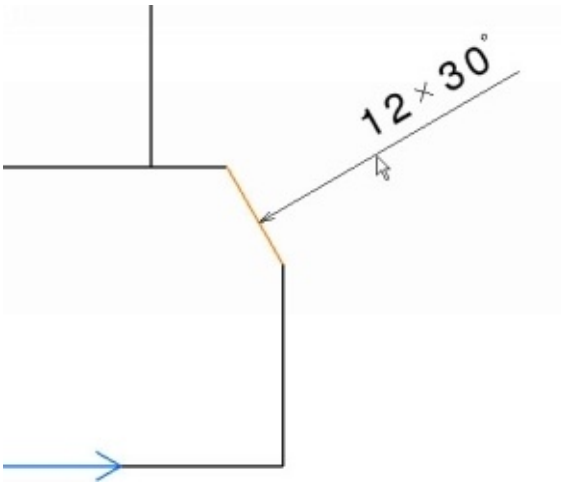
1. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Chamfer Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Chamfer Dimensions** on the Menu bar.
2. Click on the top portion or bottom portion of the chamfer.

If you click on the top portion, the chamfer dimension will be created with reference to the top edge of the chamfer.




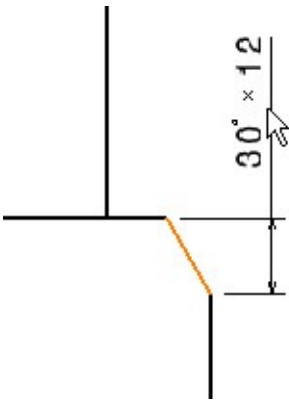
If you click on the bottom portion, the chamfer dimension will be created with reference to the bottom edge of the chamfer.





3. On the **Tools Palette** toolbar, select the format of the chamfer dimension. You can select **Length x Length**, **Length x Angle**, **Angle x Length**, or **Length**.

If you want a double-arrowed dimension, then select the **Two Symbols**  icon on the **Tools Palette** toolbar.

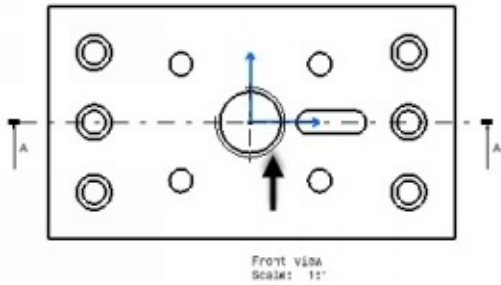


4. Move the pointer and click to position the dimension.

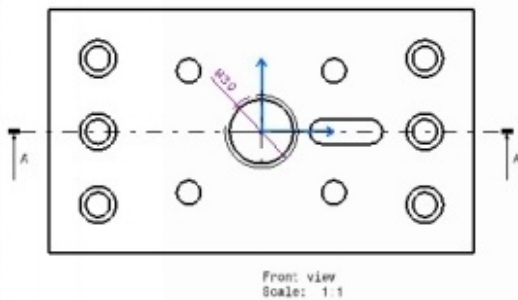
Thread Dimensions

This command generates the dimensions of a threaded feature.

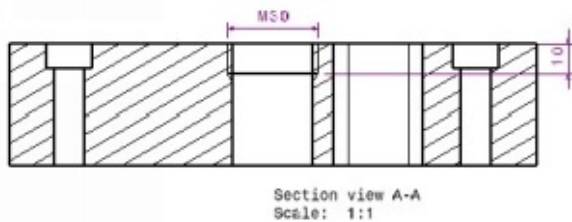
1. Click the right mouse button on the drawing view with a thread feature.
2. Select **Properties**, and then check the **Thread** option under the **Dress up** section.
3. Click **OK** to close the dialog. The thread symbol appears in the drawing view.



4. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Thread Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Thread Dimensions** on the Menu bar.
5. Select the thread symbol from the drawing view. This generates the dimension of the thread.



Likewise, you can select the thread symbol on the side or section views.

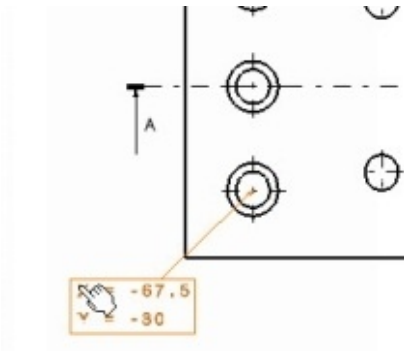




Coordinate Dimensions

This command generates the coordinate values of hole features. You need to ensure that the centerlines of the holes are created.

1. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Coordinate Dimensions** (or) click **Insert** > **Dimensioning** > **Dimensions** > **Coordinate Dimensions** on the Menu bar.
2. Select the centerlines of the hole.
3. Move the pointer and click to position the dimension.



T Text

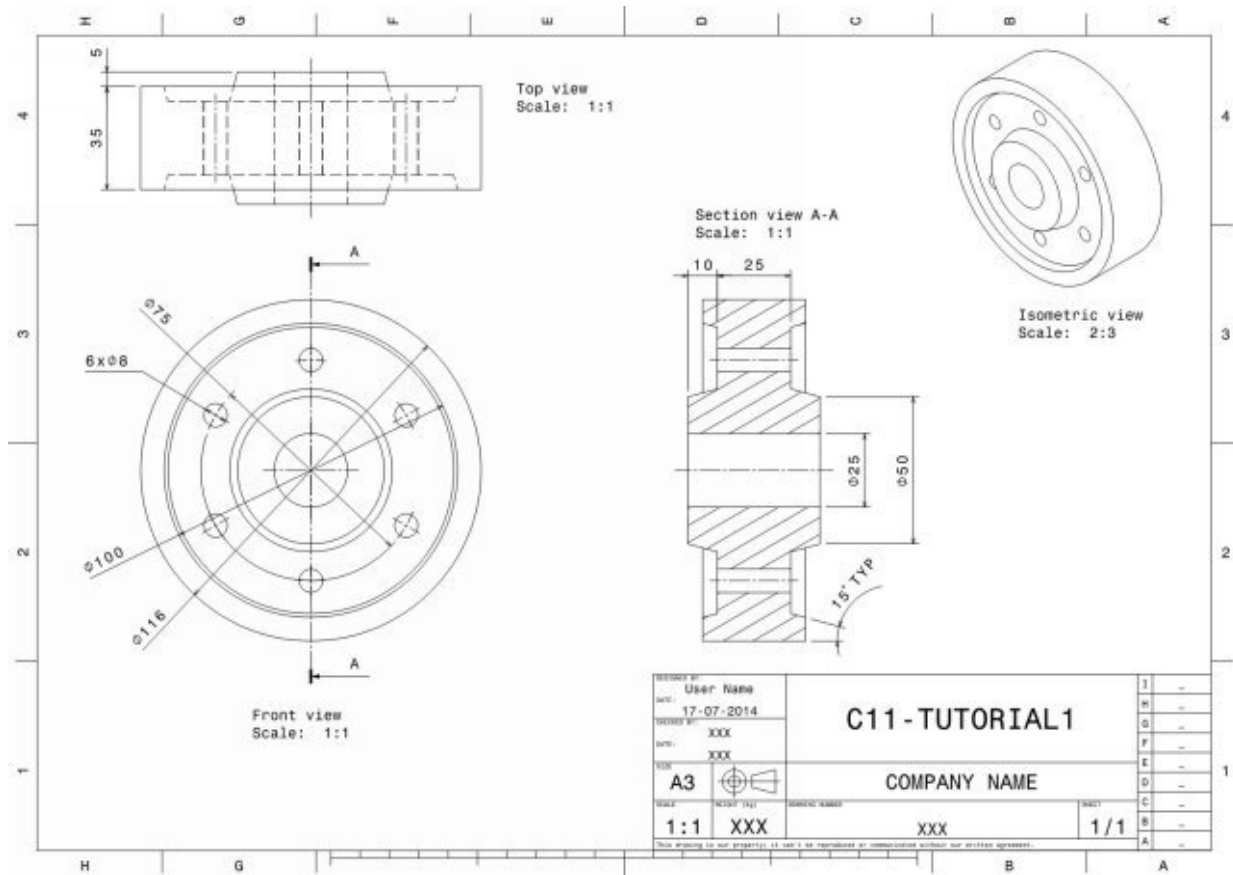
Texts are important part of a drawing. You add text to provide additional details, which cannot be done using dimensions and annotations.

1. On the **Annotations** toolbar, click **Text** drop-down > **Text** (or) click **Insert** > **Annotations** > **Text** > **Text** on the Menu bar.
2. Click on the drawing sheet to define the location of the text.
3. Type-in text in the **Text Editor** dialog, and then click **OK**.

Examples

Example 1


In this example, you will create 2D drawing of the part shown below.

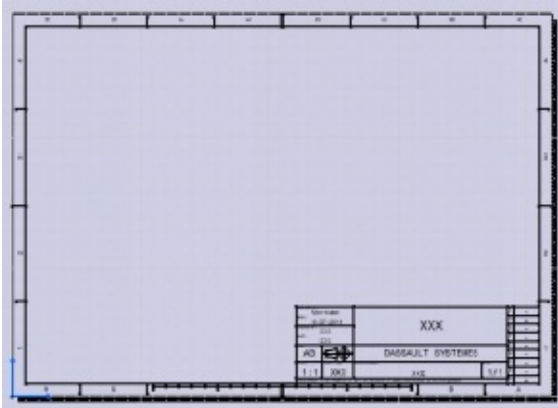


Starting a New Drawing

1. Start CATIA V5-6R2015.
2. Open the Exercise 1 file of Chapter 5.
3. On the **Standard** toolbar, click the **New** button.
4. On the **New** dialog, click **List of Types > Drawing**. Click **OK**.
5. On the **New Drawing** dialog, select **Standard > ISO**.
6. Set **Sheet Style** to **A3 ISO**.
7. Select **Landscape** option and click **OK**.
8. At the left side of the window, click the right mouse button on **Sheet.1** and select **Properties**.
9. On the **Properties** dialog, set **Projection Method** to **Third angle standard**.
0. Click **OK**.


Adding Borders and Title Block

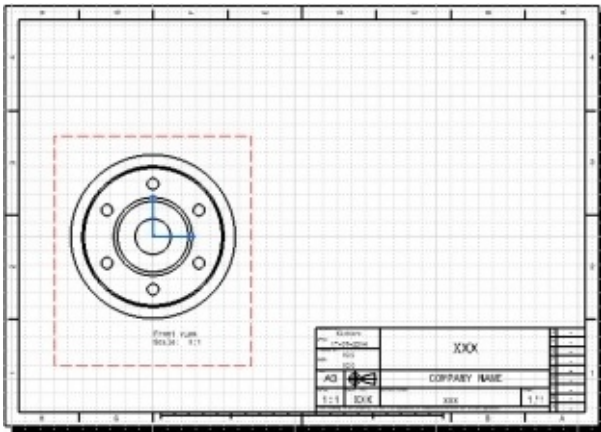
1. On the Menu bar, click **Edit > Sheet Background**. This brings up the sheet background.
2. On the Menu bar, click **Insert > Drawing > Frame and Title Block** (or) click the **Frame and Title Block**  button on the **Drawing** toolbar.
3. On the **Manage Frame and Title Block** dialog, select **Style of Title Block > Drawing Titleblock Sample 1**.
4. Select **Action > Create**, and then click **OK**.




5. In the title block, double-click on DASSAULT SYSTEMES and type-in your company name in the Text editor. Click **OK** to update the company name.
6. Double-click on the XXX located in the largest cell of the title block, and then type-in **C11-EXAMPLE1** in the Text editor. Click **OK** to update the drawing title.
7. On the Menu bar, click **Edit > Working Views**. This brings up the drawing sheet.

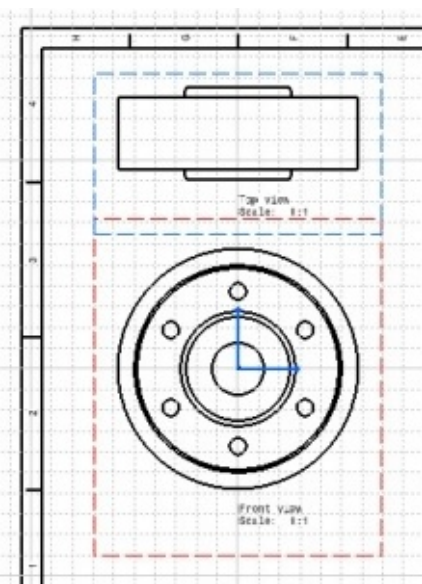
Generating Drawing Views

1. On the **View** toolbar, click the **Front View**  button (or) click **Insert > Views > Projections > Front View** on the Menu bar. Now, you have to select a model face or reference plane to define the front view.
2. On the Menu bar, click **Window > C05-Exercise1.CATPart**. This switches you to the part file window.
3. Click on the front face of the model geometry. The front view of the model appears on the drawing sheet.
4. Drag the front view to the left side of the drawing sheet (click and drag the green dotted lines).
5. Click on the drawing sheet to generate the view.



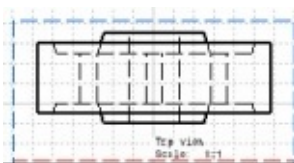
Now, you have to create the projected view.

5. On the **View** toolbar, click **Projections** drop-down > **Projection View**  (or) click **Insert > Views > Projections > Projection**.
7. Move the pointer up and click to position the projected view.



Now, you have to turn-on the hidden lines of the projected view.

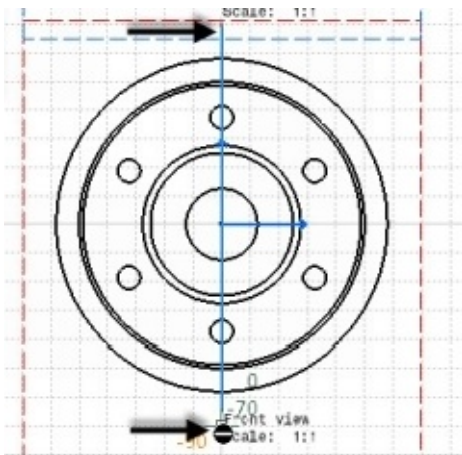
3. Click the right mouse button on the projected view and select **Properties**.
5. On the **Properties** dialog, under the **Dress-Up** section, check the **Hidden Lines** option, and then click **OK**. The hidden lines appear on the projected view.



Now, you have to create the section view.

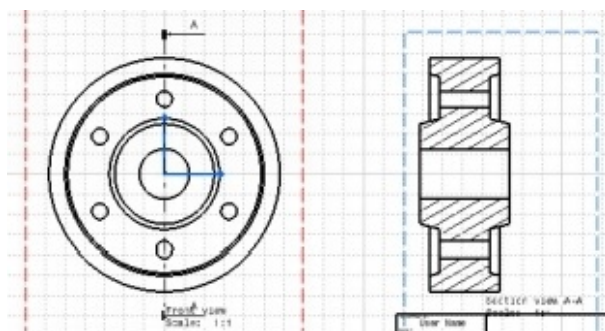
0. On the **View** toolbar, click the **Offset Section View**  button (or) click **Insert > Views > Sections > Offset Section View**.

1. Draw a vertical line passing through the center of the front view.




2. Double-click on the endpoint of line.

3. Move the pointer toward right and click to position the section view.



Now, you have to create the Isometric view.

4. On the **View** toolbar, click **Projections** drop-down > **Isometric View**  (or) click **Insert > Views > Projections > Isometric** on the Menu bar.

5. On the Menu bar, click **Window > C05-Exercise1.CATPart**. This switches you to part file window.

6. Click on the front face of the model geometry. The Isometric view of the model appears on the drawing sheet.

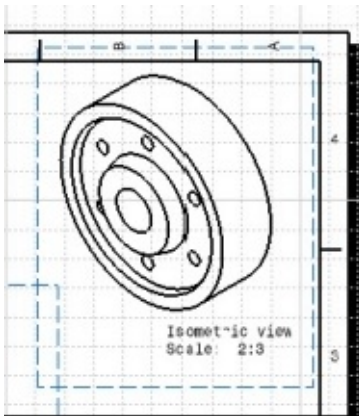
7. Drag the view to the top-right corner of the drawing sheet.

8. Click on the drawing sheet to generate the Isometric View.

You can notice that the size of the Isometric View is large. You have to scale the Isometric View to have enough space for dimensions.

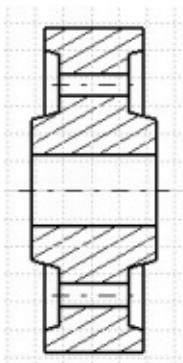
9. Click the right mouse button on the Isometric view and select **Properties**.

0. On the **Properties** dialog, under the **Scale and Orientation** section, type-in 2:3 in the **Scale** box. Click **OK** to update the view.
1. Drag the view to the top-right corner.

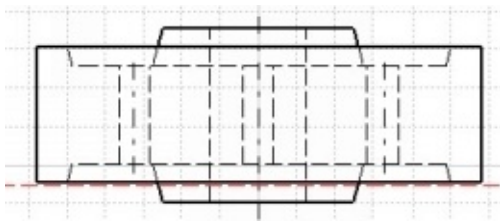


Add Axis lines


1. Click the right mouse button on the section view and select **Properties**.
2. On the **Properties** dialog, under the **Dress-up** section, check the **Axis** option.
3. Click **OK** to add axis lines to the section view.



4. Likewise, add axis lines to the top view.




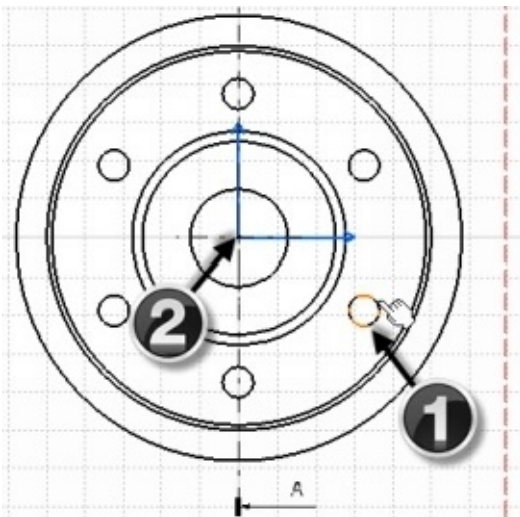
Add Center Lines to the Front View

1. On the **Dress-Up** toolbar, click **Axis and Threads** drop-down > **Center Line**  (or) click **Insert** > **Dress-up** > **Axis and Threads** > **Center Line** on the Menu bar.
2. Click on the circle located at the center of the front view. This adds a centerline the front view.

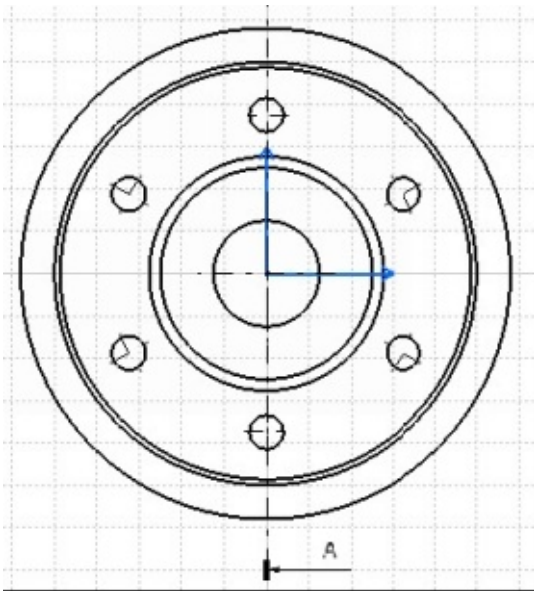


Now, you have to add centerlines to the other circles on the front view. Keep in mind that the orientation of the centerlines is different for each circle. To create centerlines with different orientation, you can use the **Center Line with Reference** command.


3. On the **Dress-Up** toolbar, click **Axis and Threads** drop-down > **Center Line with Reference**  (or) click **Insert** > **Dress -up** > **Axis and Threads** > **Center Line with Reference** on the Menu bar.
4. Select anyone of the small circles, and then the large circle located at the center. This creates a centerline on small circle with reference to the center point of the large circle.

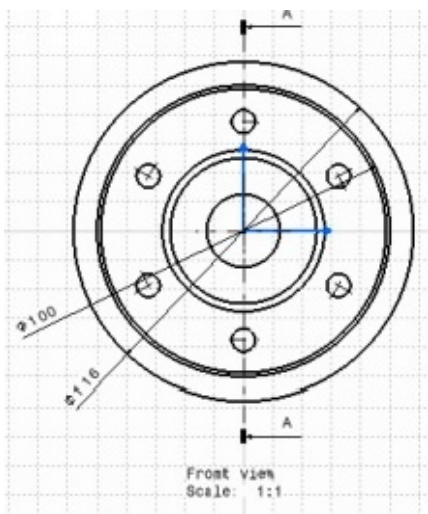


5. Likewise, create centerlines on the other small circles.

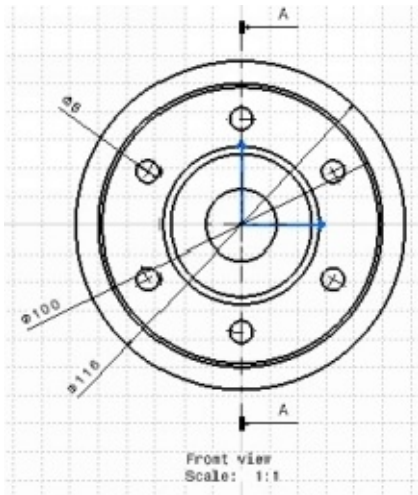


Add Dimensions to the drawing

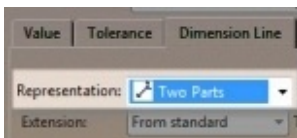
1. On the **Dimensioning** toolbar, click **Dimensions** drop-down > **Dimensions**  (or) click **Insert** > **Dimensioning** > **Dimensions** > **Dimensions** on the Menu bar.
2. Zoom into the front view and click on the outer most circle.
3. Move the pointer and click to position the dimension.
4. Likewise, dimension the second circle from outside.



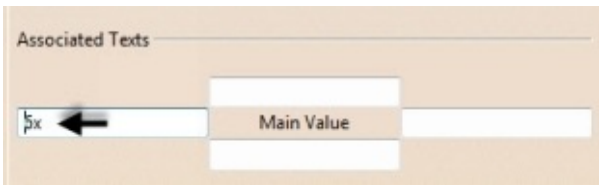
5. Activate the **Dimensions** command and click the small circle located at left side on the front view.
6. Move the pointer and click to position the dimension.



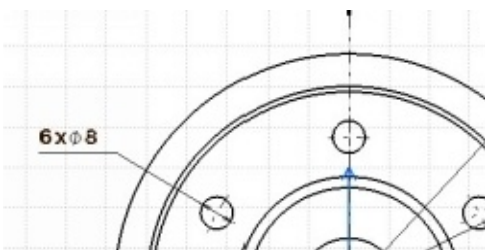
7. Click the right mouse button on the dimension and select **Properties**.
3. On the **Properties** dialog, click the **More** button located at the bottom right. This displays additional tabs on the dialog.
3. Click the **Dimension Line** tab and select **Representation > Two Parts**.



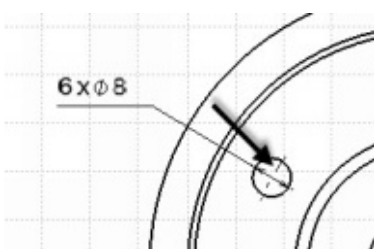
0. Click the **Dimension Texts** tab and type-in 6x in the box located on left-side of Main value.



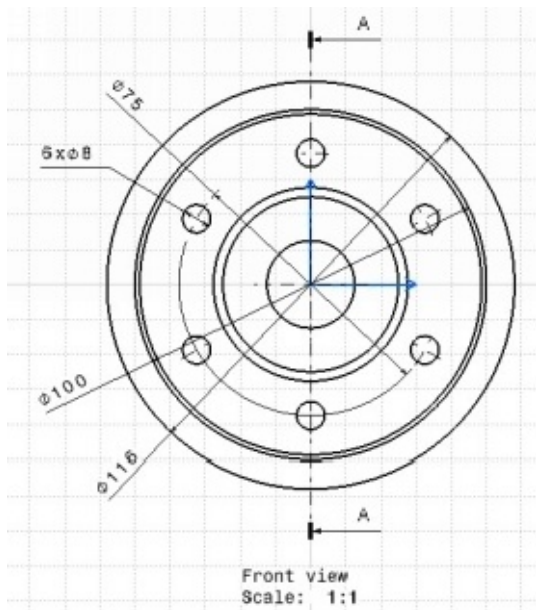
1. Click **OK**.



2. Activate the **Dimensions** command and click on the centerline of any one of the small circles.



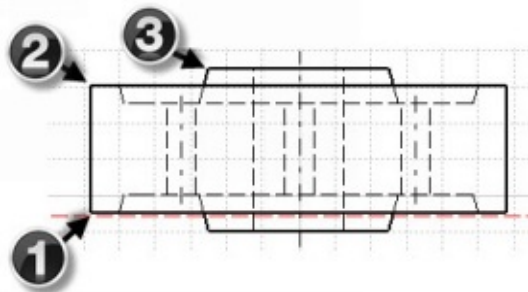
3. Move the pointer and click to position the dimension.



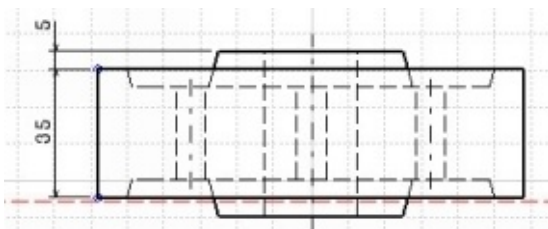
4. On the **Dimensioning** toolbar, click **Dimensions drop-down > Chained Dimensions**  (or) click **Insert > Dimensioning > Dimensions > Chained Dimensions** on the Menu bar.

5. On the **Tools Palette** toolbar, click the **Intersection point detection**  button.

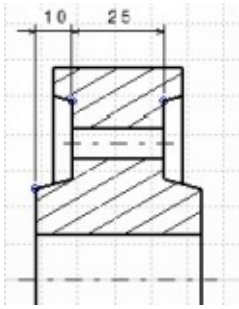
6. Zoom into the top view and click on the end points, as shown below.



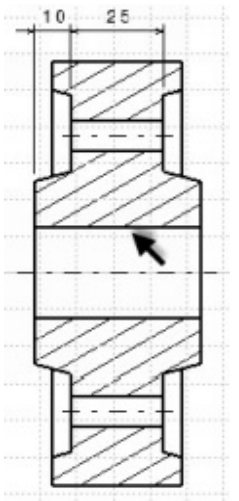
7. Move the pointer toward left and click to position the dimension.



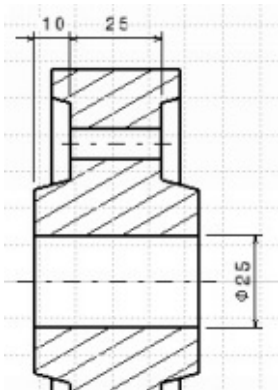
8. Likewise, create a chained dimension on the section view.



9. On the **Dimensioning** toolbar, click **Dimensions drop-down > Diameter Dimensions** (or) click **Insert > Dimensioning > Dimensions > Diameter Dimensions** on the Menu bar.
0. On the **Tools Palette** toolbar, deactivate the **Intersection point detection** button.
1. Zoom in to the section view and click on the innermost horizontal edge.



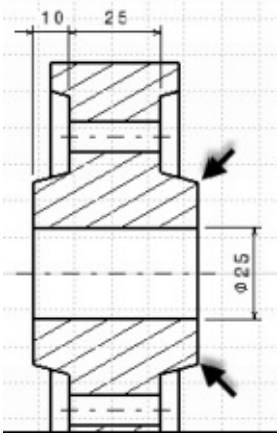
2. Move the pointer toward right and click to position the dimension.



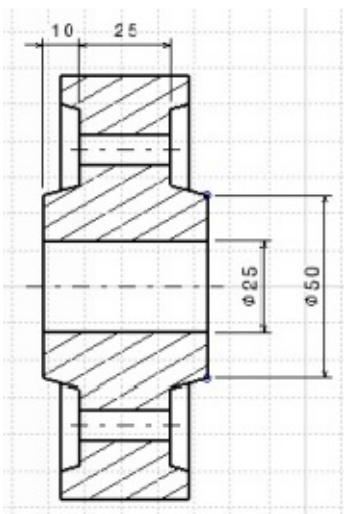
3. Activate the **Diameter Dimensions** command and select the **Intersection point detection** button on the **Tools Palette** toolbar.



4. On the section view, select the vertices of the inclined edges, as shown below.



5. Move the pointer and click to position the dimension.

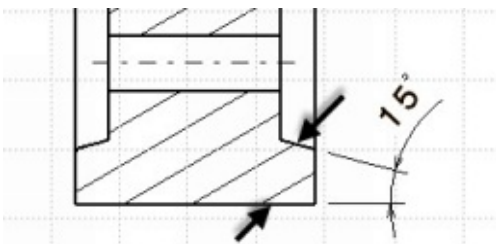


Now, you have to add an angular dimension to the inclined edge of the section view.

6. On the **Dimensioning** toolbar, click **Dimensions drop-down > Angle Dimensions**  (or) click **Insert > Dimensioning > Dimensions > Angle Dimensions** on the Menu bar.

7. Zoom into the section view and select the inclined and horizontal edges at the bottom.

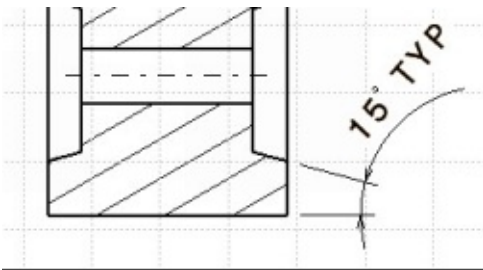
8. Move the pointer toward right and click to position the angle dimension.



9. Click the right mouse button on the angle dimension and select **Properties**.
10. On the **Properties** dialog, click the **Dimension Text** tab, and then type-in TYP in the box located at right side of Main value.

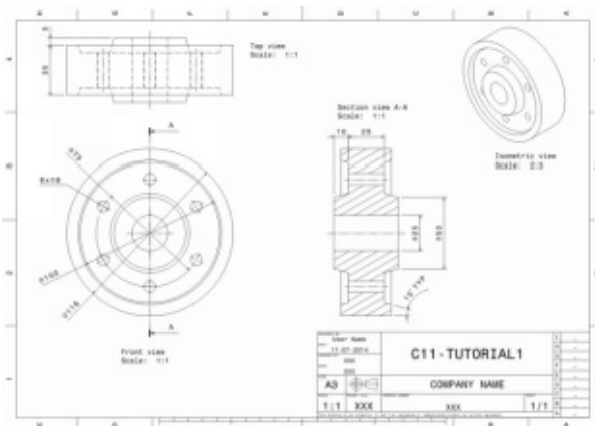


1. Click **OK** to update the dimension.



Printing the drawing

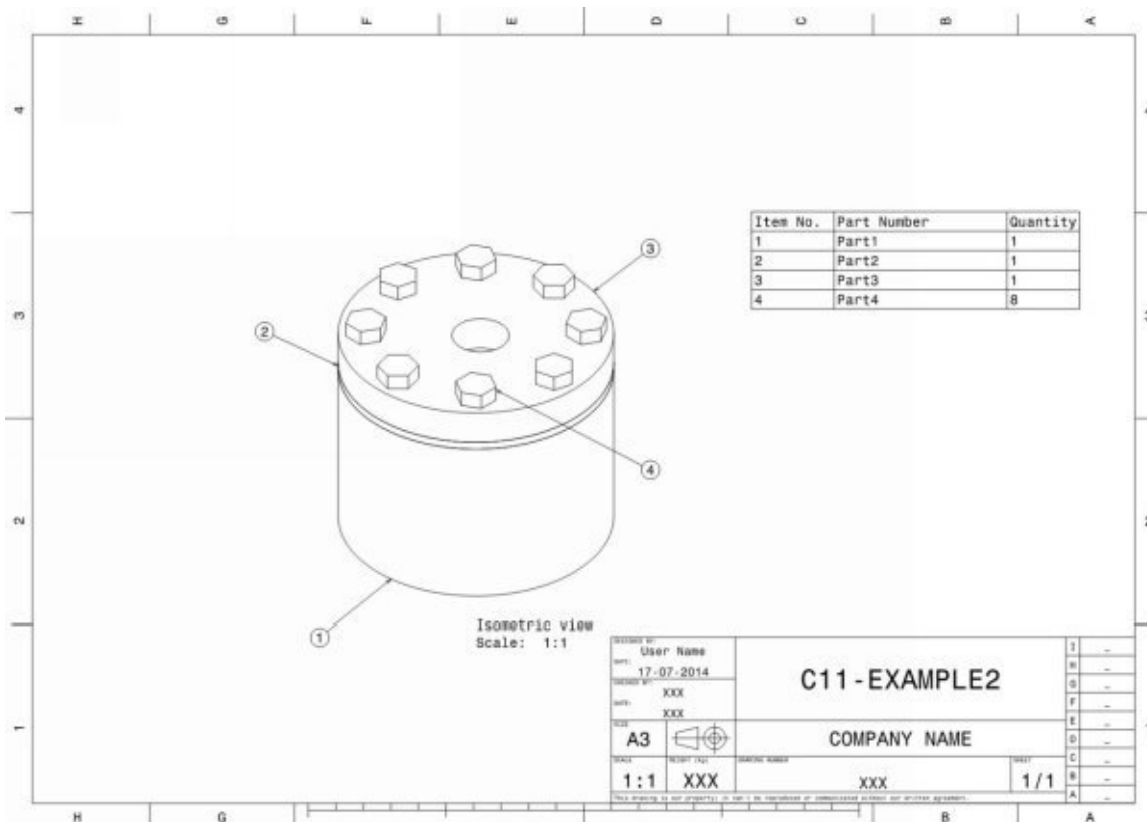
1. On the Menu bar, click **File > Print**.
2. On the **Print** dialog, click the **Page Setup** button to open the **Page Setup** dialog.
3. On the **Page Setup** dialog, check the **Use image format** option and click **OK**.
4. Select the printer from the **Printer Name** drop-down, and then click **OK**.
5. Specify the name and location of the file (in case of a PDF or image file).




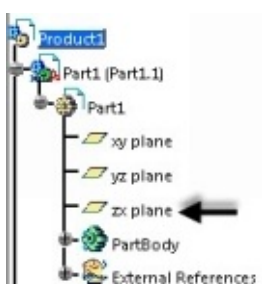
3. Save and close the drawing.

Example 2

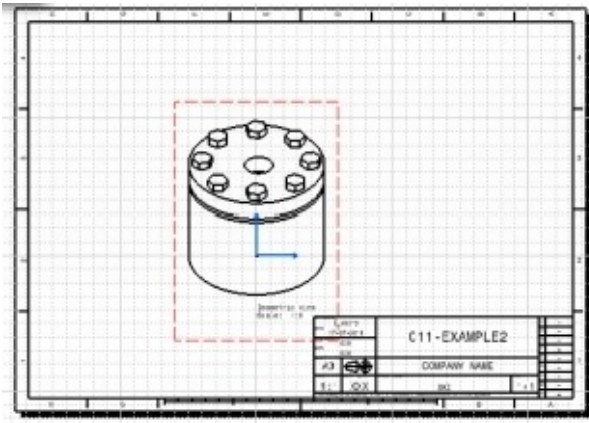
In this example, you will create an assembly drawing shown below



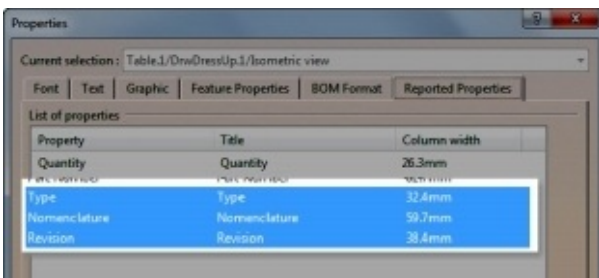
1. Open the Example 2 file that you have created in Chapter 10.
2. Click **Start > Mechanical Design > Drafting**.
3. On the **New Drawing Creation** dialog, click the **Modify** button to open the **New Drawing** dialog.
4. On this dialog, set the **Sheet Style** to **A3 ISO**, and click **OK**.
5. On the **New Drawing Creation** dialog, click the **Empty sheet** option, and then click **OK**.
6. On the Menu bar, click **Edit > Sheet Background** and add title block to the background.
7. Update the company name and drawing title.
8. On the Menu bar, click **Edit > Working Views**.
9. On the **Views** toolbar, click **Projections** drop-down > **Isometric View** .
10. On the Menu bar, click **Window > C10-Example2** to switch to the assembly file.
11. In the Specification Tree, expand Part1 and select zx plane.





2. Click on the drawing sheet to generate the Isometric view of the assembly.




3. On the Menu bar, click **Insert > Generation > Bill of Material > Advanced Bill of Material**.
4. Click **OK**.
5. Click on the top-right corner to position the bill of material.
6. Click the right mouse button on the bill of material and select **Properties**.
7. On the **Properties** dialog, click the **Reported Properties** tab to view the properties of the BOM.
8. From the **List of Properties** section, select **Type, Nomenclature, and Revision** (press and hold the Ctrl key and click on them).



9. At the lower right corner of the dialog, click the **Delete**  button.
0. Click the **New**  button to add a new property to the table.
1. At the bottom of the table, set the values, as shown in figure.



2. Select the **Item No.** property from the table and click **Move Up**  twice. This moves the **Item No.** property to the top of the table.
3. Likewise, move the **Quantity** property to the bottom of the table.

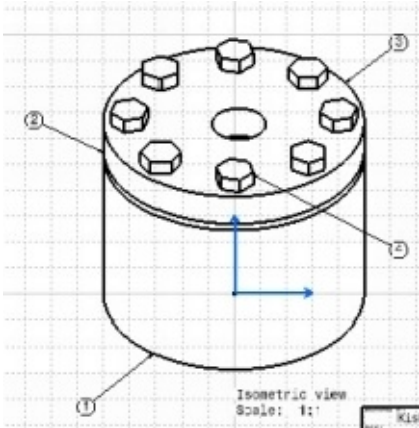
Property	Title	Column width
Item No.	Item No.	30mm
Part Number	Part Number	62.7mm
Quantity	Quantity	26.3mm

4. Click **OK** to update the BOM.

Item No.	Part Number	Quantity
1	Part1	1
2	Part2	1
3	Part3	1
4	Part4	8

5. On the Menu bar, click **Insert > Generation > Balloon Generation**. This automatically generates the balloons.

6. Drag the balloons to arrange them properly.



7. Save and close the drawing.

Questions

1. How to create drawing views using **View Creation Wizard** command?
2. How do you show or hide hidden edges of a drawing view?
3. How do you retrieve dimensions of the 3D part model?
4. How to update the drawing views when the part is edited?
5. How do you control the properties of dimensions and annotations?
6. List the commands used to create centerlines and center marks.
7. How do you add symbols and texts to a dimension?
8. How do you add break lines to drawing view?
9. How do you create revolved section views?
10. How do you create exploded view of an assembly?

Chapter 12: Sheet Metal Design

You can make sheet metal parts by bending and forming flat sheets of metal. In CATIA V5, sheet-metal parts can be folded and unfolded enabling you to show them in the flat pattern as well as their bent-up state. There are two ways to design sheet-metal parts in CATIA V5. Either you can start the sheet-metal part from scratch using sheet-metal features throughout the design process or you can design it as a regular solid part and later convert it to a sheet-metal part. Most commonly, you design sheet-metal parts in **Generative Sheet metal Design** Workbench from the beginning. In this chapter, you will learn both the approaches.

The topics covered in this chapter are:

- *Walls*
- *Walls of edges*
- *Extrusion*
- *Flanges*
- *Bend Allowance*
- *Hems*
- *Tear Drops*
- *User Flanges*
- *Bends*
- *Conical bends*
- *Bend from flat*
- *Unfolding*
- *Folding*
- *Fold/Unfold*
- *Stamps*
- *Cut outs*
- *Convert to Sheet Metal*
- *Export to DXF*

Starting a Sheet Metal part

To start a new sheet metal part, follow the steps given next.

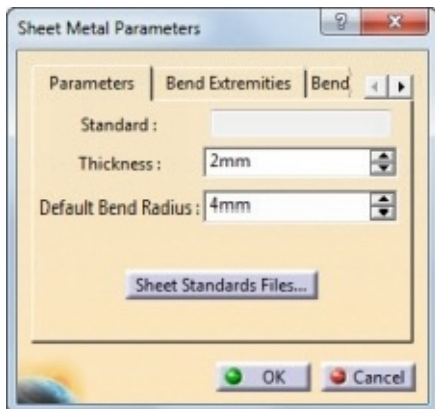
1. Select **Start > Mechanical Design > Generative Sheet Metal Design** on the Menu bar.
2. On the **New Part** dialog, type-in the name of the part, and then click **OK**.



Sheet Metal Parameters

Sheet Metal Parameters define the material thickness, bend size, bend extremities, and bend allowances. You can define these parameters by using the **Sheet Metal Parameters** command.

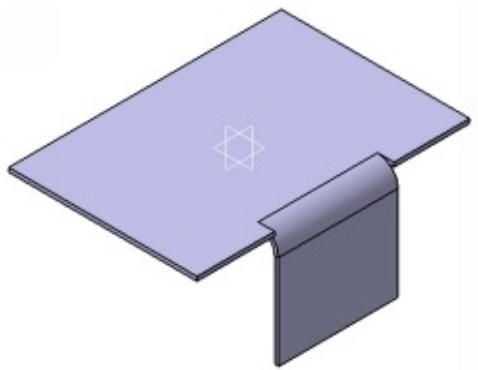
1. On the **Walls** toolbar, click the **Sheet Metal Parameters** button (or) click **Insert > Sheet Metal Parameters** on the Menu bar.
2. On the **Sheet Metal Parameters** dialog, type-in **Thickness** and **Default Bend Radius** values. You can also use an excel sheet to define these values. Click the **Sheet Standard files** button and select the worksheet or text file containing the thickness and bend radius values.



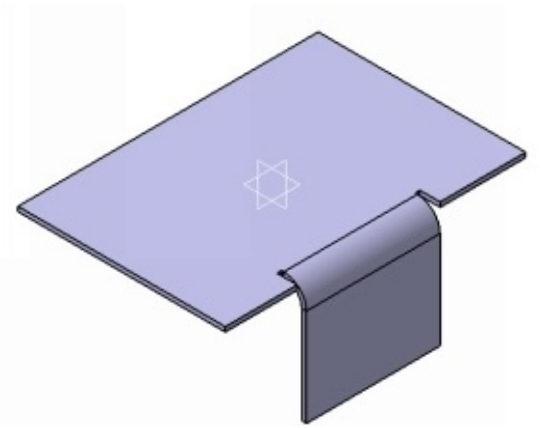
3. Click the **Bend Extremities** tab and select the type of relief to be provided to a bend. The options available in this tab are given next.



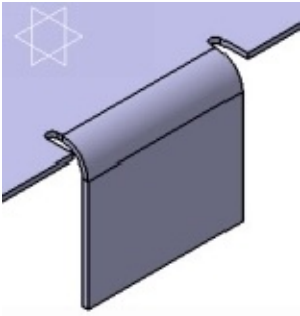
Minimum with no relief: This option provides no relief at the bend corners.



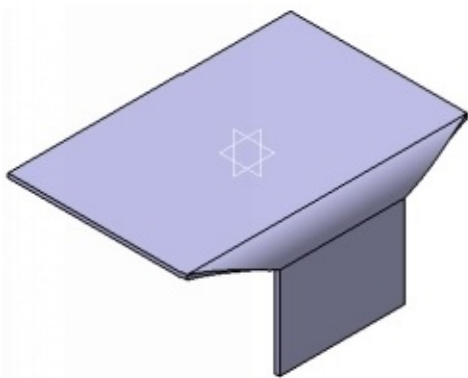
Square relief: A square relief is applied to bend extremes.



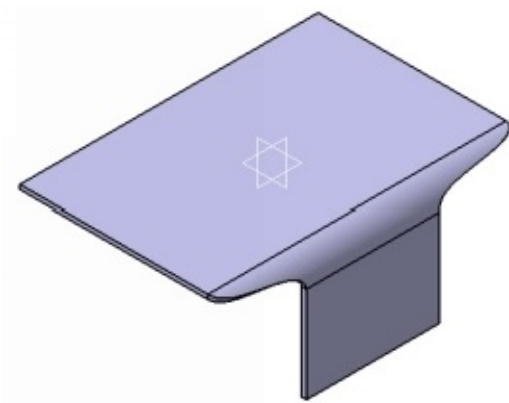
Round relief: A round relief is applied to bend extremes.



Linear: A linear relief is added between the end faces of the bend and supporting wall.

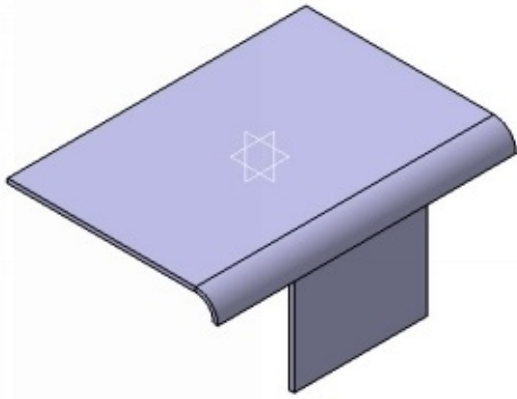


Tangent: The end faces of the bend will be tangent to the end faces of the supporting wall.

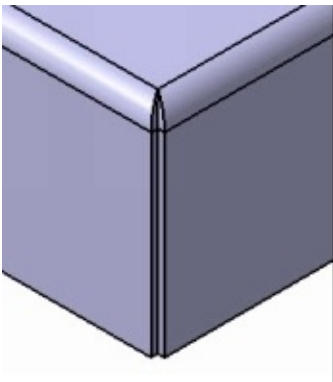




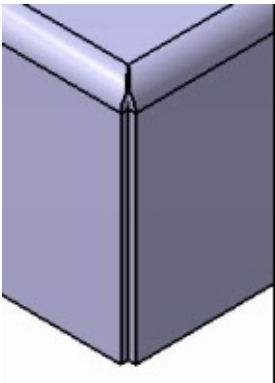
Maximum: This option provides maximum relief at the bend extremes.



Closed: This option closes the corner formed by two intersecting bends.



Flat joint: This option applies no relief between two intersecting bends.



4. Click the **Bend Allowance** tab to view the K factor.

The **K Factor** is the ratio that represents the location of the neutral sheet measured from the inside face with respect to the thickness of the sheet metal part. The Neutral Factor defines the bend allowance of the sheet metal part. The standard formula that calculates the bend allowance is given below.

$$BA = \frac{\pi(R + KT)A}{180}$$

$A = \text{Bend Allowance}$

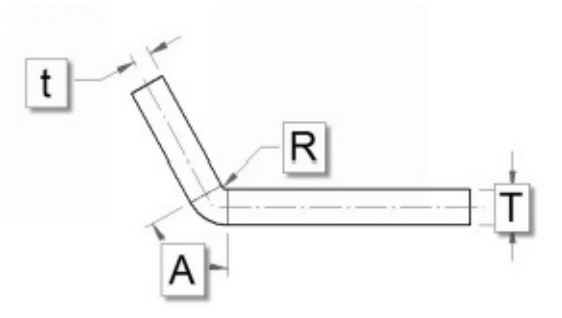
$R = \text{Bend Radius}$

$K = \text{Neutral Factor} = t/T$

$T = \text{Material Thickness}$

$t = \text{Distance from inside face to the neutral sheet}$

$A = \text{Bend Angle}$



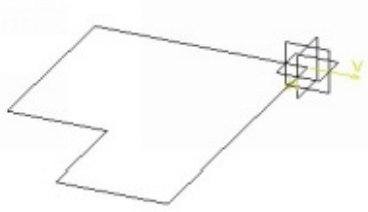
You can also define the bend allowance by using your own bend allowance formula. To enter a bend allowance formula, select the **formula editor** button located next to the **K factor** box. On the **Formula Editor** dialog, define a new formula using the available parameters.


5. Click **OK**.

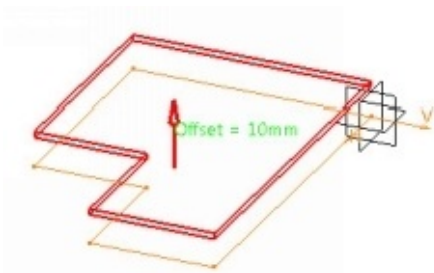
Wall

The wall is a basic type of sheet metal feature.

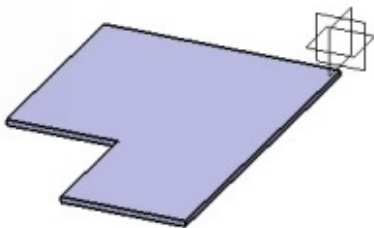
1. To create a wall, create a closed sketch on a plane.



2. On the **Walls** toolbar, click the **Wall** button (or) click **Insert > Walls > Wall** on the Menu bar.
3. Click on the sketch.
4. On the **Wall Definition** dialog, click the **Sketch at middle position**  button, if you want the sketch to be at the middle of the wall thickness.
5. Type-in a value in the **Offset** box, if you want to create the wall at an offset from the sketch.



6. Click the **Invert Side** button to reverse the direction of the wall.
7. Click **OK**.

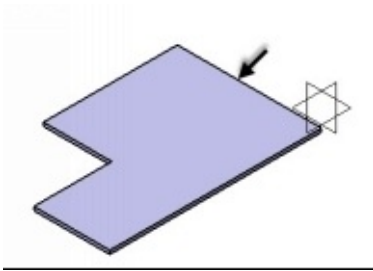




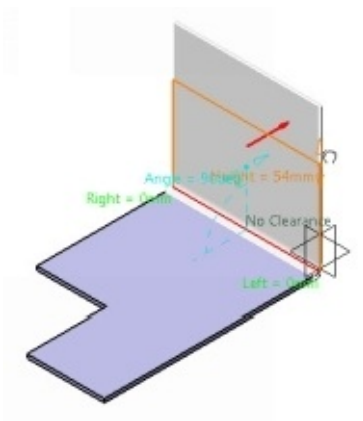
Wall on Edge

The second feature after creating a wall is wall on edge. You can create this feature along an edge or multiple edges of a sheet metal part.

1. On the **Walls** toolbar, click the **Wall on Edge** button (or) click **Insert > Walls > Wall on Edge** on the Menu bar.
2. Click an edge of the wall feature. The wall preview appears on the selected edge.



3. Click the green arrow and drag the pointer to change the height of the wall. You can also type-in a value in the **Height** box.



4. Set the **Length** type. The length type options are given next.



This option measures the length of the wall from the outer face of the base wall up to the outer edge of the new wall.



This option measures the length of wall from the inner face of the base wall up to the outer edge of the new wall.



This option measures the length of wall excluding the bend.



This option measures the length of wall from its outer edge up to the intersection

point between the outer faces of new wall and base wall.



This option measures the length of the wall from its outer edge up to the intersection point between the inner faces of new wall and base wall.

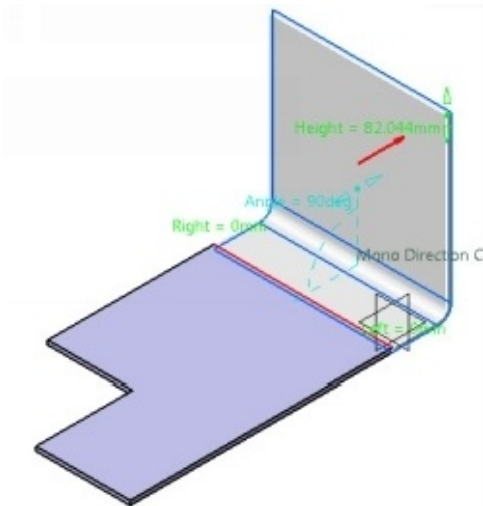
5. Type-in a value in the **Angle** box.

If you want to match the outer face of the wall with a plane or face, then click **Angle > Orientation plane** and select a plane. The wall will be oriented according to the selected plane.

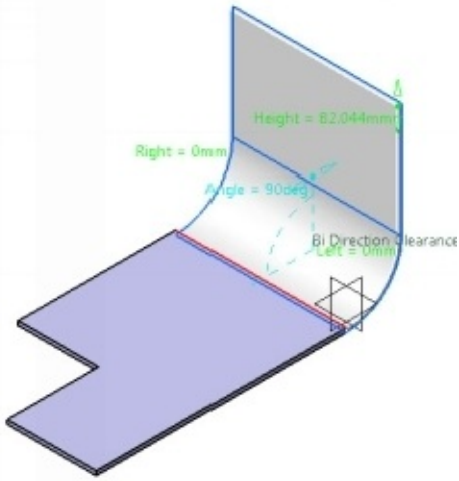
5. Select an option from the **Clearance mode** drop-down. These options are given next.

No clearance: This option creates a wall without any clearance.

Monodirectional: Applies clearance between the selected edge and bend portion of the wall.

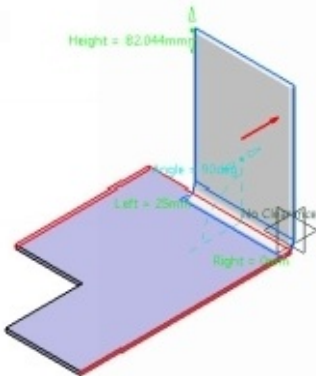


Bidirectional: Applies clearance on both sides of the bend. You can type-in a clearance value or use a formula to define the clearance.

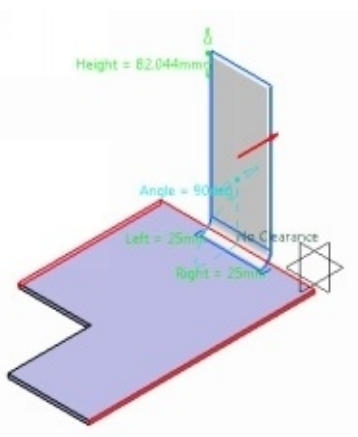


Click the **Reverse Direction** button, if you want to reverse the side of the wall. Use the **Invert Material Side** button to reverse the material side.

On the **Extremities** tab, click in the **Left limit** box and select a plane or face to define the left limit of the wall. You can type-in a value in the **Left offset** box to offset the left limit.



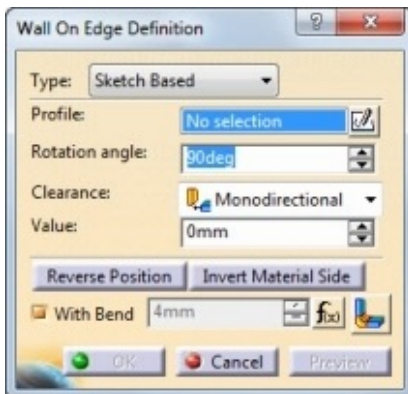
Likewise, define the right limit, offset using the **Right limit** and **Right offset** options.



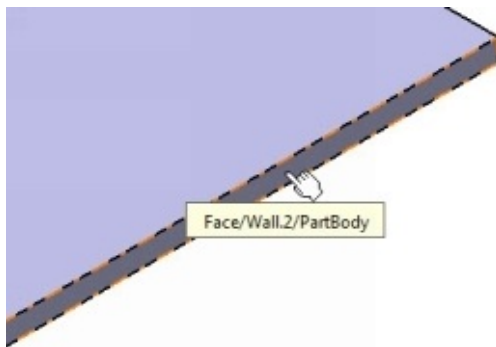
7. Click **OK**.

Sketch Based Wall on Edge

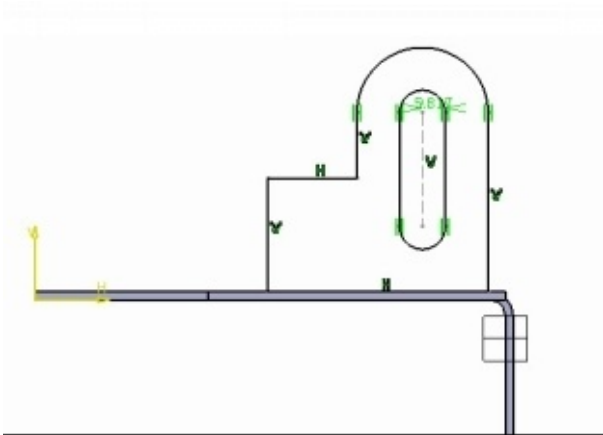
1. Activate the **Wall on Edge** command and select **Type > Sketch Based** on the **Wall On Edge Definition** dialog.



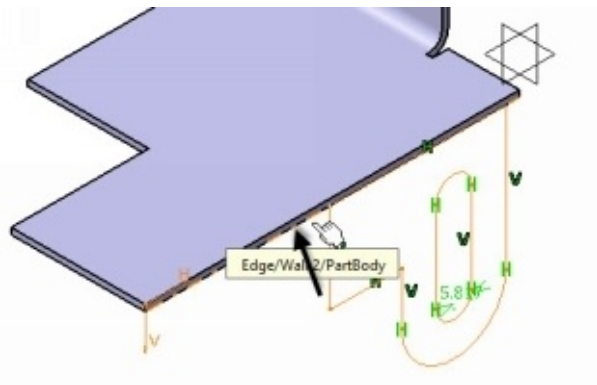
2. Click the sketch icon on the dialog and select the end face of the sheet metal wall.



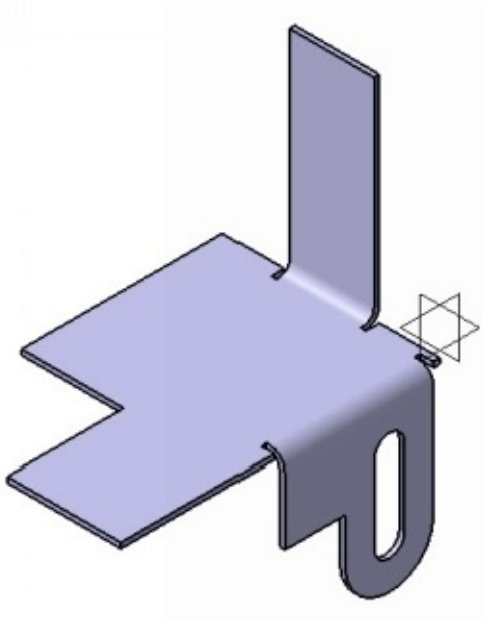
3. Draw the profile of the wall and exit the Sketecher workbench.



4. Select the edge coinciding with the sketch.



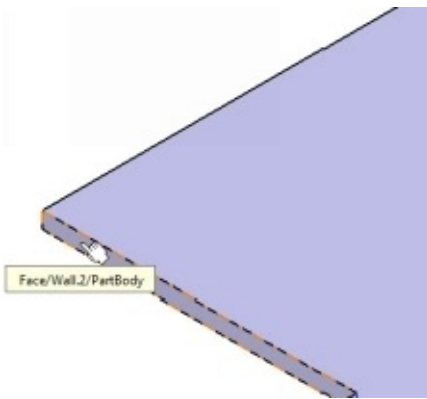
5. Type-in a value in the **Rotation Angle** box, if you want to create an inclined wall.
5. Click **OK** to create the wall.



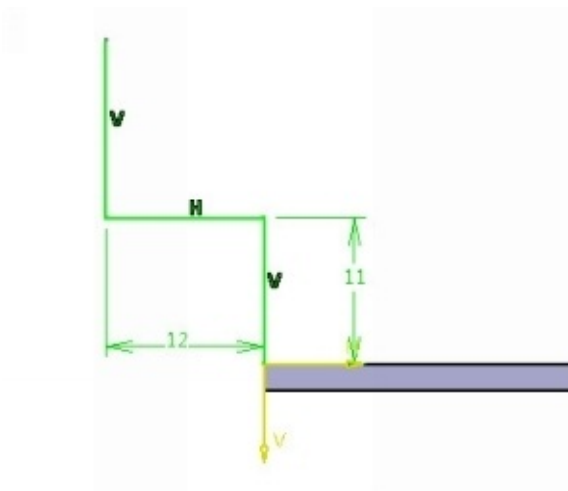
Extrusion

The **Extrusion** command extrudes an open sketch.

1. On the **Walls** toolbar, click the **Extrusion** button (or) click **Insert > Walls > Extrusion** on the Menu bar.
2. On the **Extrusion Definition** dialog, click the sketch icon, and then select a face or plane.



3. Draw open profile and exit the sketch.



4. Select **Limit 1 dimension** from the **Limit 1** drop-down, and then type-in the extrusion distance.

You can select a **Limit 1 up to plane** or **Limit 1 up to surface** to define the first limit of the extrusion.

5. Likewise, define the extrusion distance along second direction using the **Limit 2** drop-down.
6. Check the **Mirrored extent** option, if you want to extrude the sketch in both the

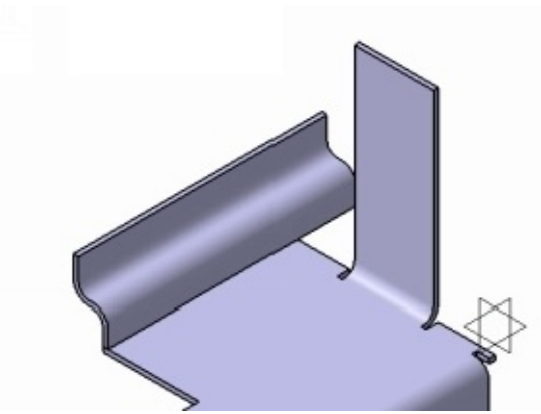
directions.

The **Automatic bend** option creates bends at the intersections between the sketch elements.

The **Exploded mode** creates individual features of each sketch elements.

Use the **Invert material side** and **Invert direction** button to reverse the material side and extrusion direction, respectively.

7. Click the **More** button to view additional options of the extrusion.
8. Click **OK** to complete the extrusion feature.

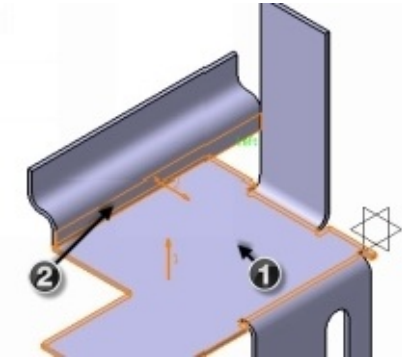


You will notice that there is no bend between the extrusion and the base wall. You can create a bend using the **Bend** command.

Bend

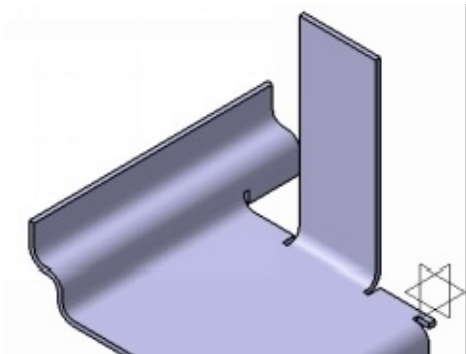
This command creates a bend between two faces.

1. On the **Bending** toolbar, click **Bends** drop-down > **Bend** (or) click **Insert** > **Bending** > **Bend** on the Menu bar.
2. Select the support faces.



You will notice that all the bend parameters are greyed out on the dialog because the default sheet metal parameters will be used.

Click OK.

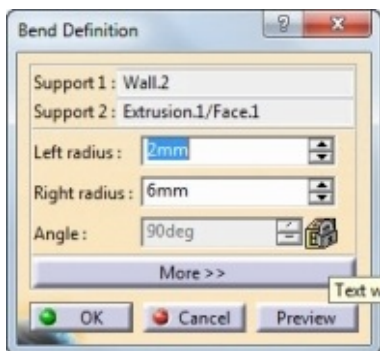




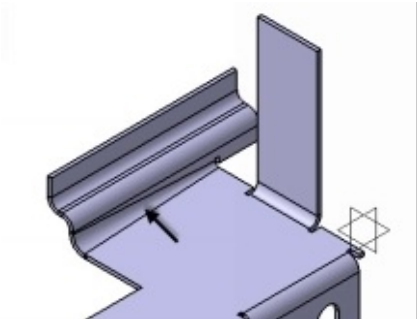
Conical Bend

This command creates a variable radius bend between two faces.

1. On the **Bending** toolbar, click **Bends** drop-down > **Conical Bend** (or) click **Insert** > **Bending** > **Conical Bend** on the Menu bar.
2. Select the support faces.
3. On the **Bend Definition** dialog, specify the **Left radius** and **Right radius** values.



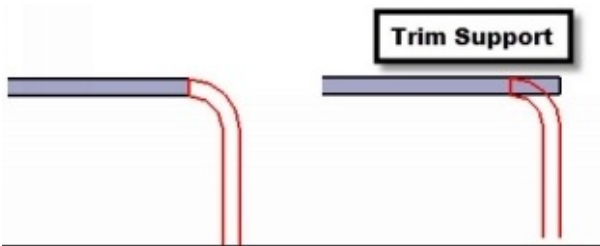
4. Click **OK**.



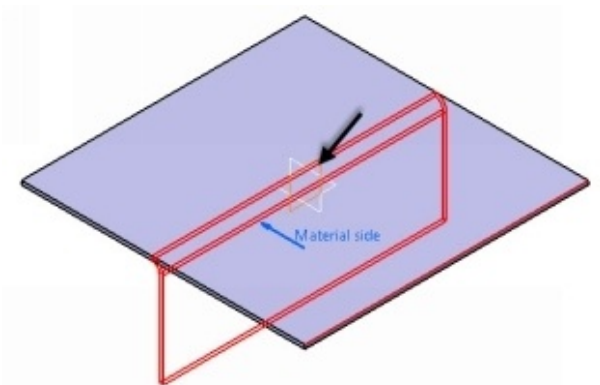
Flange

This command creates a flange by sweeping a parametric profile along the selected edge.

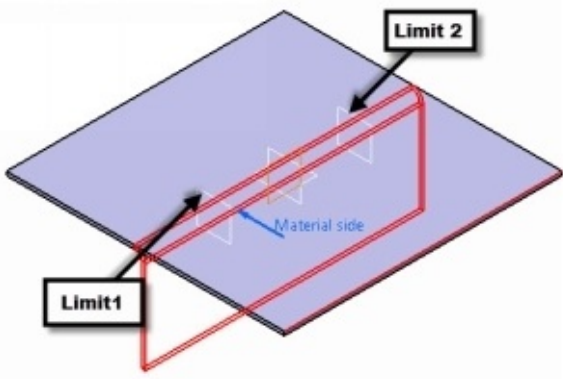
1. On the **Walls** toolbar, click **Swept Walls** drop-down > **Flange** (or) click **Insert** > **Walls** > **Swept Walls** > **Flange** on the Menu bar.
2. Click on an edge of the base wall.
3. Click the **Propagate** button to select the tangentially connected edges. The preview of the flange appears.
4. Type-in values in the **Length** and **Angles** boxes.
5. Define the **Length type** by using the drop-down next to the **Length** box.
6. Likewise, define the **Angle type** by using the drop-down next to the **Angle** box.
7. Check the **Trim Support** option, if you want trim the supporting wall.



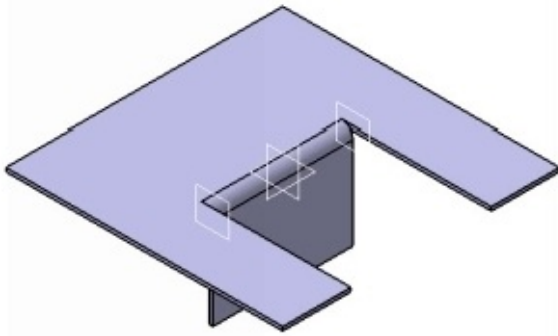
8. Check the **Flange Plane** option and click in the selection box next to it.
9. Select a plane. The flange will become coincident to that plane.



10. If you want to create the flange between two planes, then select the **Relimited** option from the drop-down located at the top on the dialog. Select the first and second limits by using the **Limit1** and **Limit2** options.



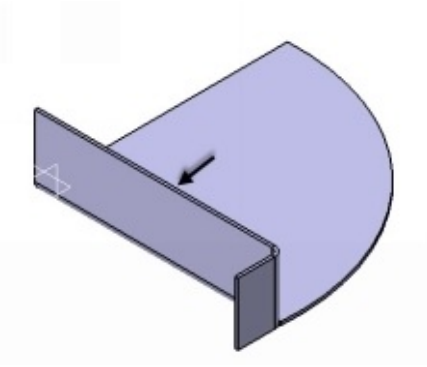
11. Click **OK**.



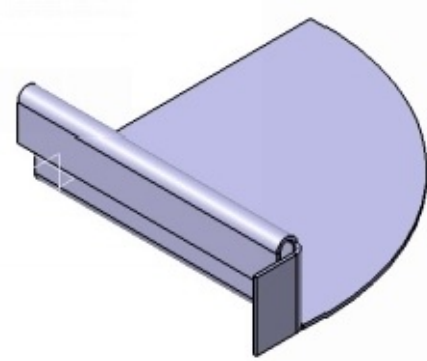
Hem

The **Hem** command folds an edge of a sheet metal part.

1. On the **Walls** toolbar, click **Swept Walls** drop-down > **Hem** (or) click **Insert** > **Walls** > **Swept Walls** > **Hem** on the Menu bar.
2. Select the edge you need to fold.



3. Type-in the length of the hem.
4. Click the **Propagate** button, if you want to fold the tangentially connected edges as well.
5. Click **OK**.

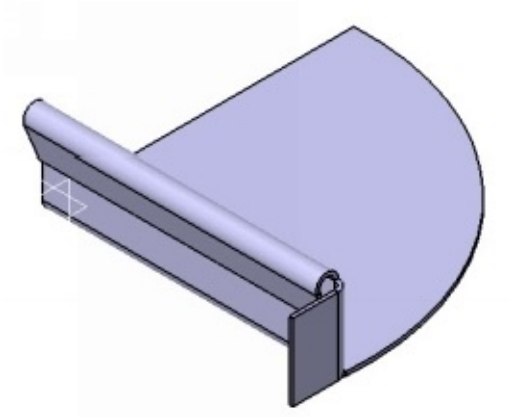




Tear Drop

This command folds and closes a sheet metal edge.

1. On the **Walls** toolbar, click **Swept Walls** drop-down > **Tear Drop** (or) click **Insert** > **Walls** > **Swept Walls** > **Tear Drop** on the Menu bar.
2. Select the edge you need to fold over.
3. Define the length of the inclined face.

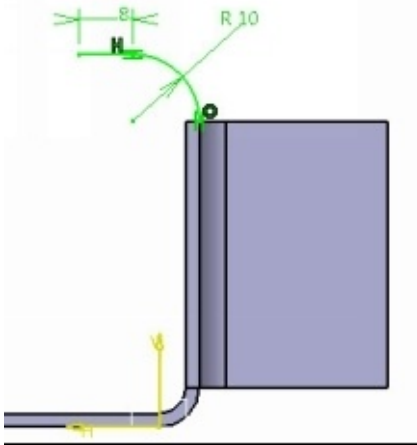




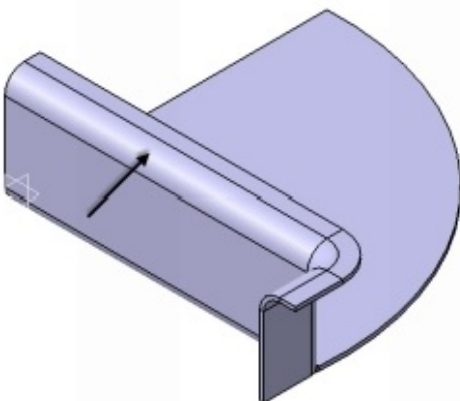
User Flange

This command sweeps a user defined profile along the selected edge. You have to ensure that the profile is tangent to the selected edge.

1. On the **Walls** toolbar, click **Swept Walls** drop-down > **User Flange** (or) click **Insert > Walls > Swept Walls > User Flange** on the Menu bar.
2. Click on an edge of the sheet metal.
3. Click the **Propagate** button to select the edges, which are connected tangentially.
4. On the dialog, click the sketch icon and select a face or plane normal to the select edge.
5. Draw a sketch, which is tangent to the support. Exit the sketch.



6. Click **OK** to create the flange.

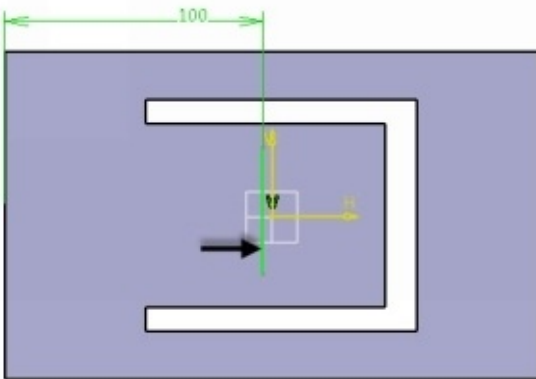




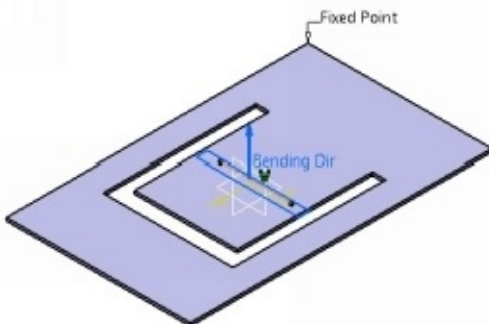
Bend From Flat

In addition to adding flanges and user flanges, you can also bend a flat sheet using the **Bend From Flat** command.

1. On the **Bending** toolbar, click **Bend From Flat** (or) click **Insert > Bending > Bend From Flat** on the Menu bar.
2. On the dialog, click the sketch icon, and then click on the face to bend.
3. Draw a sketch line on the flat sheet and click **Exit workbench**.



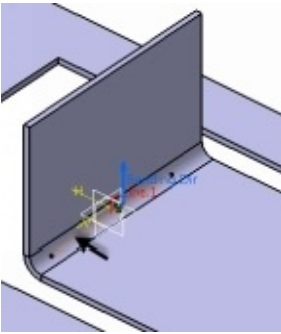
4. Click in the **Fixed Point** box and select a point to define the portion to be fixed.



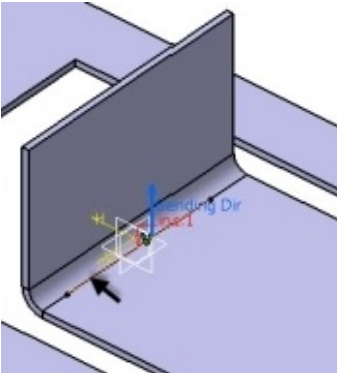
5. Click the blue arrow that appears on the bend, if you want to reverse the bend direction.
6. Type-in a value in the **Angle** box to change the folding angle.
7. Select the option to define the material side of the bend feature. These options are given next.



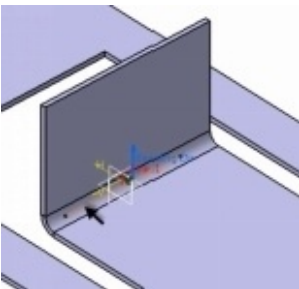
This creates the bend with the bend line at its center.



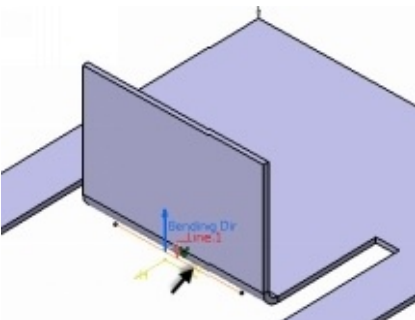
This creates the bend with the bend line at its starting edge.



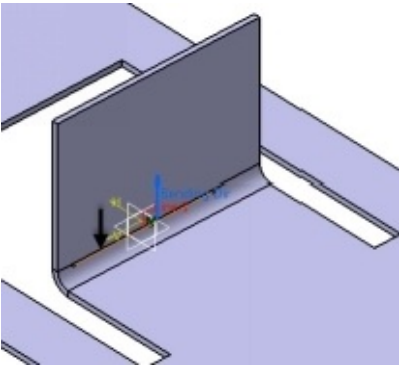
This creates the bend with the bend line at the intersection of the inner faces of the sheet metal part.



This creates the bend with the bend line at the intersection of outer faces.



This creates the bend with the bendline at its ending edge.

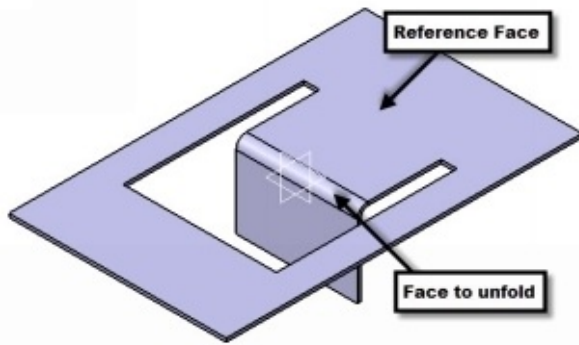


8. Click **OK** to complete the bend feature.

Unfolding

This command unfolds a bend to its original position.

1. On the **Bending** toolbar, click **Unfolding/Folding** drop-down > **Unfolding** (or) click **Insert** > **Bending** > **Unfolding** on the Menu bar.
2. Select the reference face and face to unfold.



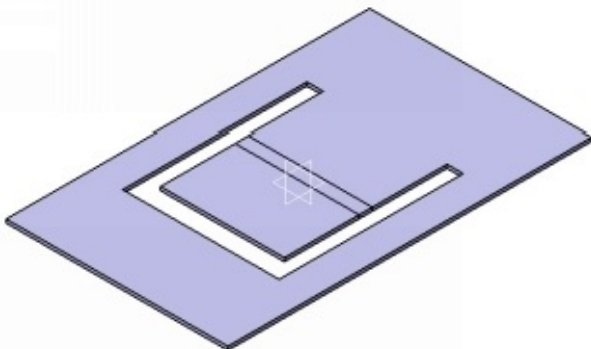
3. Select the **Angle** type. The different angle types are given next.

Natural: This option unfolds the bend to its original position.

Defined: This option unfolds the bend by the angle that you specify. The angle should not be that of the original position.

Spring back: This option unfolds the bend by using the angle that you specify. The angle is measured from its current position.

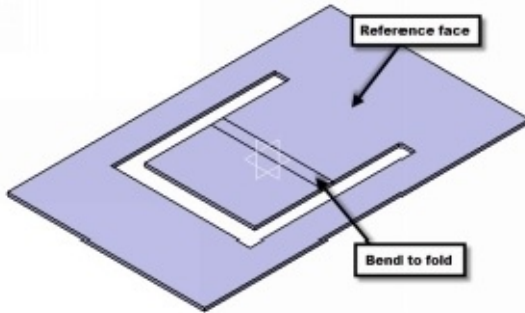
4. Click **OK** to unfold the bend.



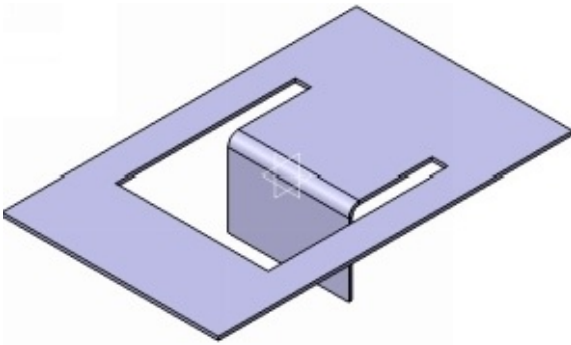
Folding

This command refolds the unfolded bend.

1. On the **Bending** toolbar, click **Unfolding/Folding** drop-down > **Folding** (or) click **Insert** > **Bending** > **Folding** on the Menu bar.
2. Select the reference face and face to refold.



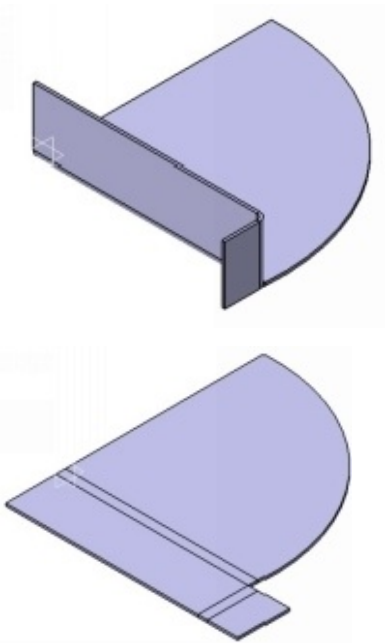
3. Select the **Angle** type, and then specify the bend angle.
4. Click **OK**.



Fold/Unfold

The **Fold/Unfold** command flattens the part so that you can easily display the manufacturing information.

1. On the **Views** toolbar, click **Fold/Unfold** drop-down > **Fold/Unfold** (or) click **Insert > Views > Fold/Unfold** on the Menu bar. This flattens the entire sheet metal part.



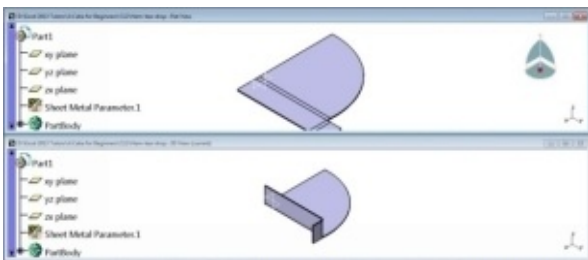
2. Again, click the **Fold/Unfold** button to refold the sheet metal part.



Multi Viewer

The **Multi Viewer** command lets you view the flattened view in a separate window.

1. On the **Views** toolbar, click **Fold/Unfold** drop-down > **Multi Viewer** (or) click **Insert > Views > Multi Viewer** on the Menu bar. A separate window will be displayed with the flattened view.
2. Select **Windows > Tile Horizontally** or **Tile Vertically** to view both the windows at a time.

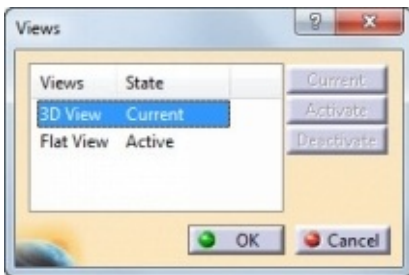




View Management

The **View Management** command lets you to switch between different views of the sheet metal part. You can also activate or deactivate views using this command.

1. On the **Views** toolbar, click the **View Management** button (or) click **Insert > Views > View Management** on the Menu bar.
2. On the **Views** dialog, select a view, and then click the **Current** button. This makes the selected view as current.



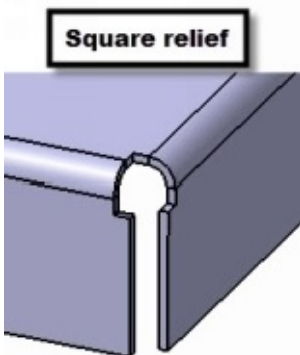
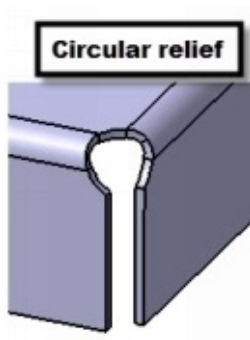
3. If you want to deactivate a view, then select it from the **Views** dialog, and then click the **Deactivate** button.



CornerRelief

The **CornerRelief** command allows you to control the appearance of sheet metal seams. For example, when two flanges meet at a corner, this command applies a relief.

1. On the **Cutting/Stamping** toolbar, click **CornerRelief** button (or) click **Insert > Cutting > CornerRelief** on the Menu bar.
2. On the **Corner Relief Definition** dialog, select **Type > Circular**. This option creates a circular relief. Select **Type > Square** to create a square corner relief.



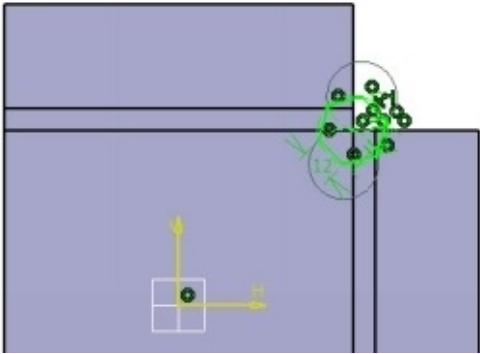
3. Select the bends forming the corner.
4. Type-in a value in the **Radius** box. In case of **Square** relief, type-in the **Length** value.
5. Click **OK**.

Corner Relief with User defined profile

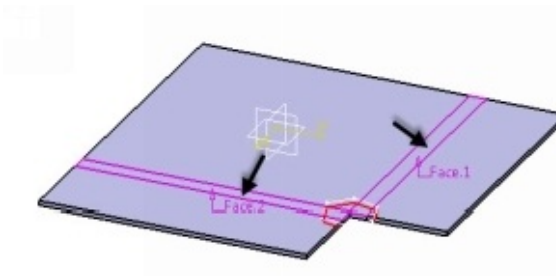
You can apply a corner relief with the user-defined profile.

1. Click the **Fold/Unfold** button on the **Views** toolbar. This changes the view to Flat view.
2. Activate the **CornerRelief** command.
3. On the dialog, select **Type > User Profile**.
4. Click the **Sketch** icon next to the **Profile** box.

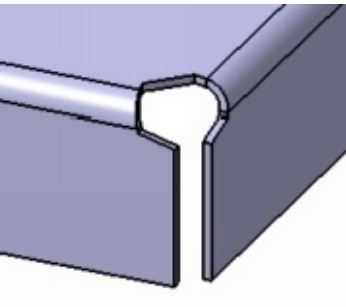
5. Click on the top face of the sheet metal part.
6. Draw a sketch at the corner, and then exit the workbench.



7. Select the bend faces forming a corner.



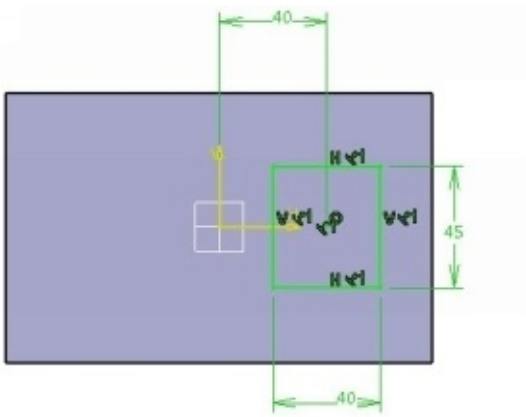
8. Click **OK** to create relief.
9. Click the **Fold/Unfold** button. This changes the view to 3D view.



Surface Stamp

The **Surface** command adds a stamp to a flat sheet by deforming it.

1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Surface Stamp** (or) click **Insert** > **Stamping** > **Surface Stamp** on the Menu bar.
2. On the dialog, click the **Sketch** icon next to the Profile box.
3. Click on the sheet metal face to add a surface stamp.
4. Draw a sketch and exit the workbench.

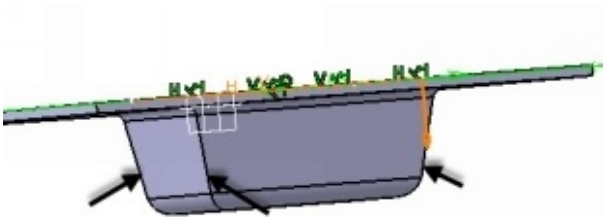


5. Select **Parameters choice** > **Angle**. This creates a stamp with a draft angle.

If you check the **Half pierce** option, the height of the stamp will be half of the sheet metal thickness. You can increase the height value up to the sheet metal thickness.

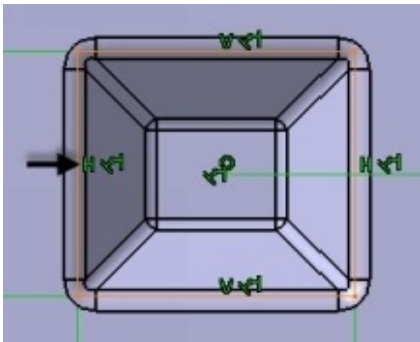
6. Specify the **Angle** and **Length** values. You can also define the length by using the **Limit** selection box. Click in this box and select a plane to define the length of the stamp.
7. Specify the **Radius R1** and **Radius R2** boxes. You can refer to the image available on the dialog to know **Radius R1** and **Radius R2**.

The **Rounded die** option rounds the edges of the sidewalls.

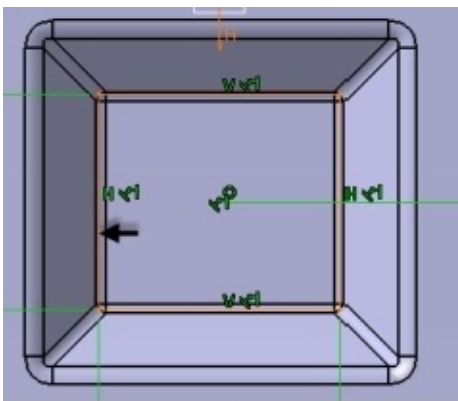




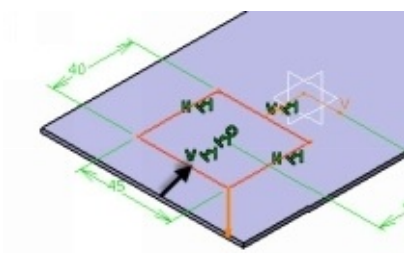
8. Click the **Upward sketch profile**  button (or) **Downward sketch profile**  button. The **Upward sketch profile** button uses the sketch to define the top face of the stamp.



The **Downward sketch profile** button uses the sketch to define the bottom face of the stamp.



If you want to remove any one of the sidewalls, then click in the **Opening Edges** box and select an edge from the sketch. The corresponding face will be removed.



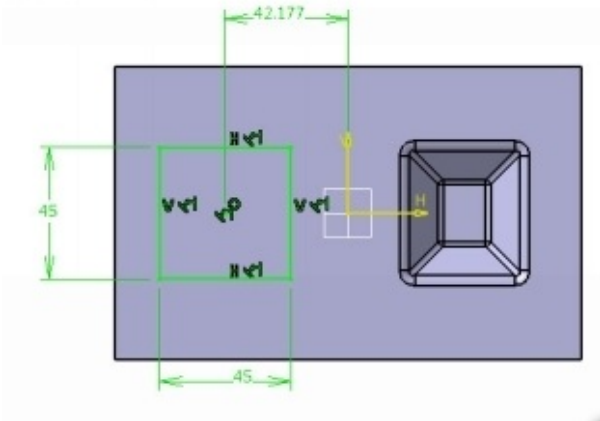


9. Click **OK** to complete the surface stamp.

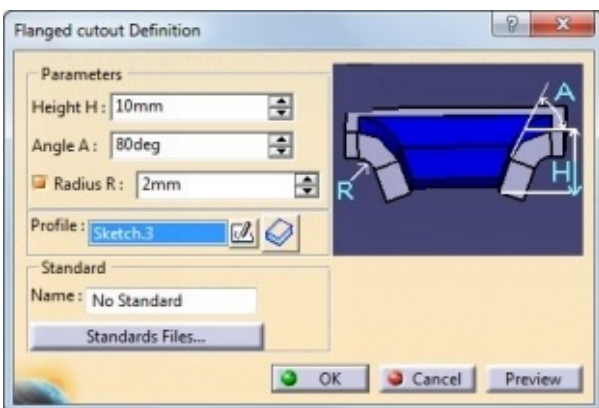
Flanged Cut Out

The flanged cut out and surface stamp feature are almost alike, except that an opening is created in case of flanged cut out.

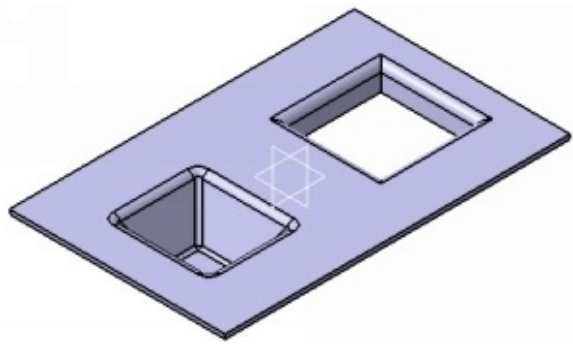
1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Flanged Cut Out** (or) click **Insert** > **Stamping** > **Flanged Cut Out** on the Menu bar.
2. On the dialog, click the **Sketch** icon, and then click a face to add flanged cut out.
3. Draw a closed sketch and click **Exit workbench**.



4. On the dialog, type-in a value in the **Height** box to define the height of the flanged cut out feature.
5. Type-in values of **Angle** and **Radius**.



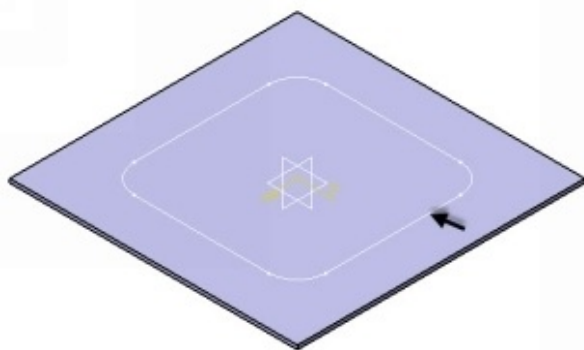
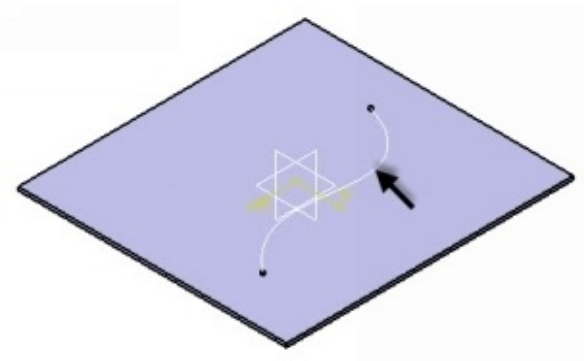
6. Click **OK** to create the flanged cut out.



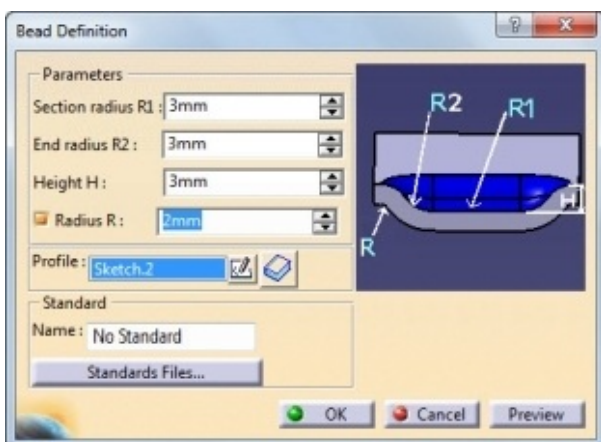
Bead

The **Bead** command creates a bead feature, which stiffens the sheet metal part. In order to create a bead feature, first you must have a sketch, which defines the bead size and shape. If the sketch is having curved edges, then ensure that they are tangent continuous.

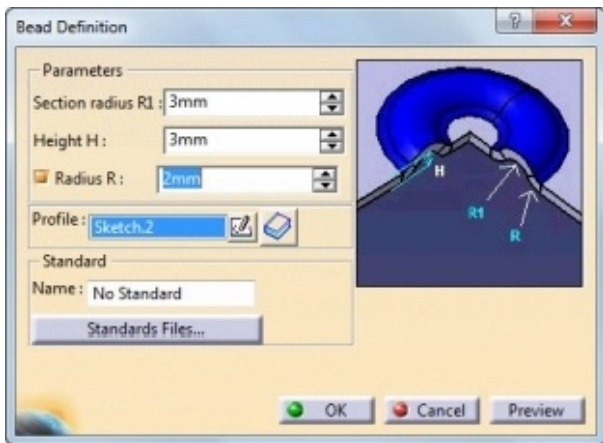
1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Bead** (or) click **Insert** > **Stamping** > **Bead** on the Menu bar.
2. Click on the open or closed sketch.



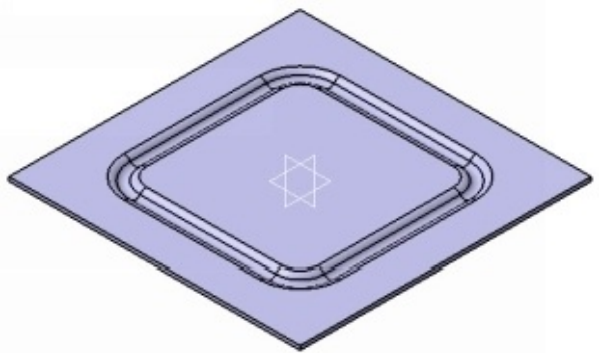
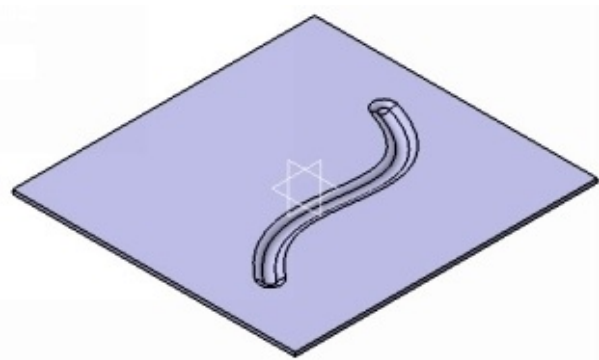
3. In case of an open profile, type-in the **Section radius**, **End radius**, **Height** and **Radius** values.



In case of a closed profile, type-in the **Section radius** and **Height** values.



4. Click **OK** to complete the bead feature.

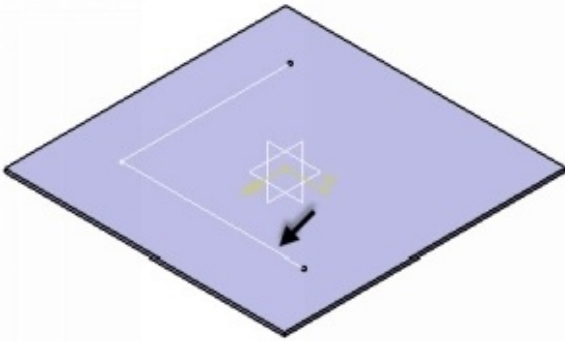




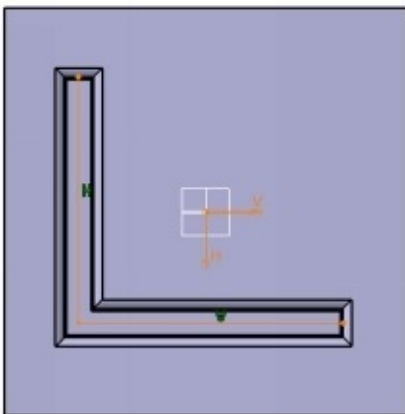
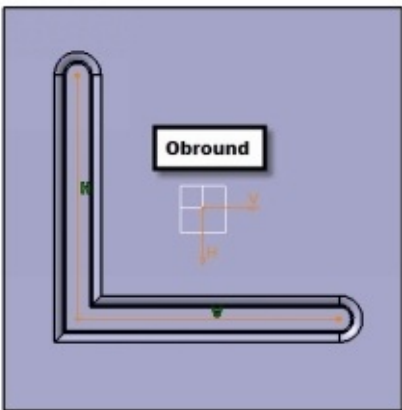
Curved Stamp

The **Curved Stamp** command creates a stamp using the sketch, which has non-tangent elements.

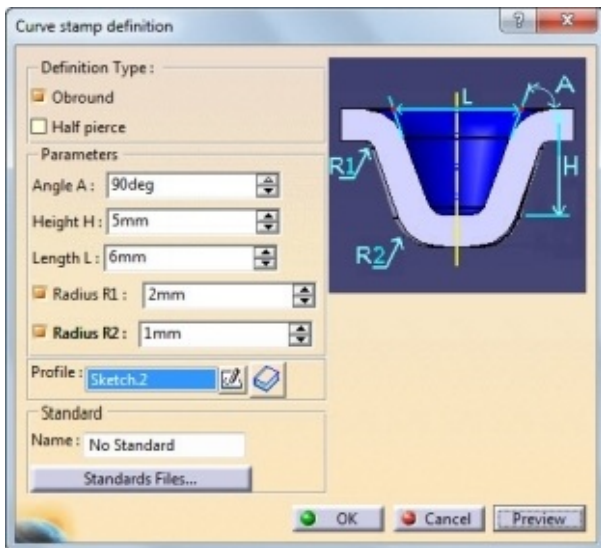
1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Curved Stamp** (or) click **Insert** > **Stamping** > **Curved Stamp** on the Menu bar.
2. Click on the sketch, which has linear elements.



3. On the dialog, check the **Obround** option to round the ends of the stamp. Uncheck this option to create a stamp with straight ends.



4. Type-in values in the **Parameters** section. The image on the dialog shows the parameters.

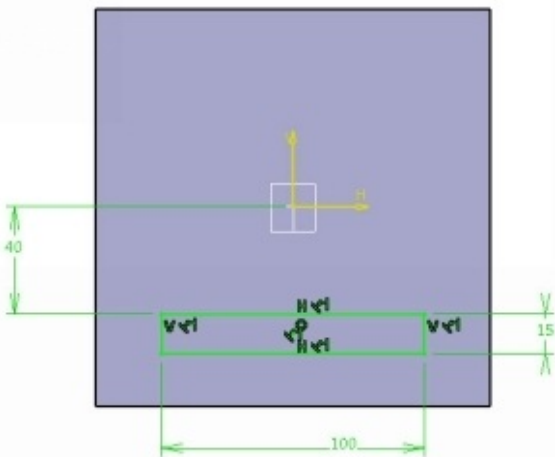


5. Click **OK** to complete the feature.

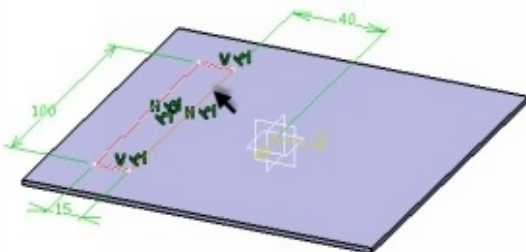
Louver

CATIA V5 provides you with the **Louver** command, which makes it easy to create louvers.

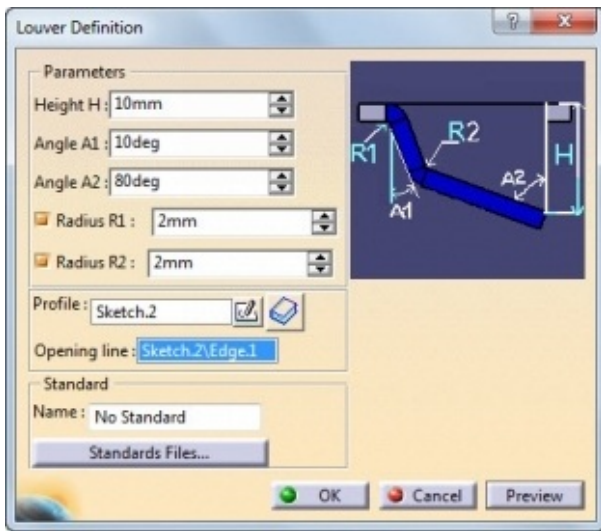
1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Louver** (or) click **Insert** > **Stamping** > **Louver** on the Menu bar.
2. On the dialog, click the sketch icon, and click on a face.
3. Draw a rectangle on the selected face and click **Exit workbench**.



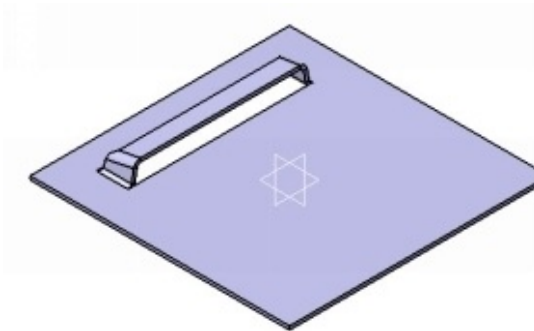
4. Click on a line of the sketch to define the opening edge.



5. On the **Louver Definition** dialog, type-in a value in the **Height** box. This defines the height of the louver.
6. Click the arrow that appears on the louver to reverse the direction.
7. Type-in a value in the **Angle A1** box, if you want an inclined side face.
8. Type-in a value other than 90 in the **Angle A2** box, if you want an inclined bottom/top face.
9. Type-in the **R1** and **R2** values. This rounds the edges of the louver.



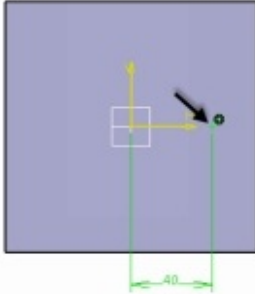
10. Click **OK**.



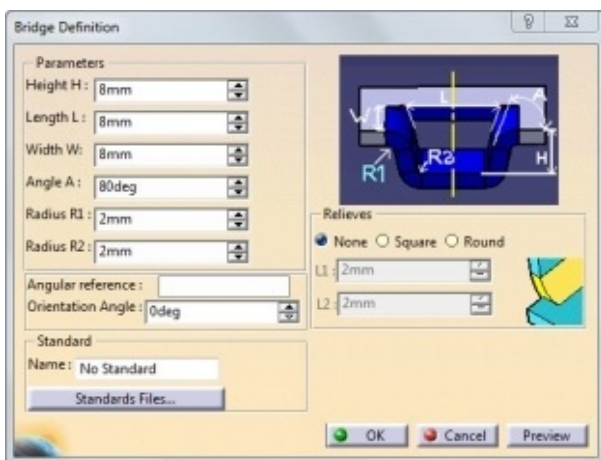
Bridge

This command creates a bridge, which is similar to the louver.

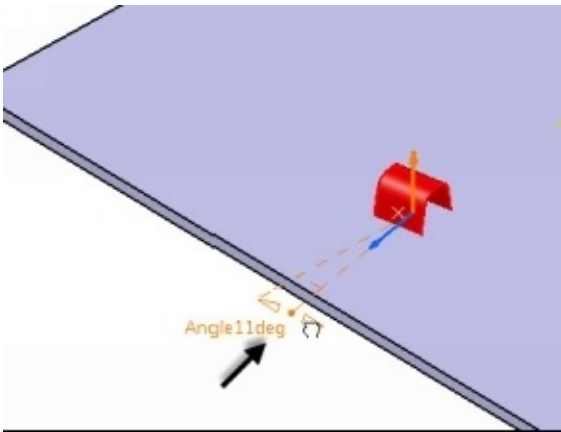
1. Create a point on the sheet metal to define the location of the bridge.



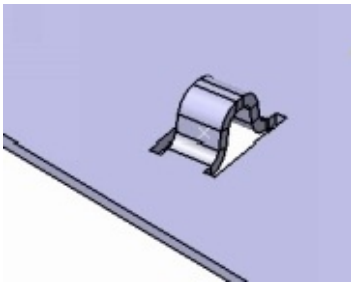
2. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Bridge** (or) click **Insert** > **Stamping** > **Bridge** on the Menu bar.
3. Select the point, and then click on the face to define the reference.
4. On the dialog, type-in values in the **Parameters** section. You can see the image displayed on the dialog to get a better understanding of these parameters.



5. Click the vertical arrow that appears on the bridge. This reverses the direction of the bridge.
6. Click and drag the orientation handle attached to the bridge. This changes the orientation angle of the bridge.



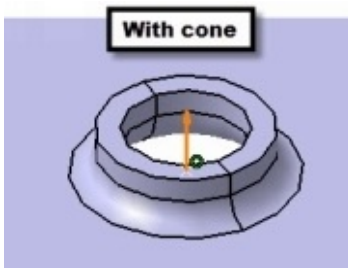
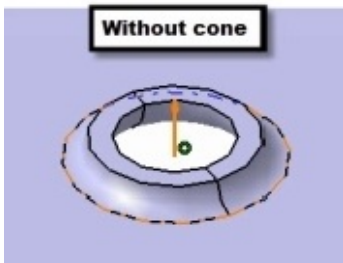
7. In the **Relieves** section, select the **Square** or **Round** option to apply relief at corners and sides.
8. Click **OK** to complete the feature.



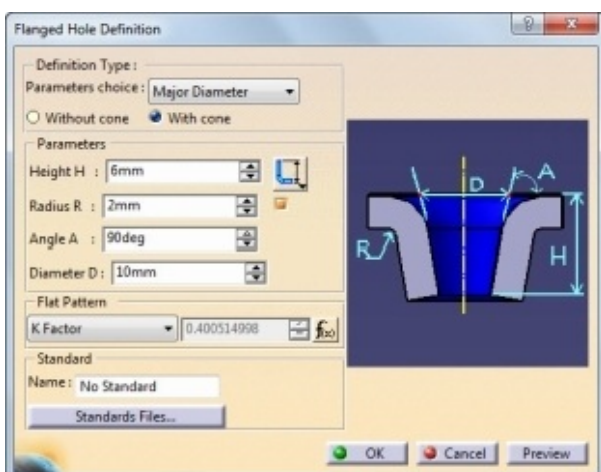
Flanged Hole

This command adds a circular flanged opening to the sheet metal part.

1. Create a point on the sheet metal to define the location of the flanged hole.
2. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Flanged Hole** (or) click **Insert** > **Stamping** > **Flanged Hole** on the Menu bar.
3. Select the point, and then click on the sheet metal face to define the reference.
4. On the dialog, select the **Parameters** choice. There are four parameter choices: **Major Diameter**, **Minor Diameter**, **Two diameters**, **Punch & Die**.
5. Select the **Without cone** or **With cone** option. The **Without cone** option creates the hole without the conical protrusion.



6. Type-in values in the **Parameters** section. Refer to the image located on the dialog to get a better understanding of the parameters.



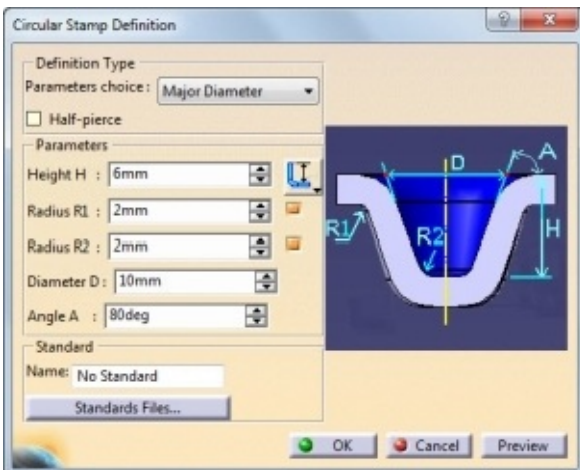
7. Click the arrow displayed on the flanged hole to reverse its direction.
8. Click **OK** to create the flanged hole

Circular Stamp

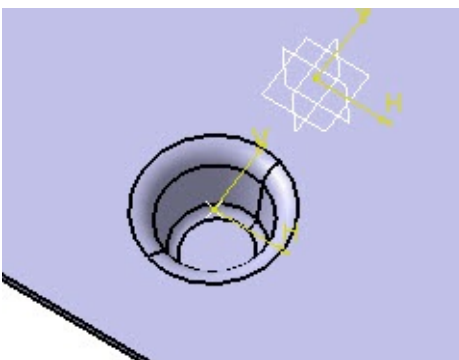
This command adds a circular stamp to the sheet metal part. This command is similar to the **Flanged Hole** command except that it creates a closed circular stamp.

1. Create a point on the sheet metal to define the location of the flanged hole.
2. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Circular Stamp** (or) click **Insert** > **Stamping** > **Circular Stamp** on the Menu bar.
3. Select the point, and then click on the sheet metal face to define the reference.

The options on the **Circular Stamp Definition** dialog are similar to the **Flanged Hole Definition** dialog.



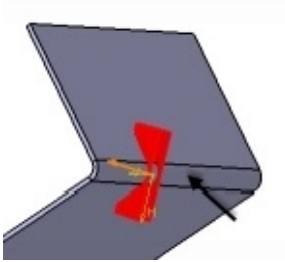
4. Click **OK**.



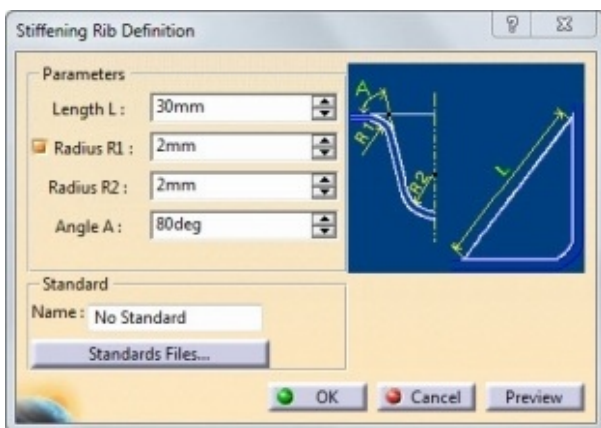
Stiffening Rib

Stiffening ribs are features created across a bend to reinforce the sheet metal part.

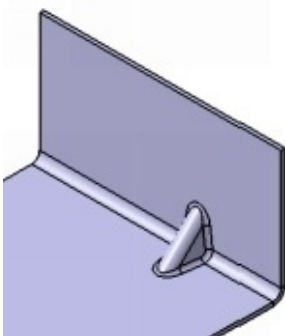
1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Stiffening Rib** (or) click **Insert** > **Stamping** > **Stiffening Rib** on the Menu bar.
2. Click on the external face of the bend. This defines the position of the bend.



3. On the dialog, type-in values in the **Parameters** section.



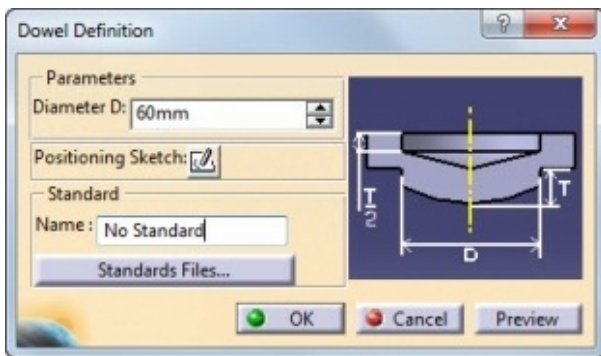
4. Click **OK**.



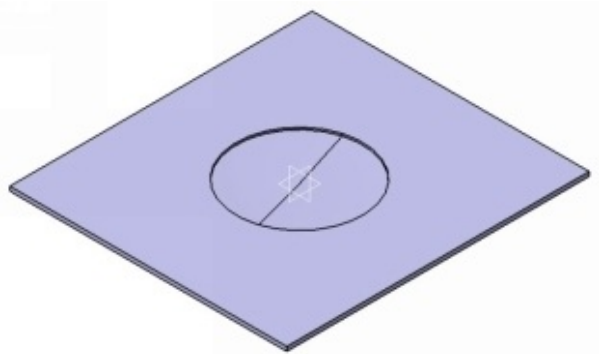
Dowel Stamp

This command creates a stamp resembling a dowel end.

1. On the **Cutting/Stamping** toolbar, click **Stamping** drop-down > **Dowel Stamp** (or) click **Insert** > **Stamping** > **Dowel Stamp** on the Menu bar.
2. Click on the face to add stamp.
3. On the dialog, click the **Positioned sketch** button, and then add dimensions to define the position of the stamp. Exit the sketch.
4. On the dialog, type-in a value in the **Diameter** box. This calculates the other parameters of the dowel, automatically.



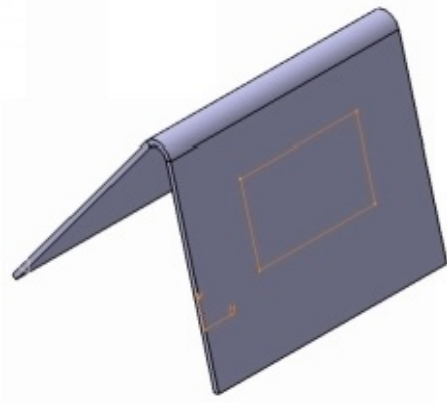
5. Click **OK**.



Cut out

When it is necessary to remove material from a sheet metal part, you must use the **Cut out** command.

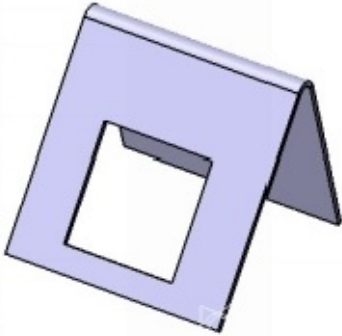
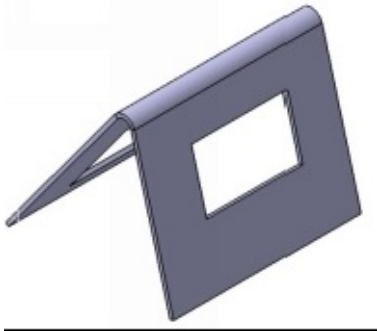
1. Draw a sketch, and then click the **Cut out** button on the **Cutting/Stamping** toolbar.
2. Select the sketch, if not selected.



3. On the dialog, click **Cutout Type > Sheetmetal standard**. This allows you to cut the sheet metal up to its entire depth. If you select **Cutout Type > Sheetmetal pocket**, cutout will be created only up to the thickness of the sheet.
4. Define the **End Limit**.



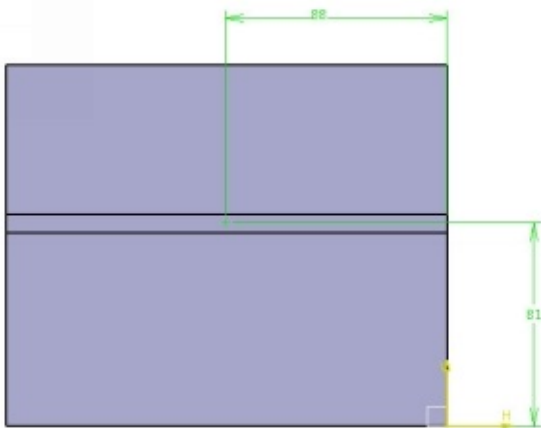
5. Click **OK**.



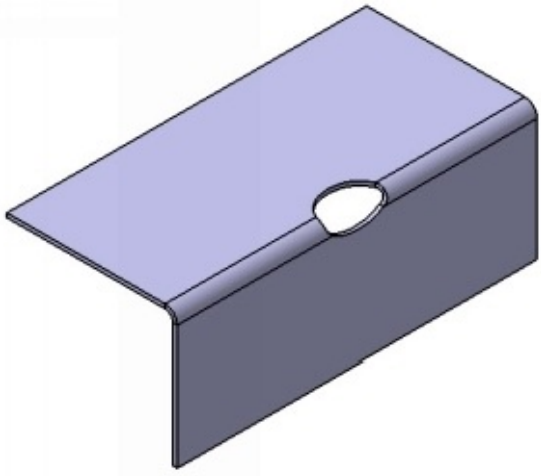
Circular Cutout

This command creates a circular cutout across a bend.

1. Create a point on the bend portion of the sheet metal part. To do this, you can unfold the sheet metal and create a sketch (or) use the reference **Point** command.



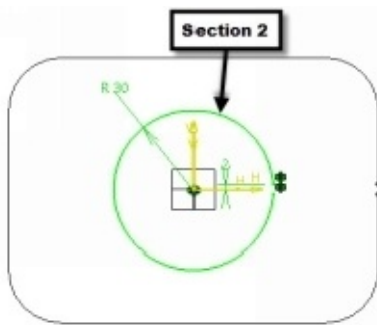
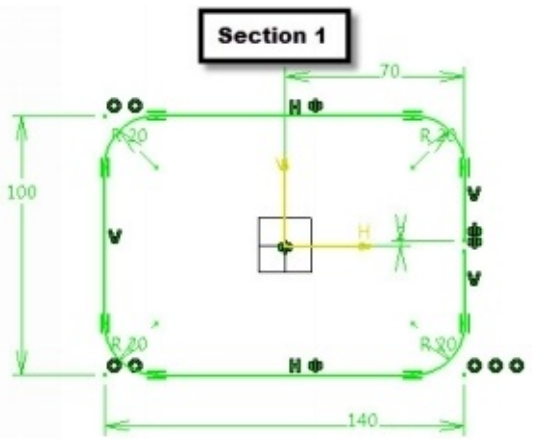
2. Refold the bend, if you have unfolded it.
3. On the **Cutting/Stamping** toolbar, click **Holes** drop-down > **Circular Cutout** (or) click **Insert** > **Cutting** > **Circular Cutout** on the Menu bar.
4. Select the point located on the bend.
5. On the dialog, type-in the **Diameter** value, and then click **OK**.



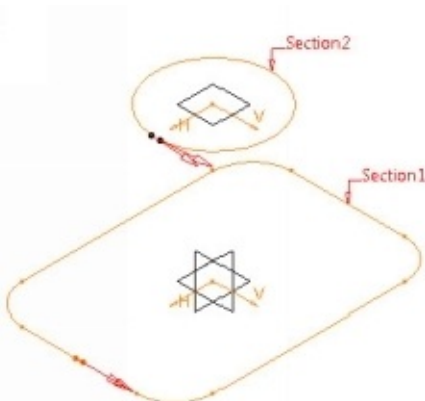
Hopper

The **Hopper** command creates a funnel that can be unfolded into flat pattern.

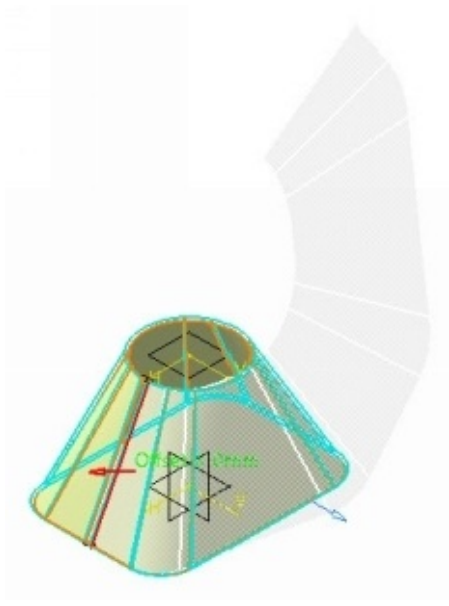
1. Create two sketches on planes parallel to each other. Ensure that the sketches do not have sharp edge and are open. In addition, the openings should be in the same direction.



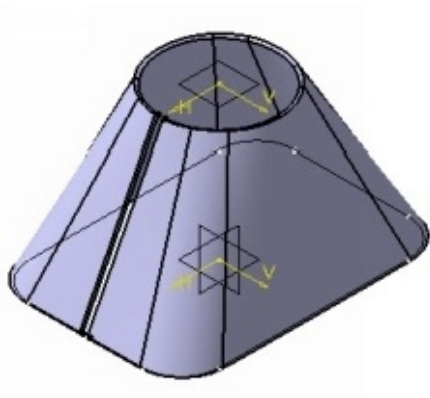
2. On the **Rolled Walls** toolbar, click the **Hopper** button (or) click **Insert > Rolled Walls > Hopper** on the Menu bar.
3. On the dialog, select the **Surfacic Hopper** option from the drop-down.
4. On the **Hopper** dialog, click the right mouse button in the **Selection** box, and then select **Create Multi-sections Surface**.
5. Select the first and second cross-sections.



6. Click **OK**.



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

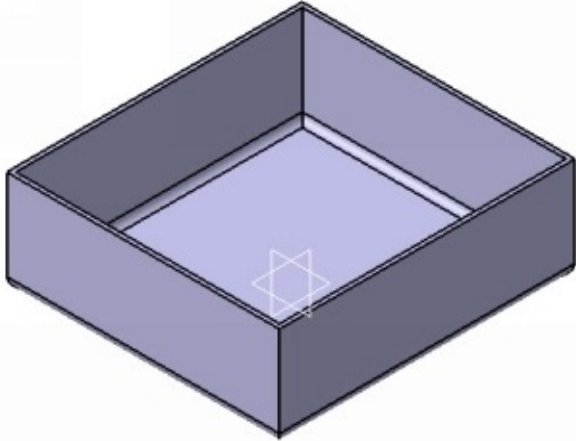




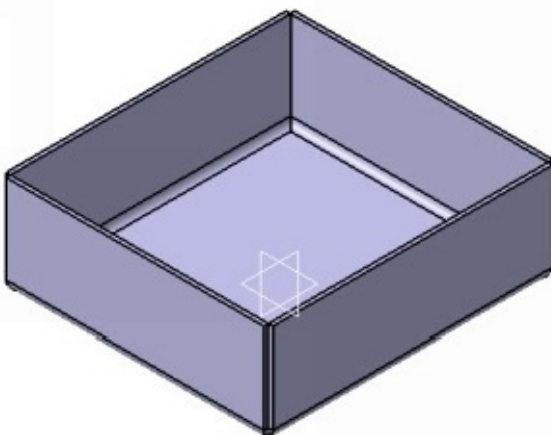
Recognize

CATIA V5 has a special command called **Recognize**, which automates the process of converting an already existing part into a sheet metal part.

1. Create a part in the **Part design** Workbench, and then shell it using the **Shell** command.



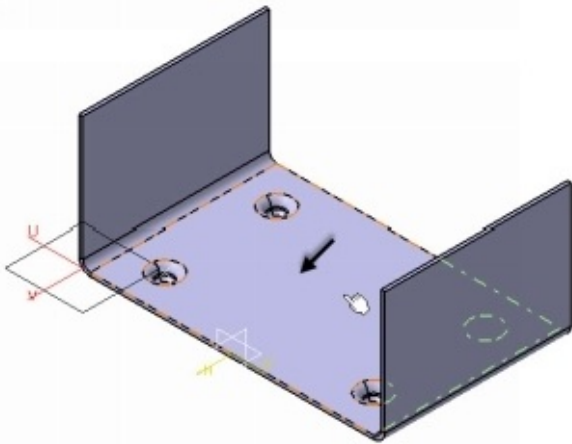
-
2. On the Menu bar, click **Start > Mechanical Design > Generative Sheetmetal Design**.
 3. On the **Walls** toolbar, click the **Recognize** button (or) click **Insert > Recognize** on the Menu bar.
 4. Click on the horizontal face to define the reference.
 5. Click **OK** to convert the part into a sheet metal part. Now, you can perform other sheet metal operations.



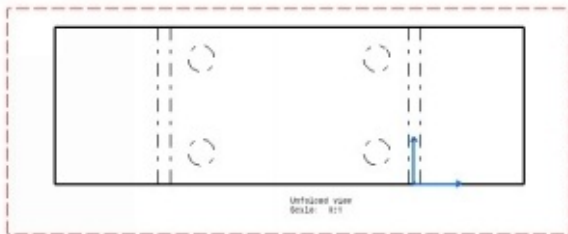
Sheet Metal Drawings

Creating drawing of a sheet metal part is same as creating any other drawing. However, you have to create the unfolded view of the sheet metal part. You can do this using the **Unfolded view** command.

1. Start a new drawing file.
2. On the **Views** toolbar, click **Projections** drop-down > **Unfolded view** (or) click **Insert > Views > Projections > Unfolded view** on the Menu bar.
3. Switch to the sheetmetal part window and select a face of the sheet metal part.



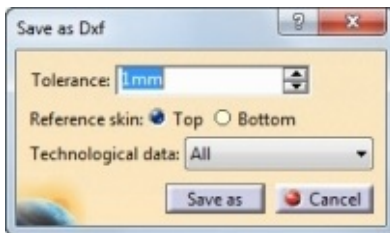
4. Click on the drawing sheet to place the unfolded view.



Save as DXF

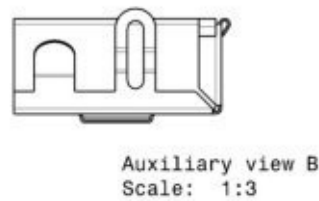
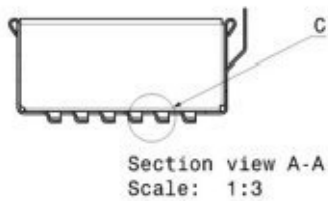
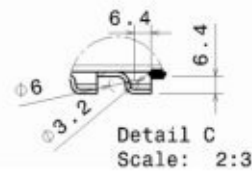
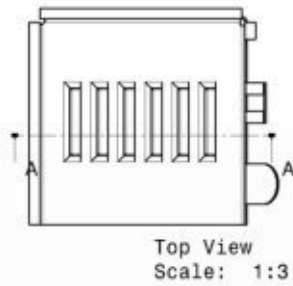
In addition to creating drawings, you can directly export a sheet metal to DWF format.


1. On the **Manufacturing Preparation** toolbar, click the **Save as DXF** button (or) click **Insert > Manufacturing Preparation > Save As DXF** on the Menu bar.
2. On the **Save as DXF** dialog, type-in the tolerance value.
3. Set the **Reference skin** to **Top** or **Bottom**.
4. Select the data that you want to export from the **Technological data** drop-down.

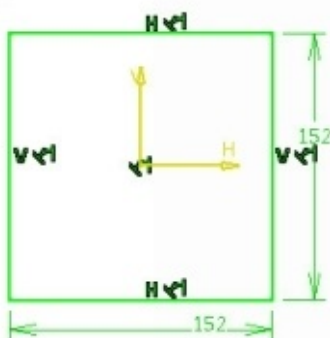



5. Click the **Save as** button, and specify the location of the file.
6. You can open the DXF file in AutoCAD or DXF viewer.



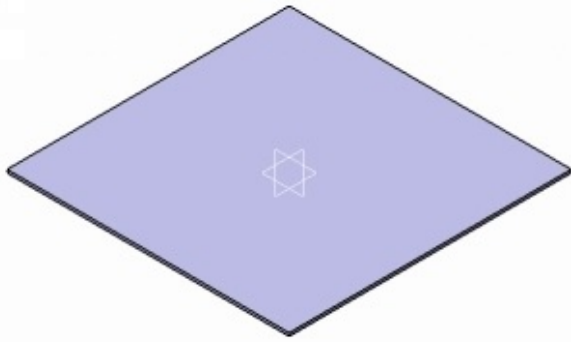




1. Start **CATIA V5-6R2015**.
2. On the Menu bar, click **Start > Mechanical Design > Generative Sheet Metal Design**.
3. On the **New Part** dialog, type-in C12-Example1 in the **Enter part name** box, and then click **OK**.
4. On the **Sheet Metal** toolbar, click the **Sheet Metal Parameters**  button (or) click **Insert > Sheet Metal Parameters** on the Menu bar.
5. On the **Sheet Metal Parameters** dialog, type-in 1.6 and 2.4 in the **Thickness** and **Default Bend Radius** boxes, respectively.
6. Click the **Bend Extremities** tab and select **Square relief** from the drop-down menu.
7. Type-in 1.2 and 2.4 in the **L1** and **L2** boxes, respectively.
8. Click **OK** to update the sheet metal parameters.
9. Create a sketch on the XY plane and exit the workbench.

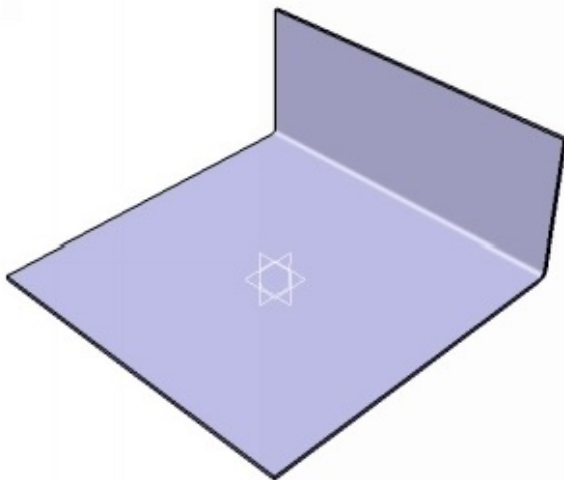


0. On the **Walls** toolbar, click the **Wall**  button (or) click **Insert > Walls > Wall** on the menu bar.

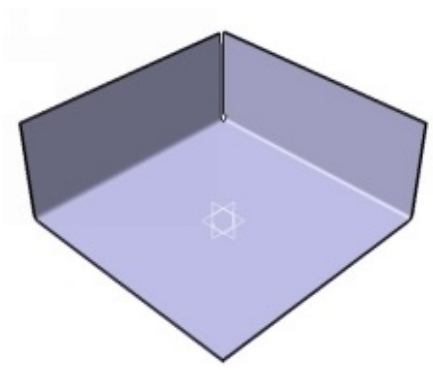
1. Click **OK** to create the sheet metal wall.



2. On the **Walls** toolbar, click the **Wall on Edge**  button (or) click **Insert > Walls > Wall on Edge** on the Menu bar.
3. Click on the back edge of the sheet metal wall.
4. On the **Wall on Edge Definition** dialog, type-in **65** in the **Height** box.
5. Set the **Length type** to outside .
6. Ensure that the red arrow on the preview is pointing outside. If not, click the **Invert Material Side** button.
7. Click **OK** to create the wall on edge.

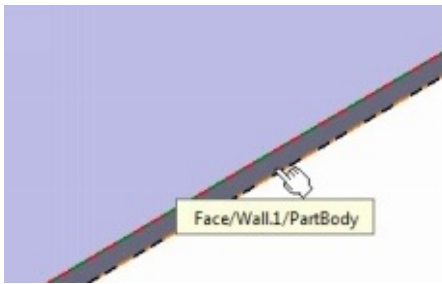


8. Create another wall on the left edge. The wall length is 65 mm.

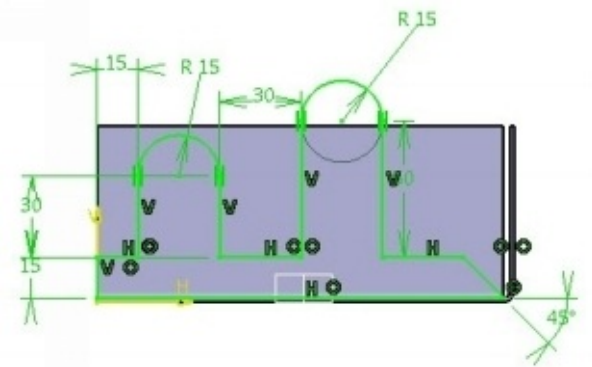


9. Activate the **Wall on Edge** command and select **Type > Sketch-Based** on the dialog.
0. Click on the right edge of sheet metal geometry to define the edge on which the wall will be created.

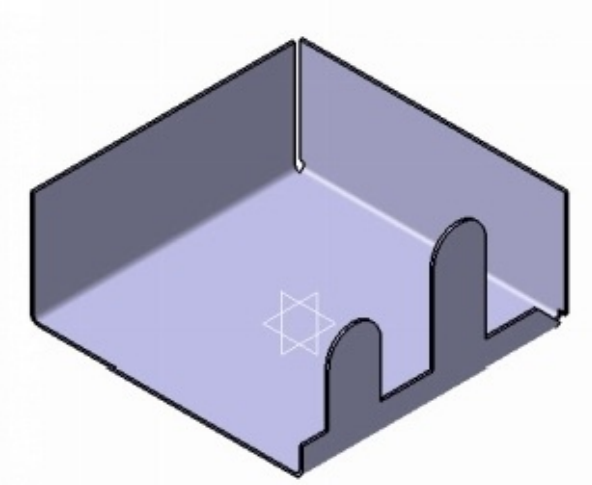
1. Click the sketch icon on the **Wall On Edge Definition** dialog and zoom into the model.
2. Click on the right end face of the model.



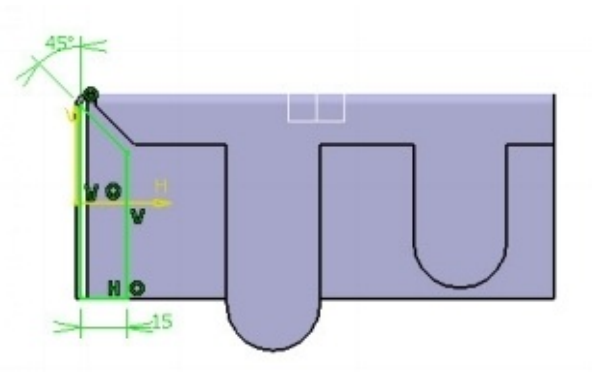
3. Draw the sketch of the wall and click **Exit workbench**.

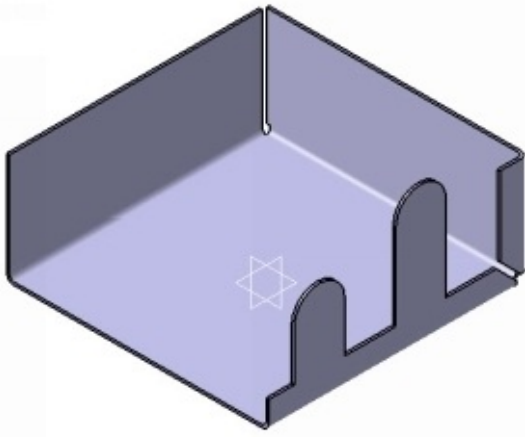


4. Click **OK** to create the sketch-based wall.

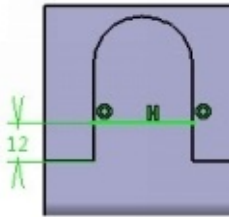



5. Likewise, create another wall on the vertical edge.

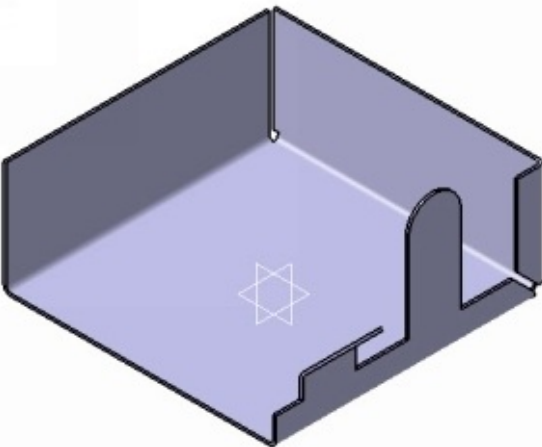




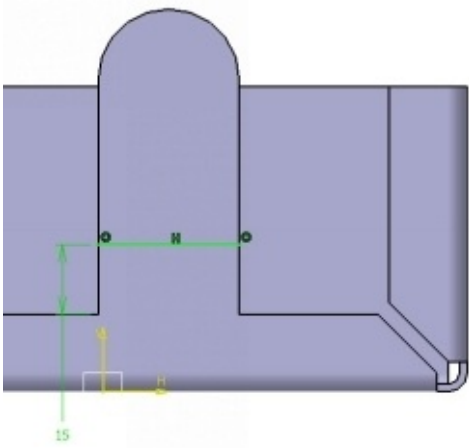
6. Draw a line on the outer face of the sketch-based wall and click **Exit workbench**.



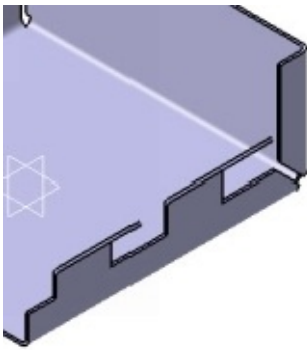
7. On the **Bending** toolbar, click the **Bend From Flat**  button (or) click **Insert > Bending > Bend From Flat** on the Menu bar.
8. On the **Bend From Flat Definition** dialog, type-in 135 in the **Angle** box and click **OK**. This bends the wall by using the sketched line.



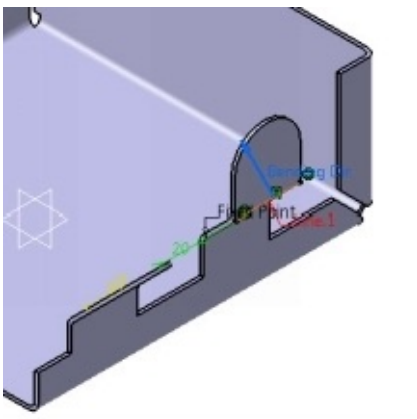
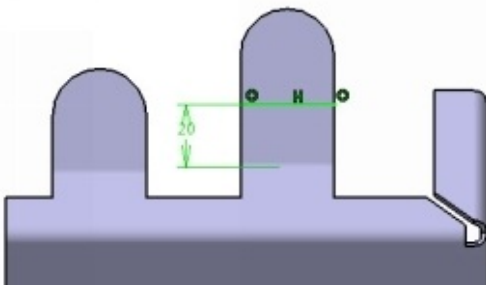
9. Sketch a horizontal line on the vertical wall and click **Exit workbench**.




0. Activate the **Bend From Flat** command and bend the wall using the sketched line. The bend angle is 135 degrees.

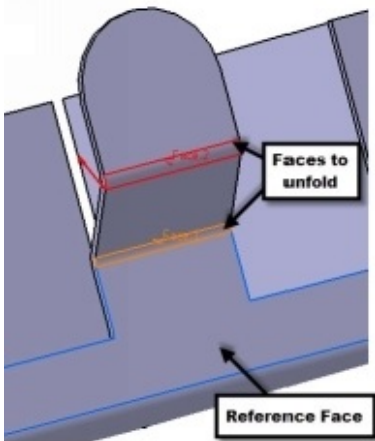


1. Sketch another horizontal line on the inclined face of the wall and bend it in the reverse direction. The bend angle is 135 degrees.

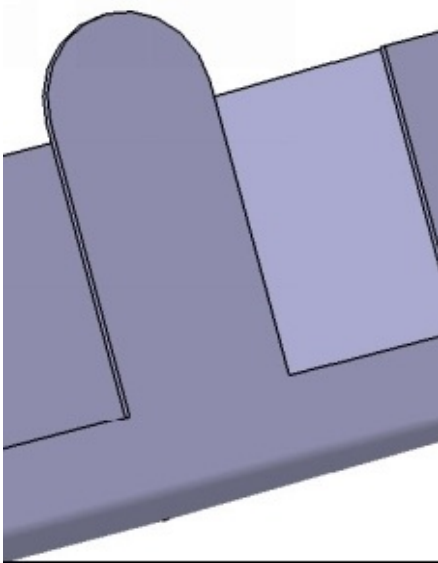


2. On the **Bending** toolbar, click **Folding/Unfolding** drop-down > **Unfolding**  (or) click **Insert** > **Bending** > **Unfolding** on the Menu bar.

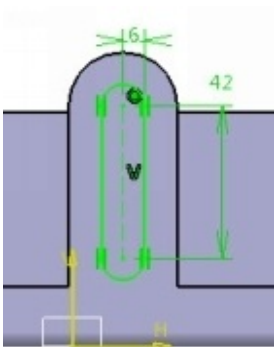
3. Click on the vertical face of the wall to define the reference face.
4. Click on the two bends to be unfolded.




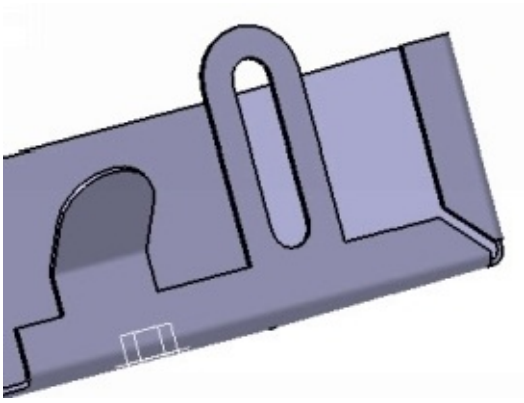
5. Click **OK** on the dialog.




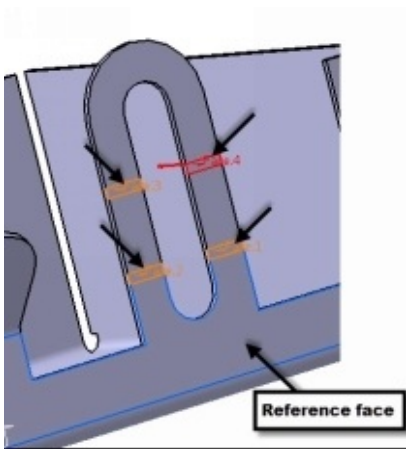
6. Draw a sketch on the unfolded face and click **Exit workbench**.



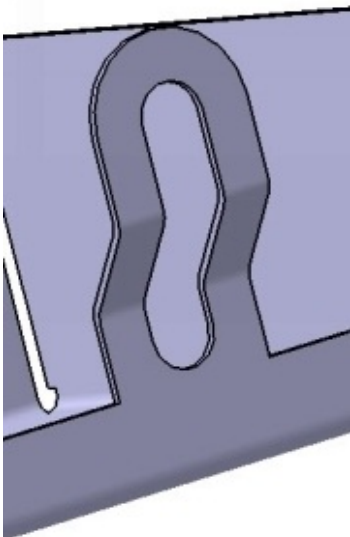
37. On the **Cutting/Stamping** toolbar, click **Cut Out**  button (or) click **Insert > Cutting > Cut Out** on the Menu bar.
38. Leave the default settings on the **Cutout Definition** dialog and click **OK**.




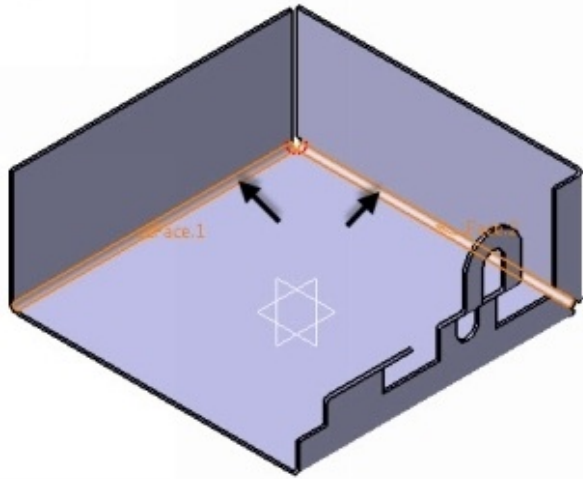
9. On the **Bending** toolbar, click **Folding/Unfolding** drop-down > **Folding**  (or) click **Insert** > **Bending** > **Folding** on the Menu bar.
0. Click on the vertical face of the wall to define the reference face.
1. Click on the unfolded bend faces to be folded.



2. Click **OK**.

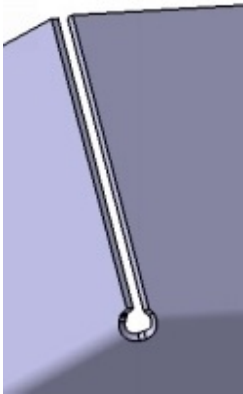



43. On the **Cutting/Stamping** toolbar, click **CornerRelief**  button (or) click **Insert** > **Cutting** > **CornerRelief** on the Menu bar.
44. Zoom into the model and click on the bends, as shown below.

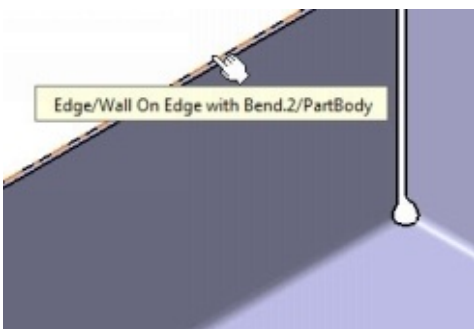


- 45. **Radius** box.
- 46.

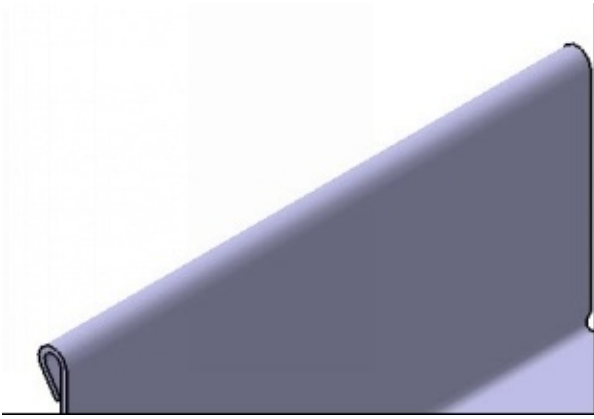
Select **Type** > **Circular** and type-in **4** in the
 Click **OK** to create the corner relief.



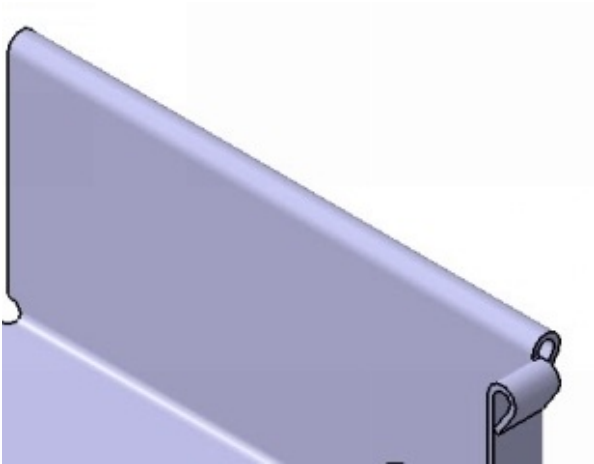
- 7. On the **Walls** toolbar, click **Swept Walls** drop-down > **Tear Drop**  (or) click **Insert** > **Walls** > **Swept Walls** > **Tear Drop** on the Menu bar.
- 8. On the **Tear Drop Definition** dialog, select **Basic** from the drop-down menu, and then type-in **8** in the **Length** box.
- 9. Click on the outer edge of the left-side wall.



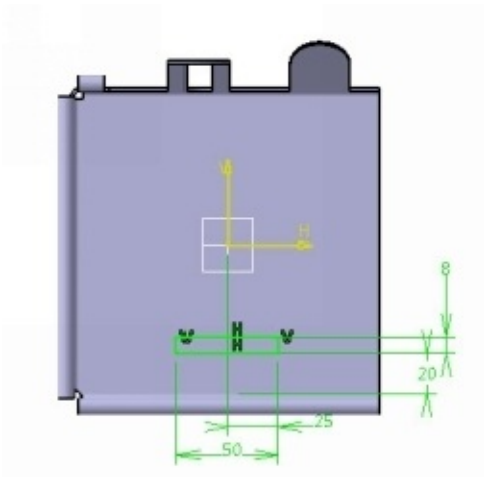
- 0. Click the **Reverse Direction** button on the dialog.
- 1. Click **OK** to create the teardrop.



2. Likewise, create teardrops on other vertical walls.




3. Draw a sketch on the top face of the base wall, and then click **Exit workbench**.

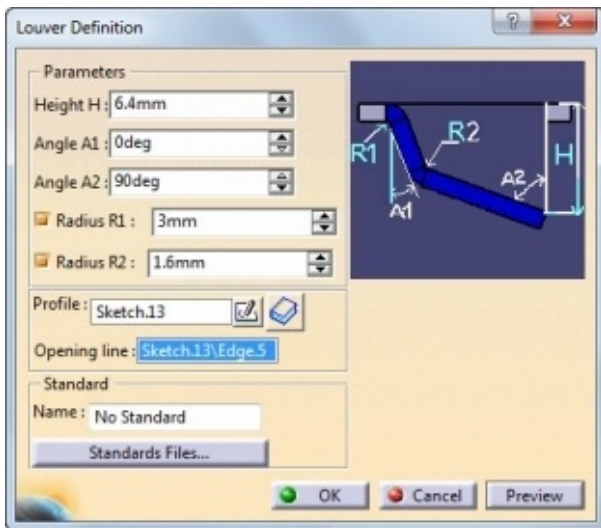


54.

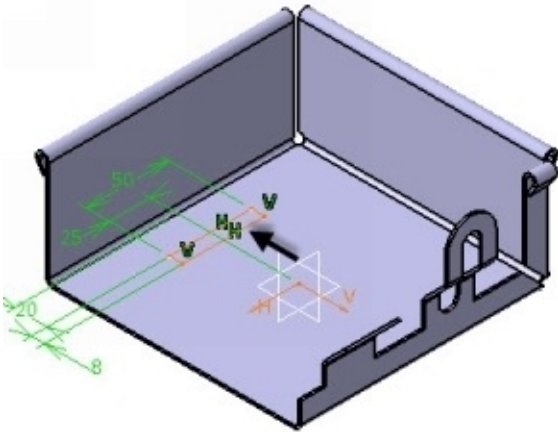
On the **Cutting/Stamping** toolbar, click

Stamping drop-down > **Louver**  (or) click **Insert** > **Stamping** > **Louver** on the Menu bar. You will notice that the sketch is selected, automatically. If not, you need to select the sketch.

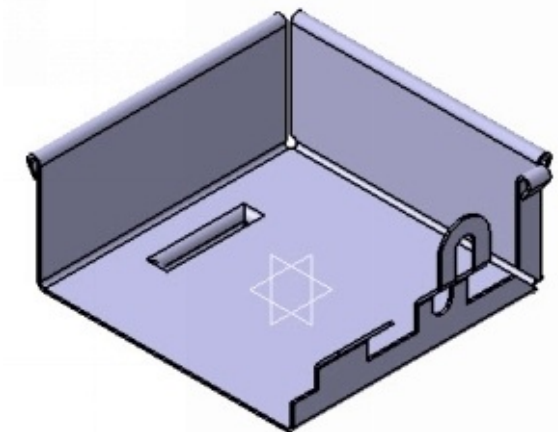
5. On the **Louver Definition** dialog, type-in values, as shown below.




6. Select the inner horizontal line of the sketch to define the opening line.

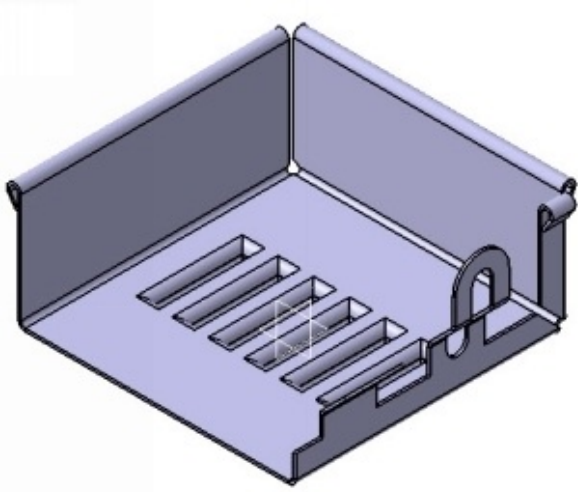



7. Click **OK** to create the louver.

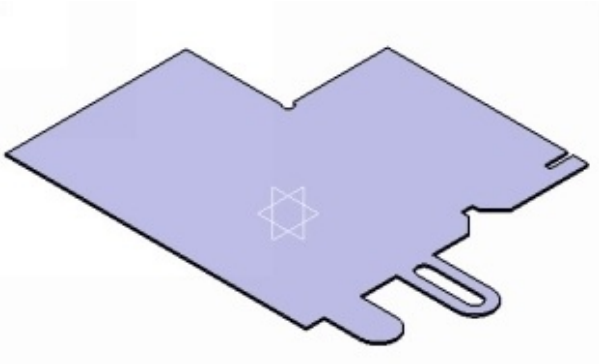



8. On the **Transformation** toolbar, click **Pattern** drop-down > **Rectangular Pattern**  (or) click **Insert** > **Transformations** > **Rectangular Pattern** on the Menu bar.
9. On the **Rectangular Pattern** dialog, click in the **Object** selection box and select the louver from the geometry.
0. Click in the **Reference element** selection box and click on the front edge of the geometry. This defines the first direction of the rectangular pattern.
1. On the dialog, select **Parameters** > **Instance(s) & Length**.

2. Type-in **6** and **100** in the **Instance(s)** and **Length** boxes, respectively.
3. Click **OK** to create the pattern.



4. On the **Views** toolbar, click **Fold/Unfold** drop-down > **Fold/Unfold**  (or) click **Insert > Views > Fold/Unfold** on the Menu bar. This displays the flattened view of the sheet metal part.



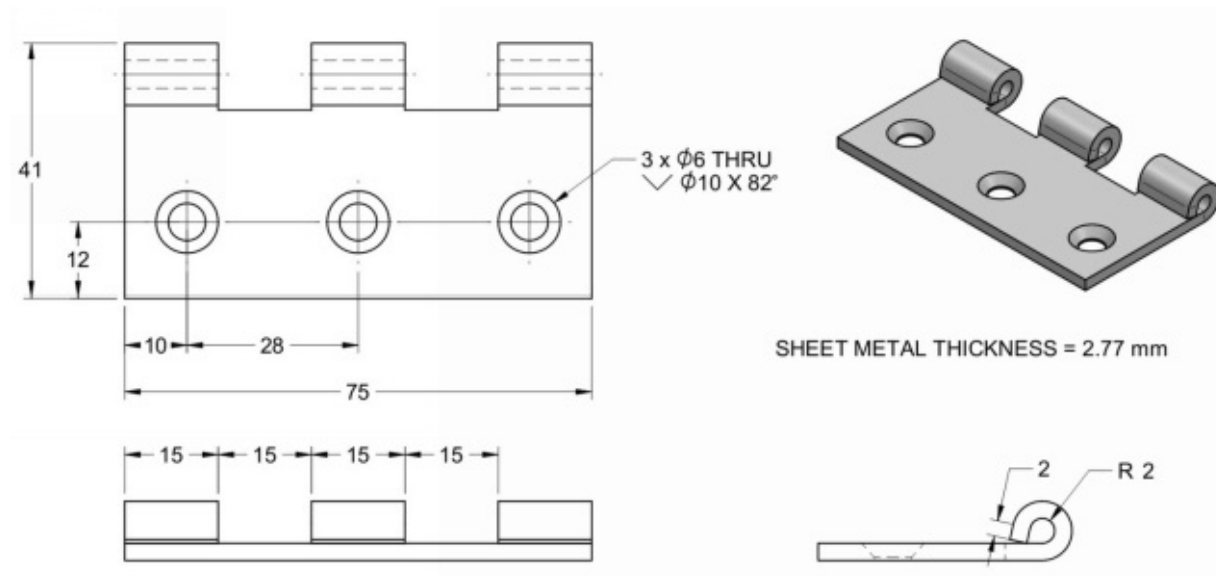
5. Again, click the **Fold/Unfold**  button to switch back to the folded view
6. Save and close the sheet metal part.

Questions

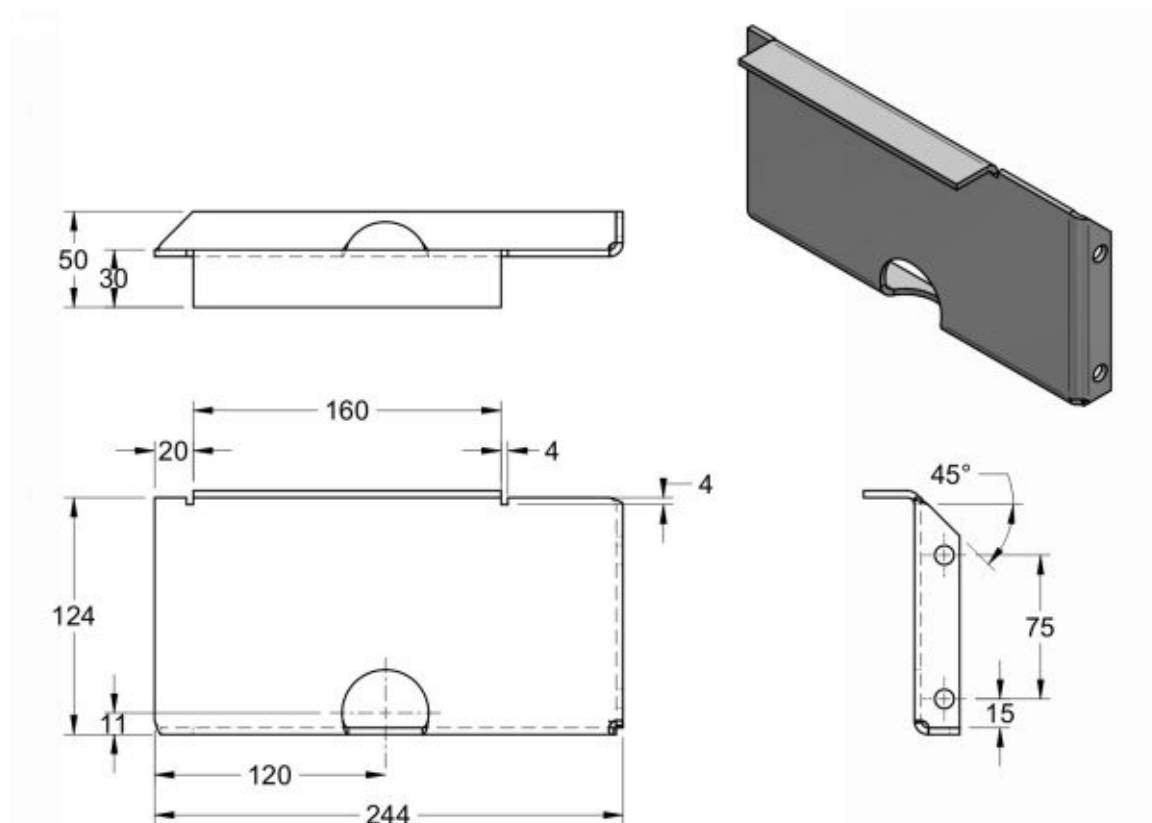
1. How do you insert an unfolded view into a drawing?
2. Describe parameters that can be specified on the **Sheet Metal Parameters** dialog.
3. Define the term 'K Factor'.
4. List any two sheet metal part parameters that can be overridden when creating a feature.
5. What is the use of the **Circular Cutout** command?
6. List the types of swept walls that can be created in CATIA V5?
7. What does the **CornerRelief** command do?
8. What are the corner relief types available?
9. What is the difference between a surface stamp and flanged cut out?

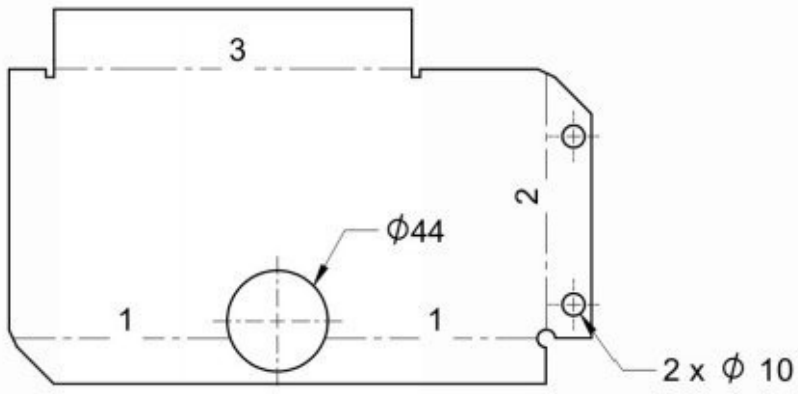
Exercises

Exercise 1



Exercise 2





Sequence	Feature	Radius	Angle	Direction	Included Angle
1	Bend 1	3.58 mm	90.00 deg	Down	90.00 deg
2	Bend 2	3.58 mm	90.00 deg	Down	90.00 deg
3	Bend 3	3.58 mm	90.00 deg	Up	90.00 deg

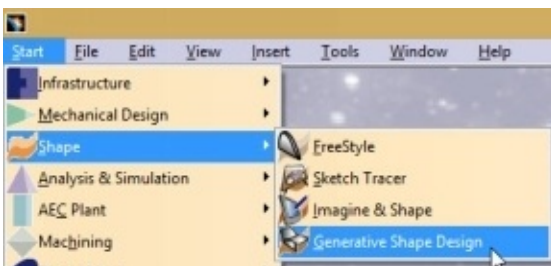
Chapter 13: Surface Design

The topics covered in this chapter are:

- *Basic surfaces*
- *Sweep surfaces*
- *Multi-sections surfaces*
- *Blend surfaces*
- *Fill surfaces*
- *Offset Surfaces*
- *Healing*
- *Extract*
- *Trim*
- *Split*
- *Extrapolate*
- *Untrim*
- *Join*
- *Translate*
- *Rotate*
- *Symmetry*
- *Split (Body)*
- *Thick Surface*
- *Close Surface*
- *Wireframe Geometry*

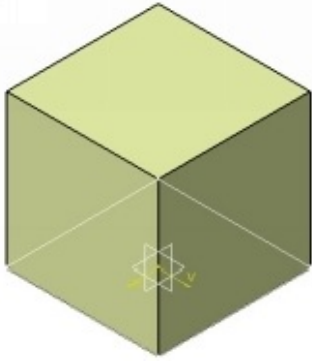
CATIA V5 Surfacing commands can be used to create complex geometries that are very difficult to create using standard pads, revolve features, and so on. Surface modeling can also be used to edit and fix the broken imported parts. In this chapter, you learn the basics of surfacing commands that are mostly used. The surfacing commands are available in the **Generative Shape Design** workbench.

To activate this workbench, click **Start > Shape > Generative Shape Design** on the Menu bar.



CATIA V5 offers a rich set of surface design commands. A surface is an infinitely thin piece of geometry. For example, consider a cube shown in figure. It has six faces. Each of

these faces is a surface, an infinitely thin piece of geometry that acts as a boundary in the 3D space. Surfaces can be simple or complex shapes.

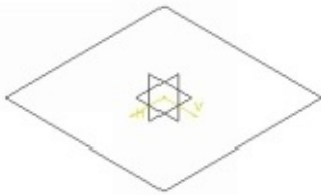


In solid modeling, when you have created solid features such as a pad feature or a Revolved feature, CATIA V5 creates a set of features (surfaces) that enclose a volume. The airtight enclosure is considered as a solid body. The advantage of using the surfacing commands is that you can design a model with more flexibility.

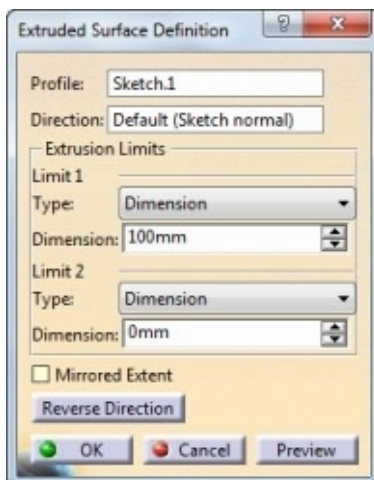


Extrude

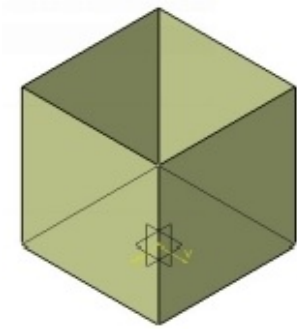
1. To create an extruded surface, first create an open or closed sketch.



2. On the **Surfaces** toolbar, click the **Extrude** button (or) click **Insert > Surfaces > Extrude** on the Menu bar.
3. Select the sketch and type-in a value in the **Dimension** box available below the **Type** drop-down.

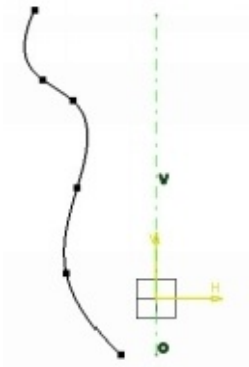


4. Click **OK** to create the extruded surface. You will notice that the extrusion is not capped at the ends.

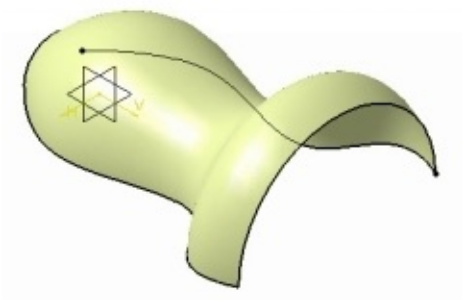


Revolve

1. To create a revolved surface, first create an open or closed profile and the axis of revolution.

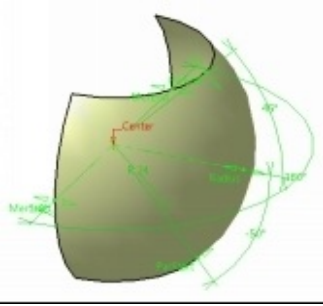


2. On the **Surfaces** toolbar, click **Extrude-Revolution** drop-down > **Revolve** (or) click **Insert** > **Surfaces** > **Revolve** on the Menu bar.
3. Select the sketch.
4. Type-in the angle of revolution in the **Angle 1** box and click **OK**.




Sphere

1. To create a sphere, first create a point to define its position.
2. On the **Surfaces** toolbar, click **Extrude-Revolution** drop-down > **Revolve** (or) click **Insert** > **Surfaces** > **Revolve** on the Menu bar.
3. Select the point. You can also create a point by right clicking in the **Center** selection box on the **Sphere Surface Definition** dialog, and selecting an option to create a point.
4. Type-in a value in the **Sphere radius** box (or) click and drag the radius handle that appears on the preview.



5. Type-in angle values on the dialog (or) click and drag the angle handles.



6. Click the whole sphere  button on the dialog, if you want to create a whole sphere.

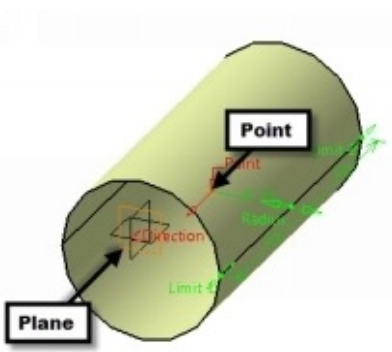
7. Click **OK** to create the sphere.

Cylinder

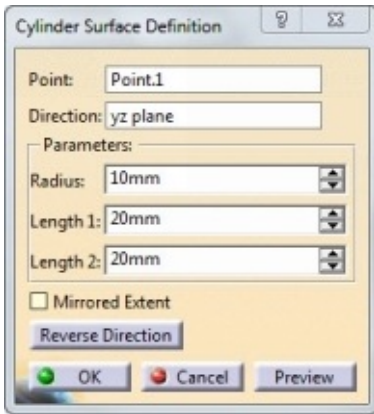
1. To create a cylinder, first create a point to define its position.

2. On the **Surfaces** toolbar, click **Extrude-Revolution** drop-down > **Cylinder** (or) click **Insert** > **Surfaces** > **Cylinder** on the Menu bar.

3. Select the point, and then click on a plane or axis. This defines the position and direction of the cylinder.



4. On the dialog, type-in values in the **Radius**, **Length 1**, **Length 2** boxes (or) using the handles to define the size of the cylinder.



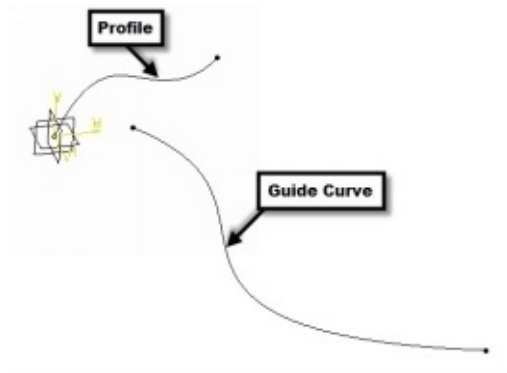
5. Click **OK** to complete the cylindrical surface.

Even if you create an enclosed surface, CATIA V5 will not recognize it as a solid body. You will learn to convert a surface body into a solid later in this chapter.

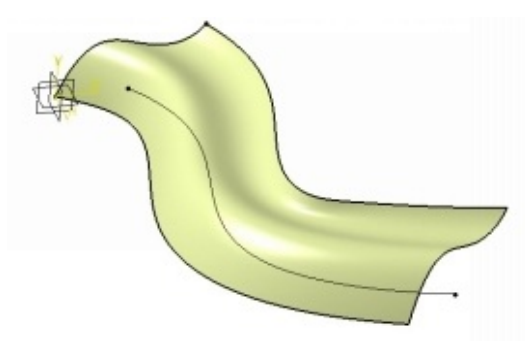
Sweep

This command creates a surface by sweeping a section along a guide curve.

1. Create a sweep profile and a guide curve.




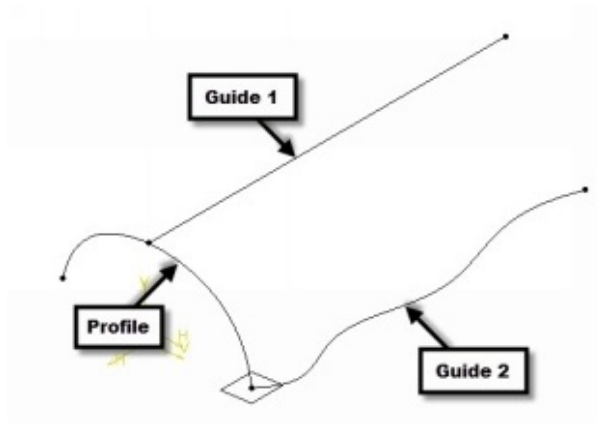
2. On the **Surfaces** toolbar, click **Sweeps** drop-down > **Sweep** (or) click **Insert** > **Surfaces** > **Sweep** on the Menu bar.
3. Click on the profile curve, and then click on the guide curve.
4. Click **OK**.



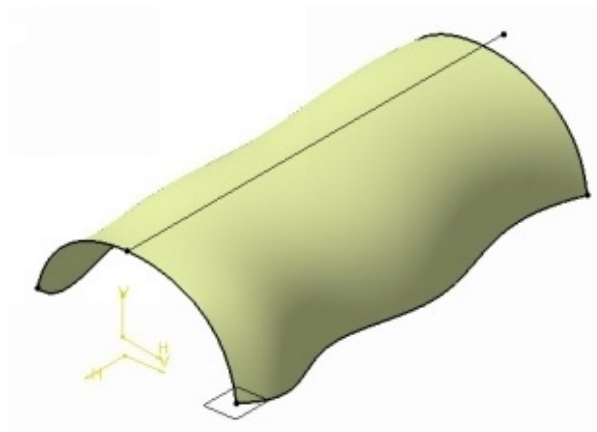
Sweep with two guide curves

The **With two guide curves** option creates a surface by sweeping a profile along two guide curves.

1. Create a profile and two guides. They should be separate sketches.
2. Activate the **Sweep**  command.
3. On the dialog, select **Subtype** > **With two guide curves**.
4. Select the profile and two guide curves.




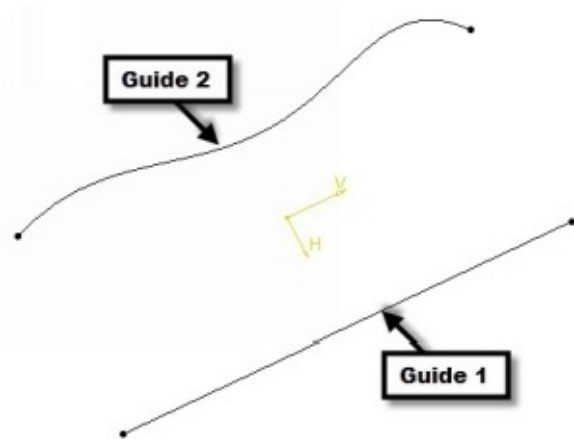
5. Select **Anchoring type** > **Two points**.
6. Click in the **Anchor point 1** box and select the intersection point between the first guide and profile.
7. Likewise, select the second anchor point.
8. Click **OK**.



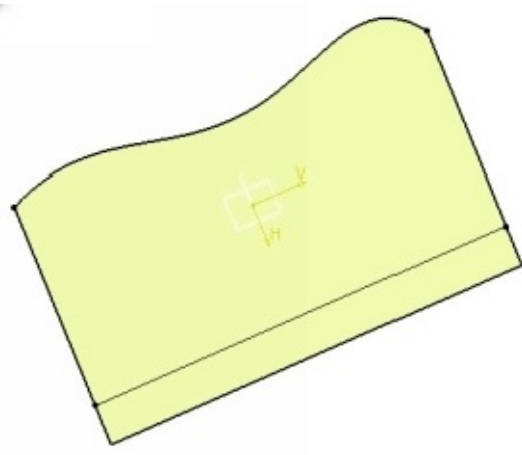
Two Limits

The **Two Limits** option creates a surface using two limiting guide curves.


1. Create two guide curves.
2. Activate the **Sweep** command.
3. On the dialog, select the **Line**  button.
4. Select **Subtype > Two Limits**.
5. Select the two guide curves. The first guide curve will act as the spine.

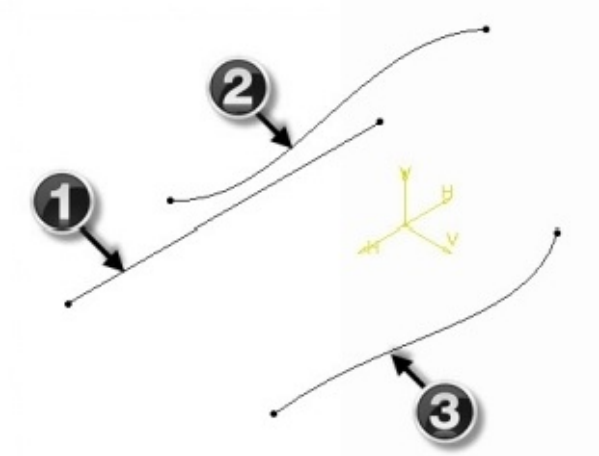


6. Type-in values in the **Length 1** and **Length 2** boxes. This defines the width of the swept surface beyond the two guide curves. You can also click the Law button to define the extension using various law types (Constant, Linear, S type, and Advance).
7. Click **OK**.

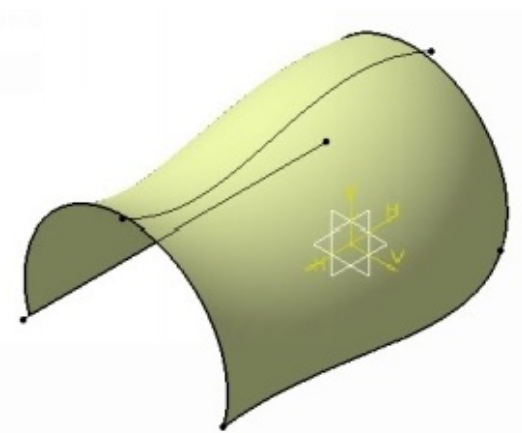


Three Guides

1. The **Three Guides** option creates a circular surface passing through three guide curves.
2. Create three guide curves.
3. Activate the **Sweep** command.
4. On the dialog, select the **Circle**  button.
5. Select **Subtype > Three guides**.
6. Select three guide curves. The first guide curve will act as the spine.



7. Click **OK**.



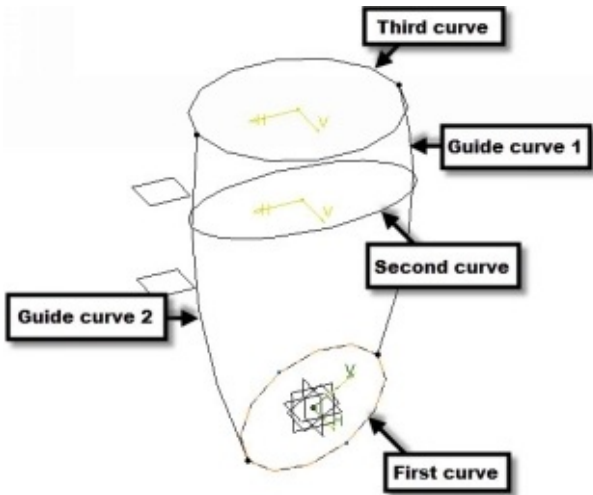
Multi-Sections Surface

This command creates a surface through multiple cross-sections. The shape of the geometry adjusts automatically to pass through the cross-sections.

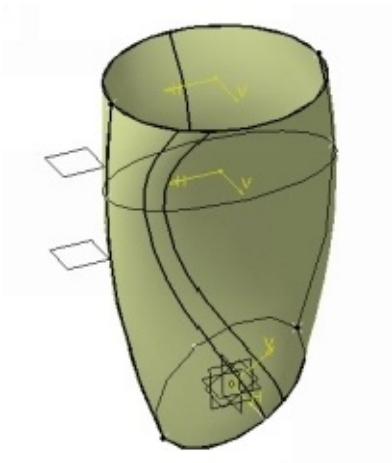
1. Create cross-sections on different planes. The cross-sections can be closed or open curves, and they are not required to be on parallel planes. You can also add guide curves connecting the cross-sections.
2. On the **Surfaces** toolbar, click the **Multi-Sections Surface** button (or) click **Insert > Surfaces > Multi-Sections Surface** on the Menu bar.
3. Select two or more cross-sections. Ensure that arrows are pointing in the same

direction.

4. Click in the **Guides** tab, and then select the guide curves.



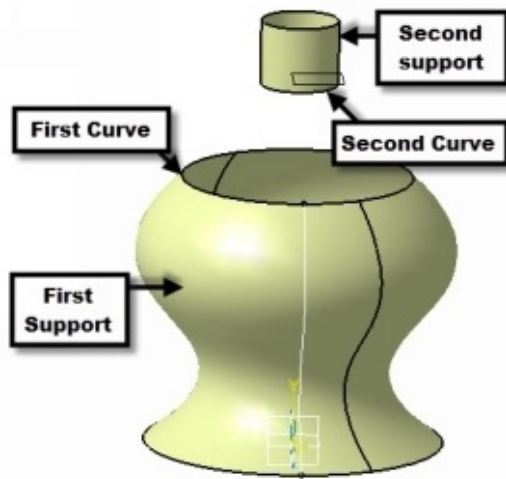
5. Click **OK**.



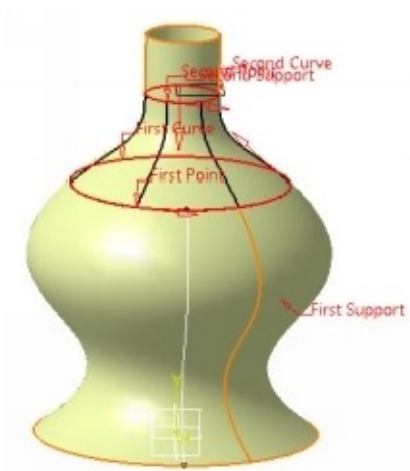
Blend

The **Blend** command creates a surface blending two surfaces. This can be tangent, or curvature, continuous in both the directions.

1. On the **Surfaces** toolbar, click the **Blend** button (or) click **Insert > Surfaces > Blend** on the Menu bar.
2. Click on the first curve and first support.
3. Click on the second curve and second support.



4. Click the **Preview** button on the dialog.

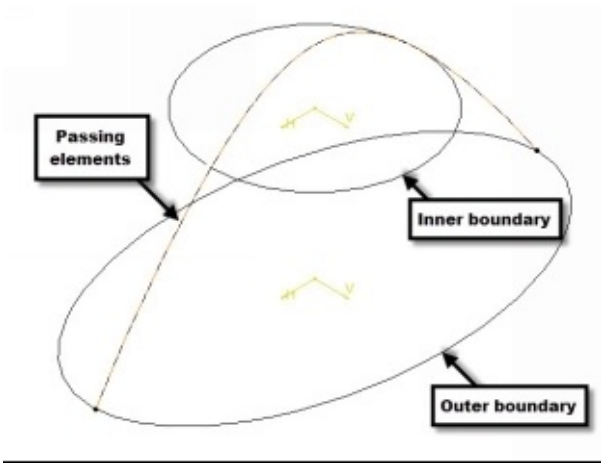


5. Set the **First Continuity** and **Second Continuity** type.
6. Click **OK** to blend the two surfaces.

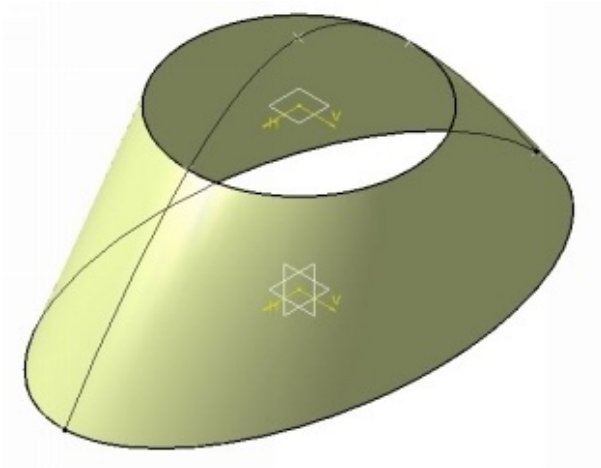
Fill

The **Fill** command can be used either to patch holes in models or to create complex surfaces. As a patching tool, the **Fill Surface** command is more robust than deleting holes or untrimming. It provides more discrete control over the definition of the resultant patch. However, you can also use this command to create complex surfaces.

1. On the **Surfaces** toolbar, click the **Fill** button (or) click **Insert > Surfaces > Fill** on the Menu bar.
2. Click on the outer boundary of the fill surface.
3. Click the **Inner Boundaries** tab and select the inner boundary.
4. Click in the **Passing element(s)** selection box, and then select the curve passing through the boundaries.



5. Click **OK**.

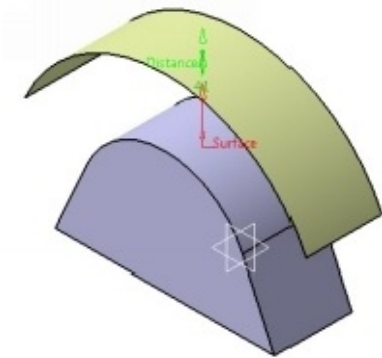




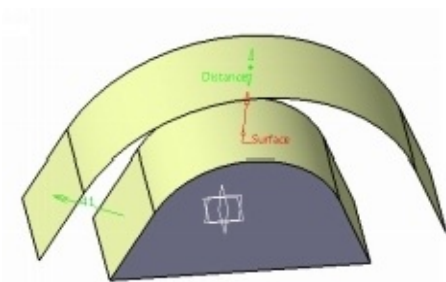
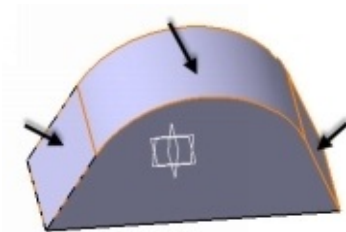
Offset Surface

To create an offset surface, follow the steps given next.

1. On the **Surfaces** toolbar, click **Offsets** drop-down > **Offset** (or) click **Insert** > **Surfaces** > **Offset** on the Menu bar.
2. Select the face to offset.
3. Type-in a value in the **Offset** box.
4. Click **Preview**.



If you want to select multiple faces to offset, then click the right mouse button on a face and select **Create Join**. Now, select the faces that are connected to each other. Click **OK**.



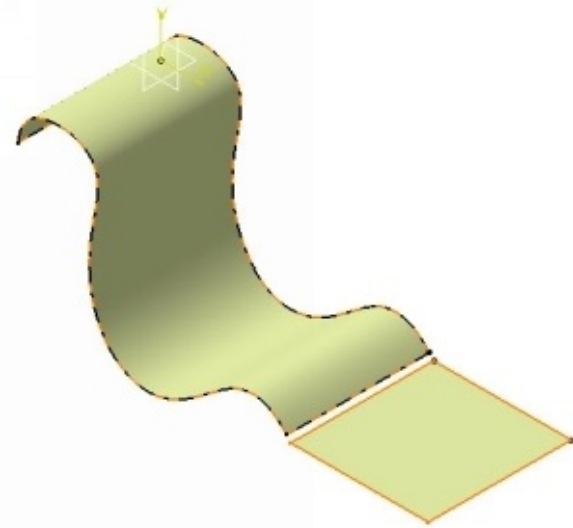
5. Click the **Reverse Direction** button, if you want to reverse the offset direction.
6. Check the **Both sides** option, if you want to create offset surface on the both sides.
7. Check the **Repeat object after OK** option, if you want to repeat the offset surfaces.

The **Smoothing** and **Regularization** options help you smooth complex geometries. Go to CATIA Help file to know more about these options.

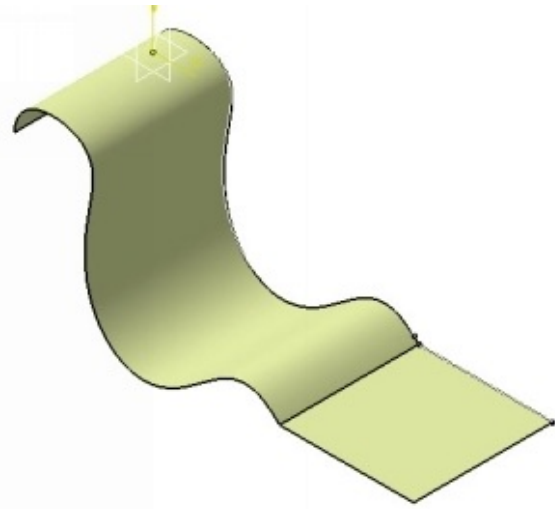
Healing

This command closes small gaps between surfaces.

1. On the **Operations** toolbar, click **Trim-Split** drop-down > **Healing** (or) click **Insert** > **Operations** > **Healing** on the Menu bar.
2. Select the surfaces to heal.



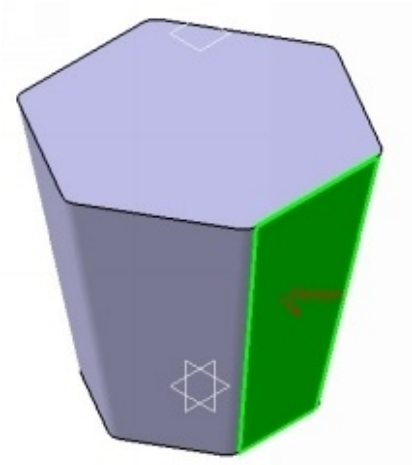
3. Select the **Continuity** type from the **Parameters** section.
4. Type-in a value in the **Merging distance** box. This is the approximate distance between the two surfaces.
5. Click **OK**.



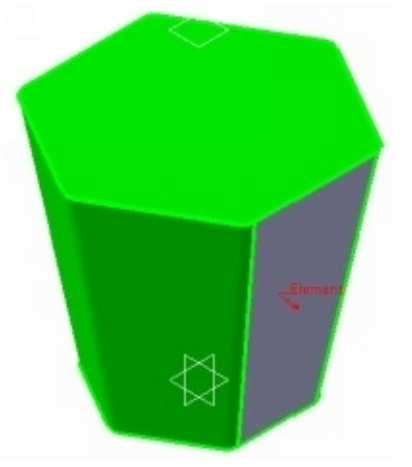
Extract

In some cases, you may need to extract the surfaces of the solid body. You can use the **Extract** command to extract the surfaces of the solid body.

1. On the **Operations** toolbar, click **Extracts** drop-down > **Extract** (or) click **Insert** > **Operations** > **Extract** on the Menu bar.
2. Click on a face of the solid body.



3. If you check the **Complimentary mode** option, the selection will be reversed.

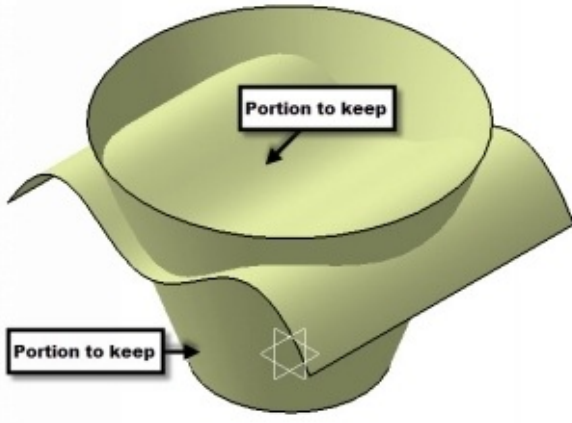


4. If you select **Propagation type** > **Tangent Continuity**, the tangentially connected faces will be selected.
5. Click **OK** to extract the selected surfaces.

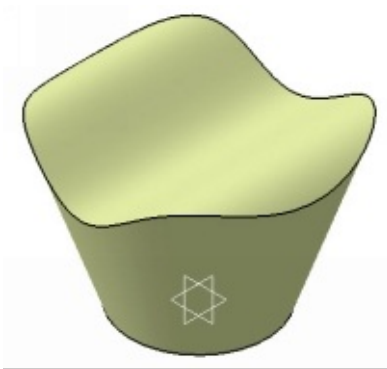
Trim

This command trims and assembles two intersecting surfaces.

1. On the **Operations** toolbar, click **Trim-Split** drop-down > **Trim** (or) click **Insert** > **Operations** > **Trim** on the Menu bar.
2. Click on the portions of the surfaces to keep.



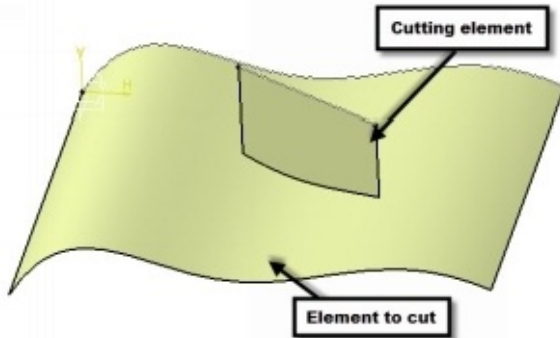
3. Click **OK**.



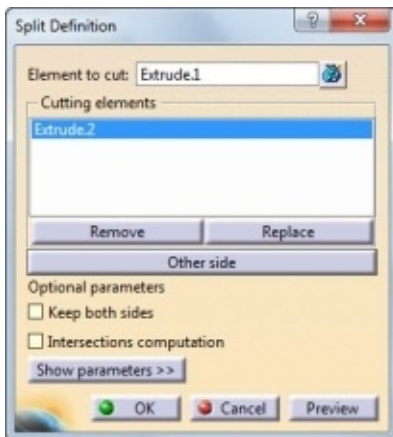
Split

This command splits and trims a surface using an intersecting surface.

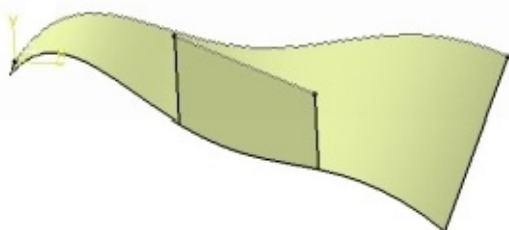
1. On the **Operations** toolbar, click **Trim-Split** drop-down > **Split** (or) click **Insert** > **Operations** > **Split** on the Menu bar.
2. Select the element to cut and cutting element.



3. On the dialog, click the **Other side** button to change the side to be removed.



4. Check the **Keep both sides** option, if you want to keep both the sides of the surface.
5. Click **OK**.

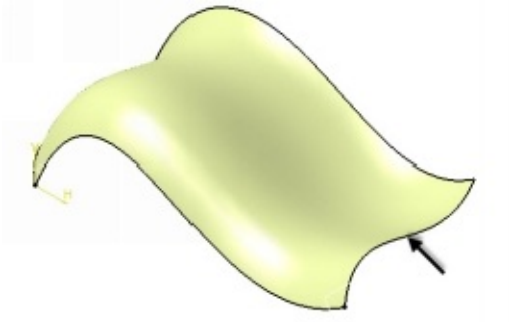


Extrapolate

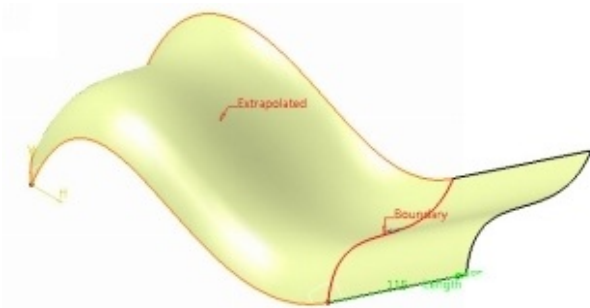
During the design process, you may sometimes need to extend a surface. You can extend a surface using the **Extrapolate** command.

1. On the **Operations** toolbar, click **Extrapolate-Invert** drop-down > **Extrapolate** (or) click **Insert** > **Operations** > **Extrapolate** on the Menu bar.

2. Click on the edge of the surface to extend.



3. Type-in a value in the **Length** box or click and drag the limit handle to define the length of the extend surface.

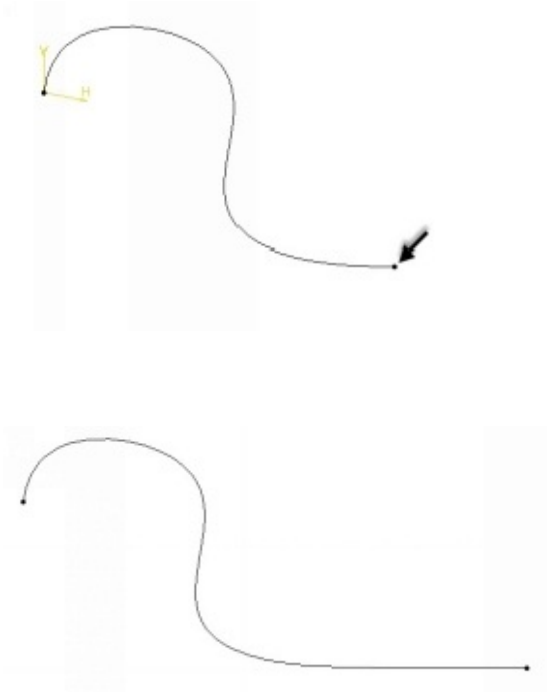


4. Set the **Continuity** type. You can make the extended surface **Tangent** or **Curvature** continuous with the original surface.

5. Likewise, set the **Extremities** type.

6. Check the **Assemble result** option, and then click **OK**.

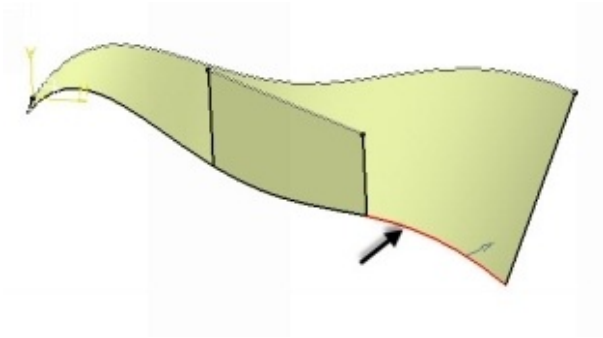
Likewise, you can extend a curve by selecting its endpoint.



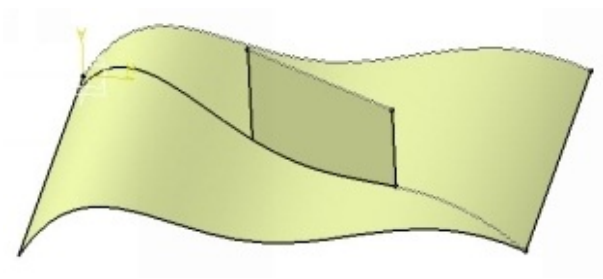
Untrim

You can untrim a trimmed surface using the **Untrim** command.

1. On the **Operations** toolbar, click **Trim-Split** drop-down > **Untrim** (or) click **Insert** > **Operations** > **Untrim** on the Menu bar.
2. Click on the edge of the trimmed surface.



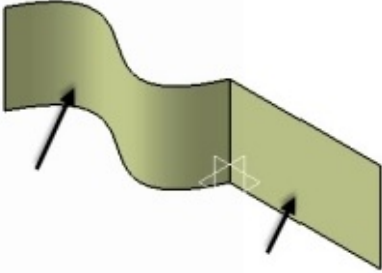
3. On the dialog, select the **Create Curves** button if you want to create a boundary curve on selected edge.
4. Click **OK**.



Join

The surfaces created act as individual surfaces unless they are joined together. The **Join** command lets you combine two or more surfaces to form a single surface.

1. On the **Operations** toolbar, click **Join-Healing** drop-down > **Join** (or) click the **Insert** > **Operations** > **Join** on the Menu bar.
2. Select the surfaces to join.



The value you type in the **Merging distance** box defines the maximum gap. All the surfaces within the gap will be joined.

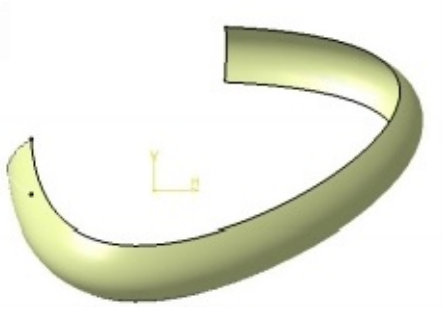
3. Click the **OK** button to join the surfaces.



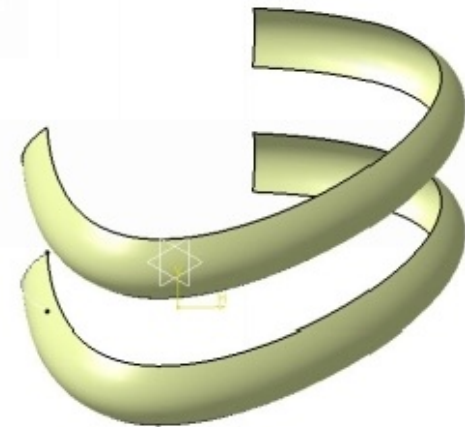
Translate

The **Translate** command moves and copies a surface.

1. On the **Operations** toolbar, click **Transformations** drop-down > **Translate** (or) click **Insert** > **Operations** > **Translate** on the Menu bar.
2. Select the surface/element to translate.



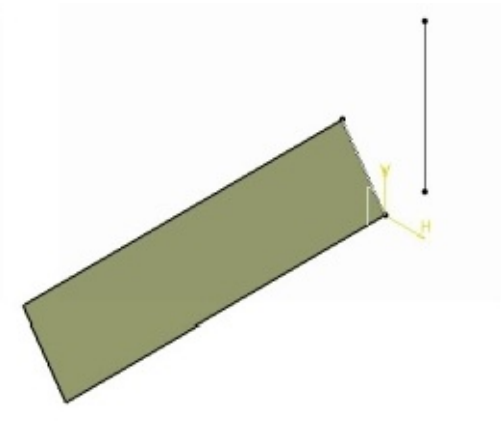
3. On the dialog, click **Vector Definition** > **Direction, distance**. You can also select **Point to Point** or **Coordinates** to define the direction and translation.
4. Select a line, axis, or plane to define the translation direction.
5. Type-in a value in the **Distance** box or drag the distance handle.
6. Click **OK**.



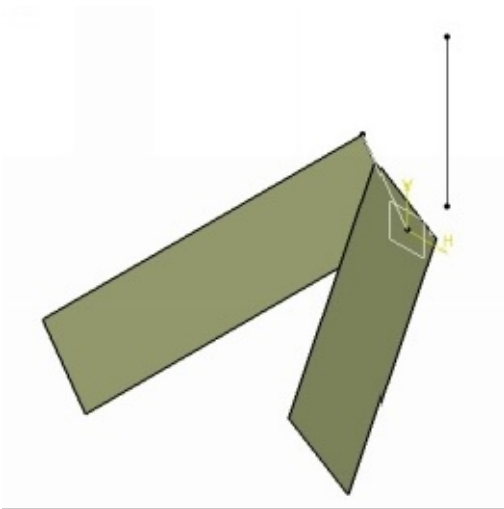
Rotate

This command rotates an element about an axis.

1. On the **Operations** toolbar, click **Transformations** drop-down > **Rotate** (or) click **Insert** > **Operations** > **Rotate** on the Menu bar.
2. Select the surface/element to rotate.



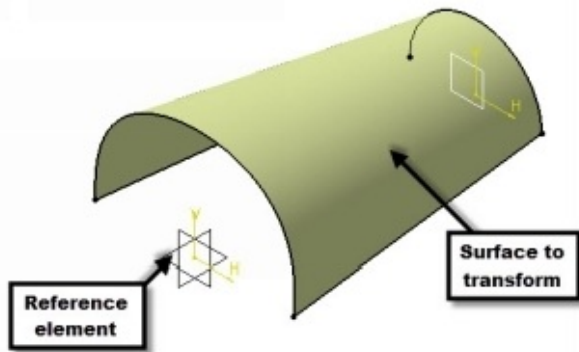
3. On the dialog, click **Definition mode** > **Axis-Angle**. You can also select **Axis-Two Elements** or **Three Points** to define the axis and rotation angle.
4. Select a line or axis to define the rotation axis.
5. Type-in a value in the **Angle** box.
6. Check the **Repeat object after OK** option, if you want to repeat the rotation after clicking **OK**.
7. Click **OK**.



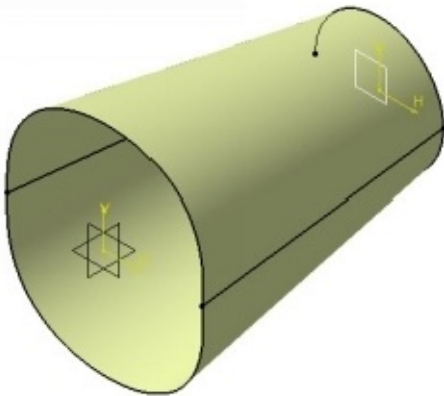
Symmetry

This command creates a symmetrical element about a reference element.

1. On the **Operations** toolbar, click **Transformations** drop-down > **Symmetry** (or) click **Insert** > **Operations** > **Symmetry** on the Menu bar.
2. Select the element to transform.
3. Select a point, line or plane.



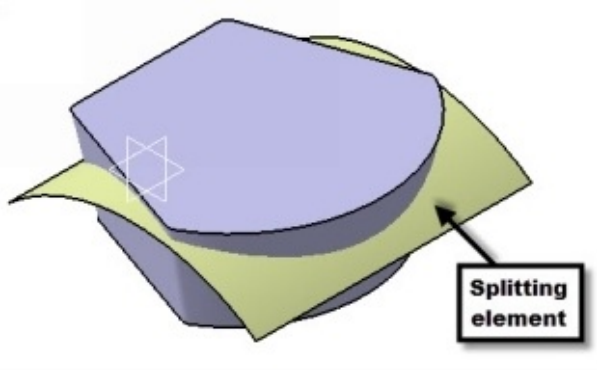
4. Click **OK**.



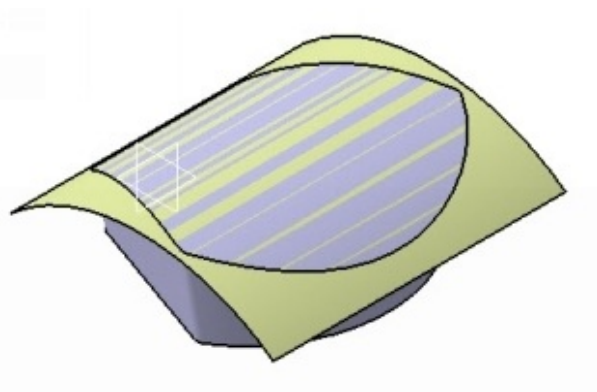
Split (Body)

This command splits a body using a plane, surface or face.

1. Create a body in the **Part Design** workbench.
2. Create a splitting surface.
3. Switch to the **Part Design** Workbench (click **Start** > **Mechanical Design** > **Part Design** on the Menu bar).
4. On the **Surface-Based Features** toolbar, click **Split** (or) click **Insert** > **Surface-Based Features** > **Split** on the Menu bar.
5. Click on the splitting surface or plane.



6. Click the arrow that appears on the geometry to reverse the side to be removed.
7. Click **OK**.

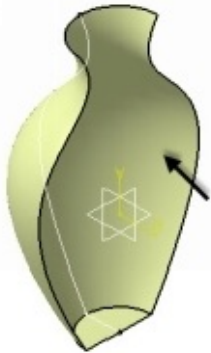




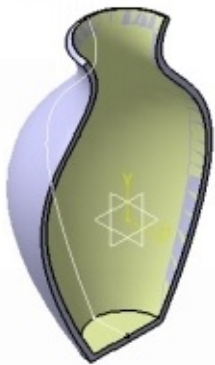
Thick Surface

Creating a solid from a surface can be accomplished by simply thickening a surface. To add thickness to a surface, follow the steps given next.

1. Switch to the **Part Design** Workbench (click **Start** > **Mechanical Design** > **Part Design** on the Menu bar).
2. On the **Surface-Based Features** toolbar, click **Thick Surface** (or) click **Insert** > **Surface-Based Features** > **Thick Surface** on the Menu bar.
3. Click on a face of the surface geometry.



4. Enter the thickness value in the **First offset** box.
5. Click the arrows that appear on the geometry reverse the side to which the material added.
6. Click **OK**.

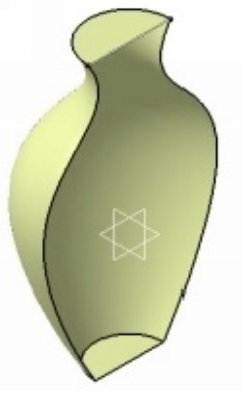




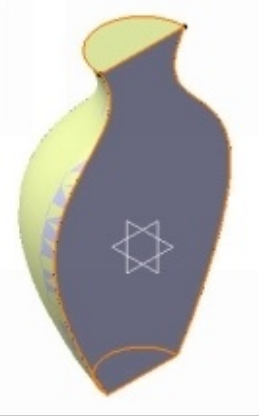
Close Surface

This command creates a solid body by filling the volume enclosed by a surface body.

1. Join the surfaces using the **Join** command.



2. Switch to the **Part Design** Workbench (click **Start** > **Mechanical Design** > **Part Design** on the Menu bar).
3. On the **Surface-Based Features** toolbar, click **Thick Surface** drop-down > **Close Surface** (or) click **Insert** > **Surface-Based Features** > **Close Surface** on the Menu bar.
4. Select the surface geometry, and then click **OK**.



Wireframe Geometry

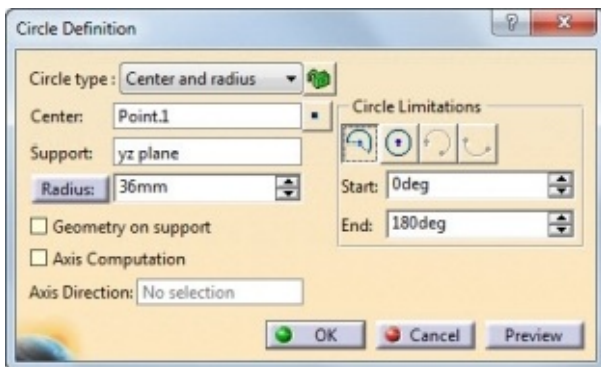
The Generative Shape Design workbench has commands to create three dimensional curves and wireframe elements. They help you to create complex surfaces.



Circle

This command creates circles or arcs.

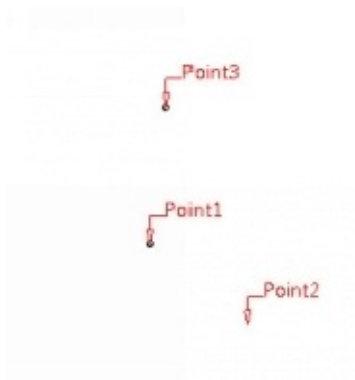
1. On the **Wireframe** toolbar, click **Circle-Conic** drop-down > **Circle** (or) **Insert** > **Wireframe** > **Circle** on the Menu bar.
2. Select a point to define the center point. You can also create a new point by clicking the right mouse button in the **Center** box, and then selecting an option to create a point.
3. Click on a plane or face to define the support surface.
4. Type-in a value in the **Radius** box or drag the Radius handle to define the radius.
5. Type-in values in the **Start** and **End** boxes to define the start and end limits of the arc.
6. Click the **Whole Circle** button on the dialog, if you want to create a complete circle.
7. Click **OK**.



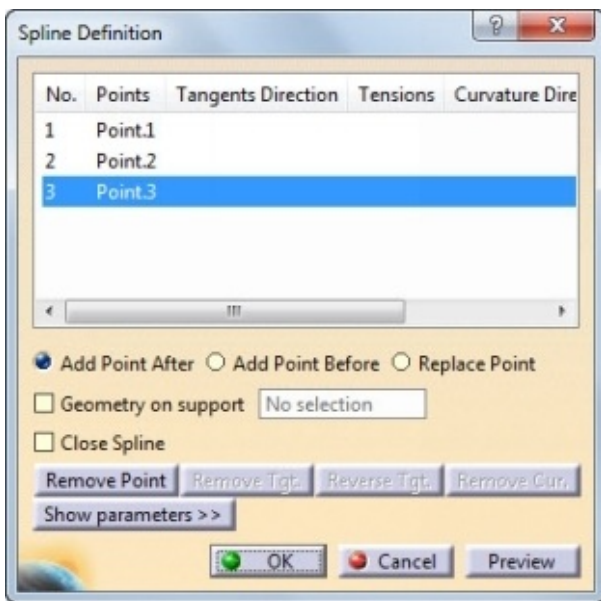
Spline

This command creates a three dimensional spline curve.

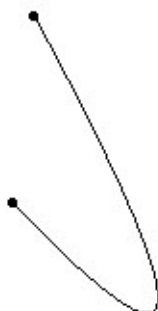
1. On the **Wireframe** toolbar, click **Curves** drop-down > **Spline** (or) **Insert** > **Wireframe** > **Spline** on the Menu bar.
2. Select a point or click the right mouse button and select an option to create a point.
3. Likewise, select or create points one-by-one.



4. Check the **Geometry on support** option, if you want to create the spline on a particular plane or surface.
5. Check the **Close Spline** option, if you want to create a closed spline.



6. Click **OK**.



Corner

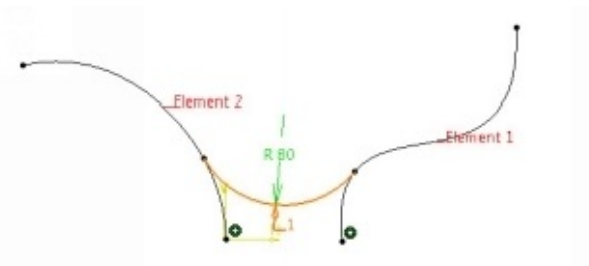
This command creates a corner curve between two curves.

1. On the **Wireframe** toolbar, click **Circle-Conic** drop-down > **Corner** (or) **Insert** > **Wireframe** > **Corner** on the Menu bar.

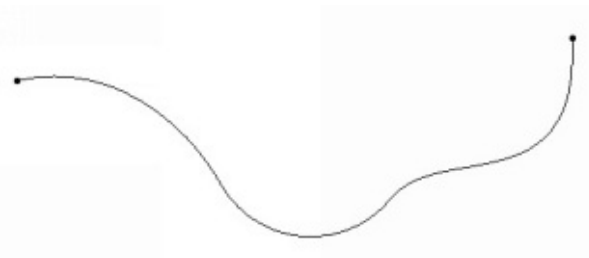
2. Click on a curve or point to define the first element.
3. Click on a curve or point to define the second element.



4. Type-in a value in the **Radius** box.
5. Click the **Next Solution** button to view different solutions of the corner.



6. Check the **Trim element 1** and **Trim element 2** options, if you want to trim the first and second element.



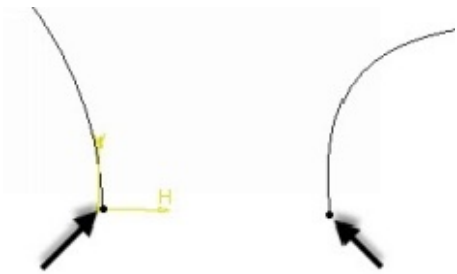
7. Click **OK**.



Connect Curve

This command creates a connecting curve between two elements.

1. On the **Wireframe** toolbar, click **Circle-Conic** drop-down > **Connect Curve** (or) **Insert** > **Wireframe** > **Connect Curve** on the Menu bar.
2. Click on the end point of the first curve.
3. Click on the end point of the second curve.



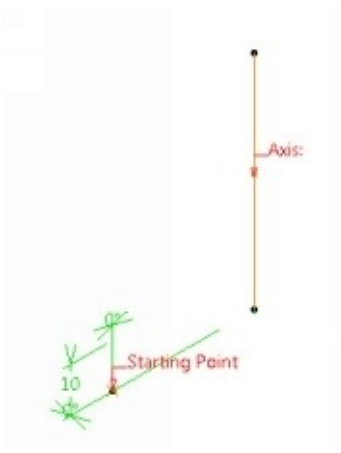
4. Define the **Continuity** type and **Tension** of the first and second curves.
5. Click **OK** to connect the two curves.



Helix

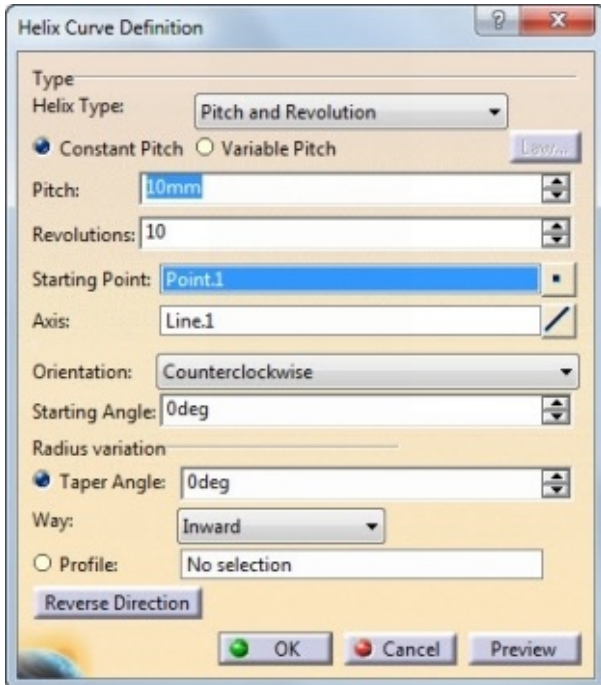
This command creates a helical curve.

1. On the **Wireframe** toolbar, click **Curves** drop-down > **Helix** (or) **Insert** > **Wireframe** > **Helix** on the Menu bar.
2. Select the starting point of the helix. You can also create a new point. To do this, click the right mouse button and select an option to create the point.
3. Select a line to define the axis or create a new line.

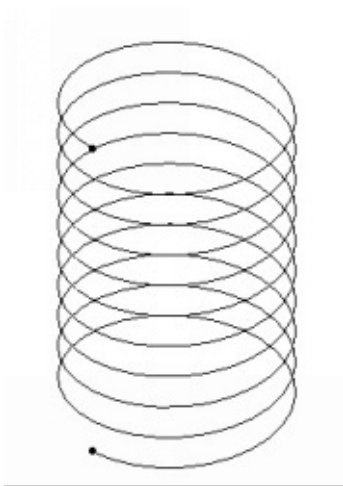


4. On the dialog, click **Helix type** > **Pitch and Revolution**. This defines the helix by using the pitch and revolutions that you specify. You can also select **Height and Pitch** or **Height and Revolution** options.
5. Select the **Constance Pitch** option. This creates a helix with a constant pitch. You can also select **Variable Pitch** option to create a helix with varying pitch. You have to define the start and end pitch of the variable pitch helix.

6. Type-in the **Pitch** and **Revolution** values.
7. Set the **Orientation** to **Counterclockwise** or **Clockwise**.
8. Type-in a value in the **Taper angle** box, if you want to create a tapered helix. You can apply an **Inward** or **Outward** taper to the helix.



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

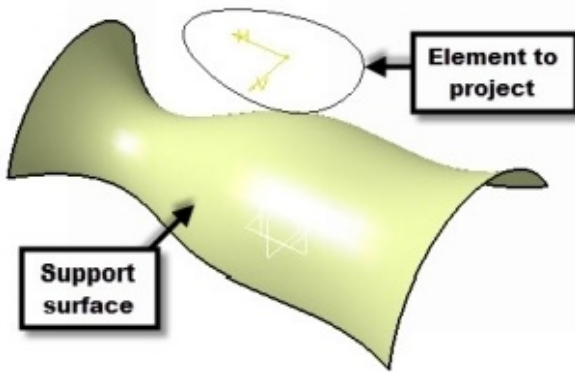


Projection

This command projects an element on to a supporting surface.

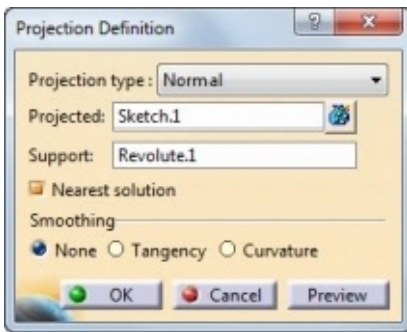
1. On the **Wireframe** toolbar, click **Project-Combine** drop-down > **Projection** (or) **Insert** > **Wireframe** > **Projection** on the Menu bar.
2. Select the element to project.

3. Select the supporting surface.

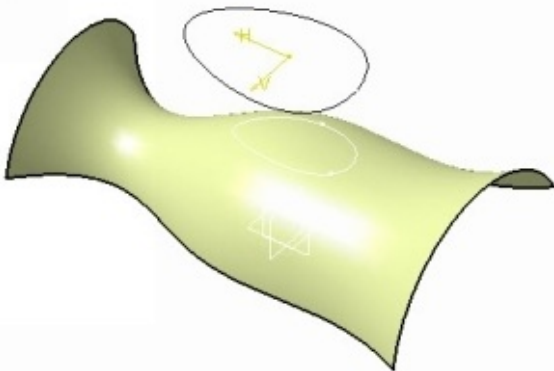


4. Select **Projection type > Normal**. This projects the element in the direction normal to the supporting surface. You can select the **Along a direction** option to define the direction of the projection. You can use a line or plane to define the direction.

5. If the curve is projected at multiple locations on the supporting surface, then check the **Nearest solution** option. This keeps the nearest projection.



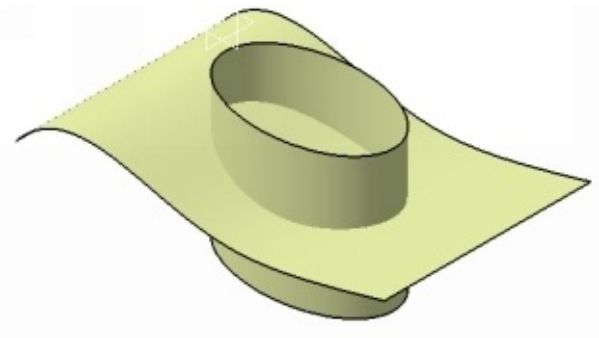
6. Click **OK**.



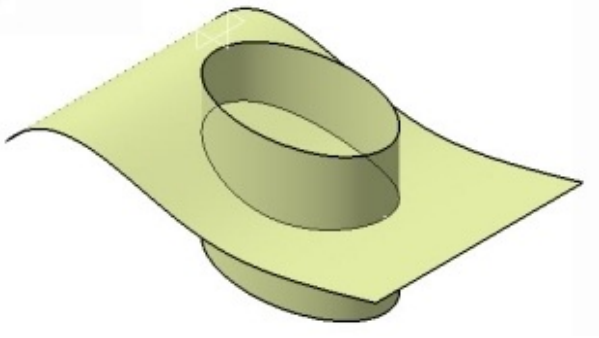
Intersection

This command creates a wireframe geometry at the intersection of two elements.

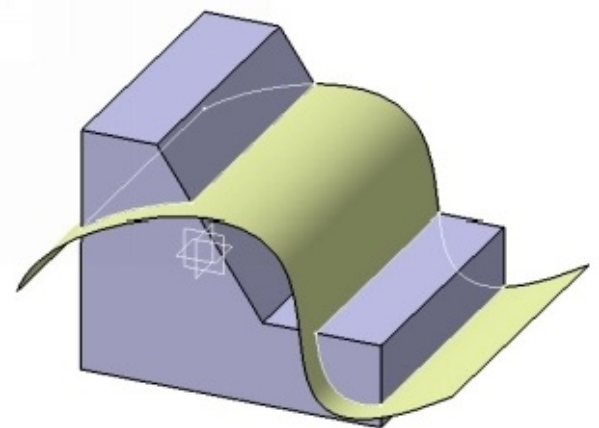
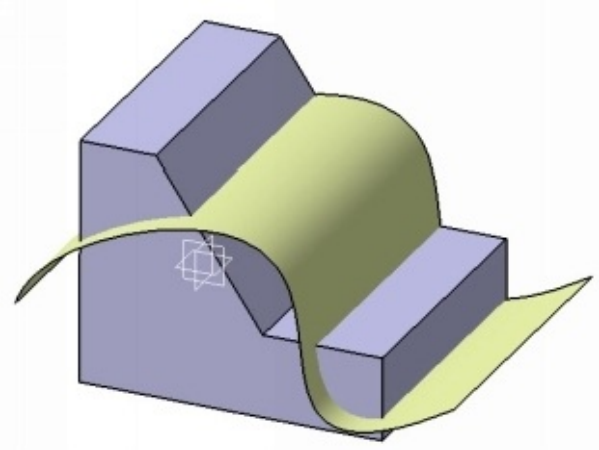
1. On the **Wireframe** toolbar, click the **Intersection** button (or) **Insert > Wireframe > Intersection** on the Menu bar.
2. Select two intersecting elements.

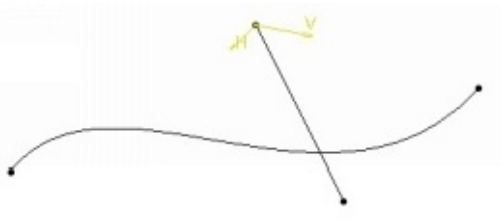


3. Click **OK**.



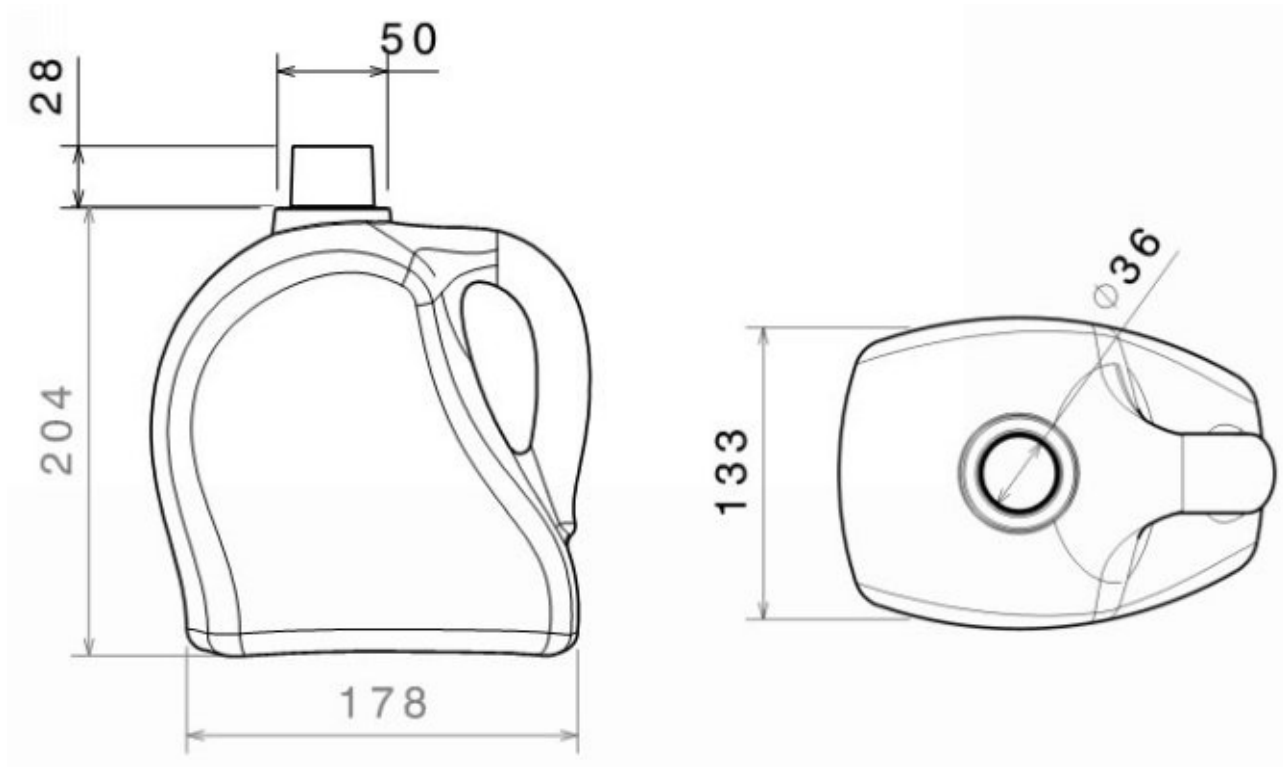
Other examples of the intersection curves are given next.





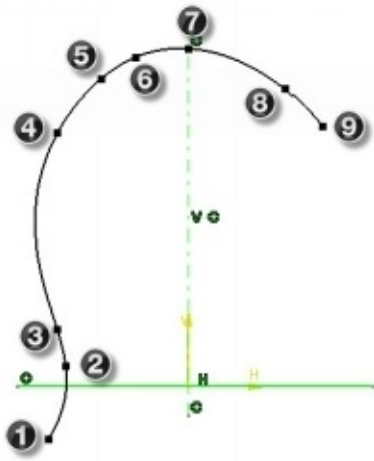
Example

In this example, you will construct the model shown below.

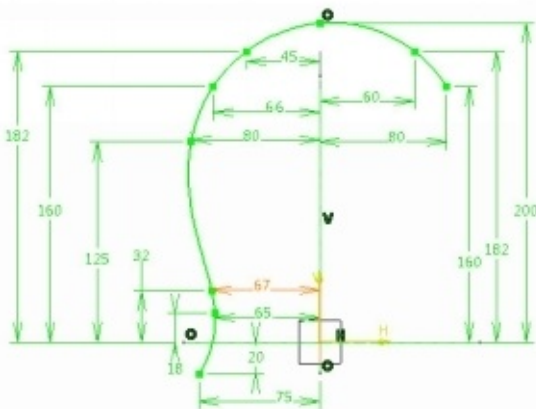


Drawing the Layout sketch

1. Start **CATIA V5-6R2015**.
2. On the Menu bar, click **Start > Shape > Generative Shape Design**.
3. Start a sketch on the YZ plane.
4. Draw a sketch similar to the one shown below. Use the **Axis** and **Spline** commands to draw this sketch.

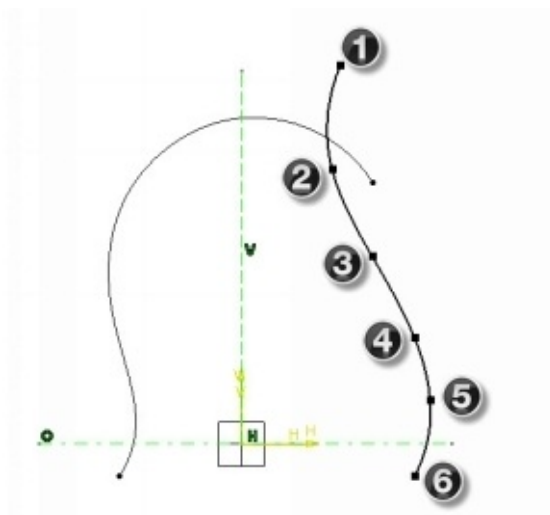


5. Activate the **Constraint** command and add dimensional constraints to the sketch.

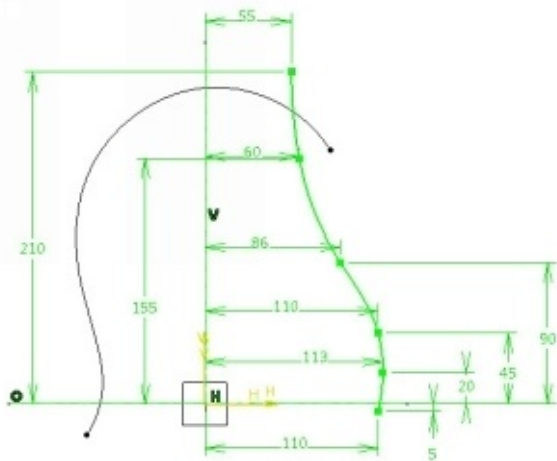


6. Exit the sketch

7. Start a new sketch on the YZ plane draw another spline curve similar to the one shown in figure.

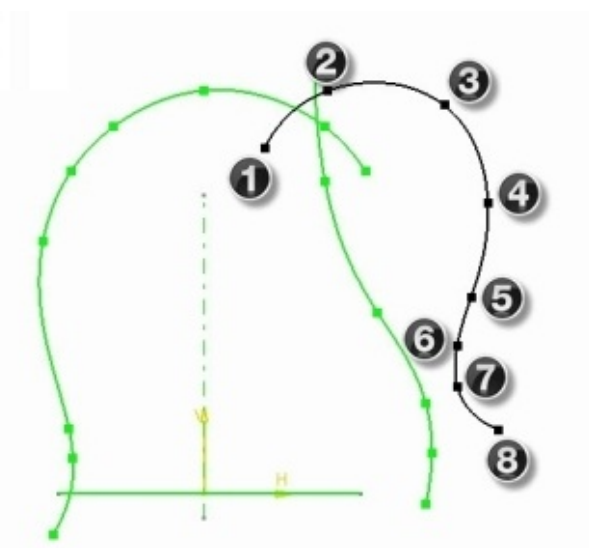


8. Add constraints to the spline.

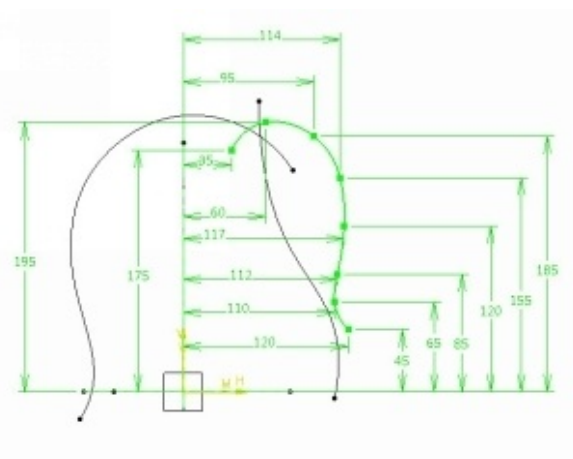


9. Exit the sketch

0. Start a new sketch on the YZ plane and create another spline similar to the one shown next.



1. Add constraints to the spline.



2. Exit the sketch.

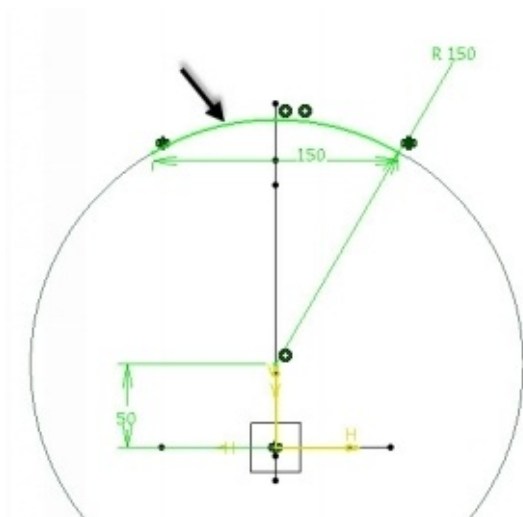
If you find it difficult to create the layout sketch, then you can download it from our website.

Creating the Front Surface

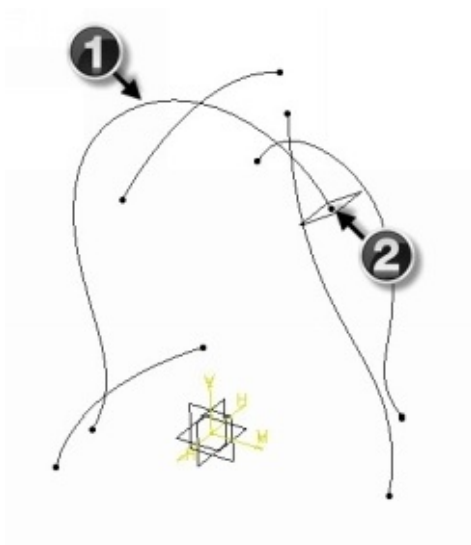
1. Create an arc on the XY Plane and add dimensions to it. Exit the sketch.



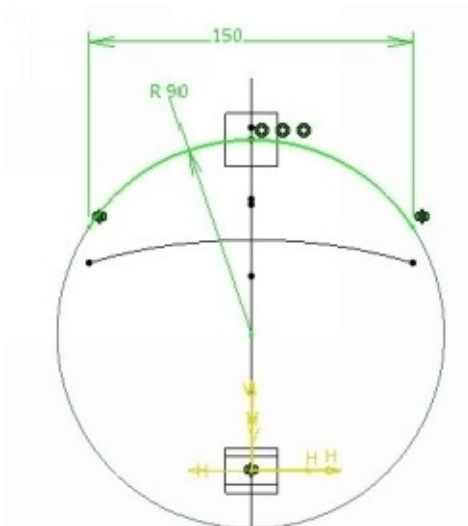
2. Create an arc on the ZX Plane and add dimensions to it. Finish the sketch.




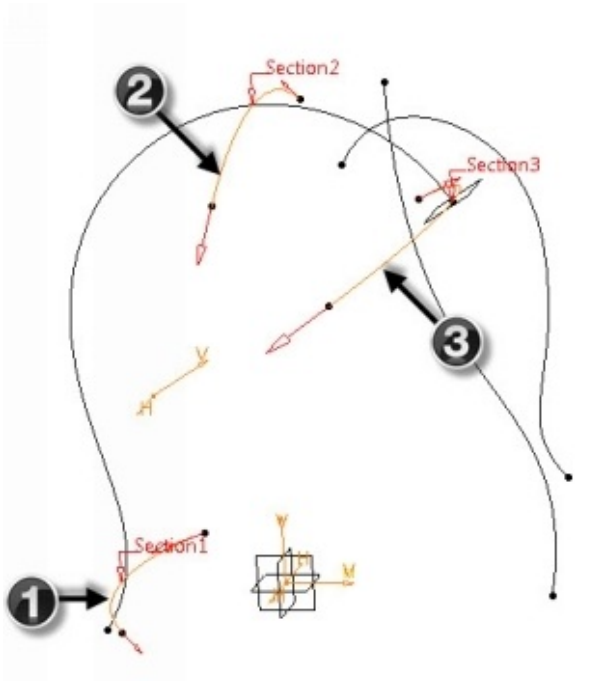
3. Create a reference plane normal to the front face spline.



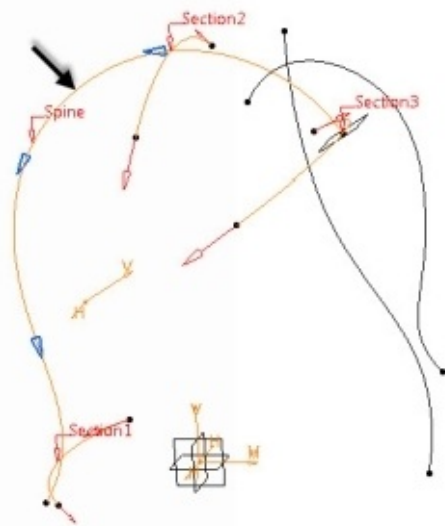
4. Create an arc on the plane normal to curve. Exit the sketch.



5. On the **Surfaces** toolbar, click the **Multi-Sections Surface**  button (or) click **Insert > Surfaces > Multi-Sections Surfaces** on the Menu bar.
5. Select the three sections from the graphics window. You have to make sure that the arrows on the sections point in the same direction. You can double-click on the arrows to change the direction.



7. On the dialog, click the **Spine** tab and select the first sketch to define the spine.



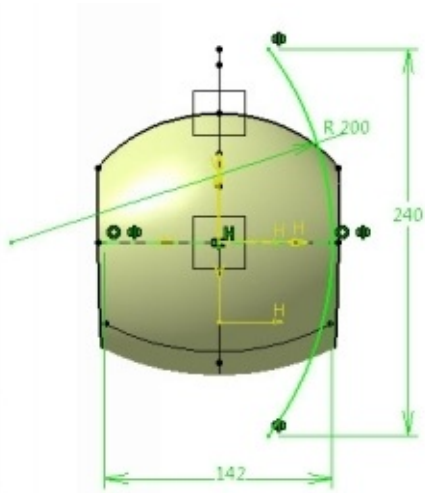
3. Click the **Preview** button to preview the surface. If you got the desired result, then click **OK** to create the surface.




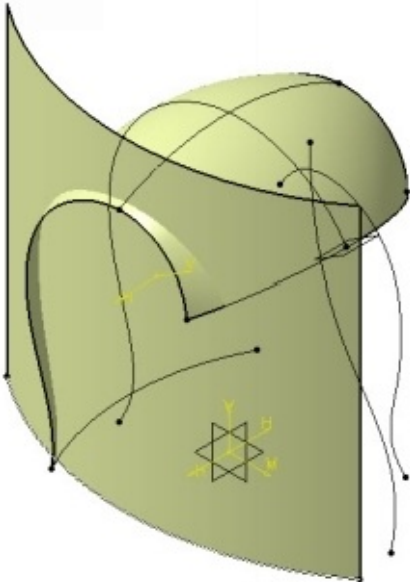
3. Save the file. As you are creating a complex geometry, it is advisable that you save the model after each operation.


Creating the Label surface

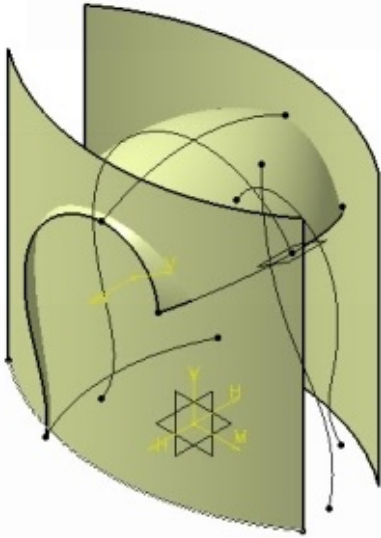
1. Create an arc on the XY plane. Exit the sketch.



2. On the **Surfaces** toolbar, click the **Extrude**  button (or) click **Insert > Surfaces > Extrude** on the Menu bar.
3. On the dialog, under the **Limit 1** section, type-in **220** in the **Dimension** box. Click **OK**.

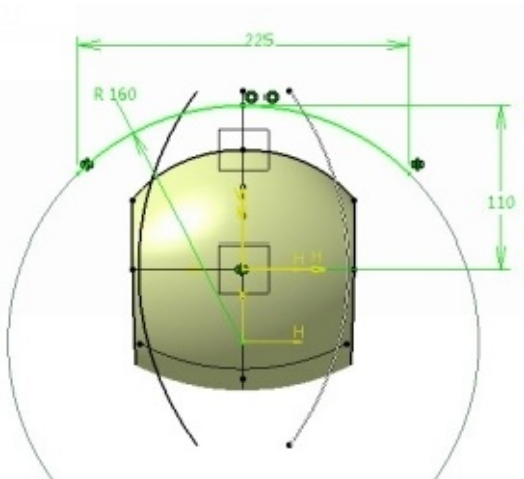



4. On the **Operations** toolbar, click **Transformations** drop-down > **Symmetry**  (or) click **Insert > Operations > Symmetry** on the Menu bar.
5. Click on the Extrude surface, and then click the YZ plane.
6. Click **OK** to symmetrically transform the extrude surface.

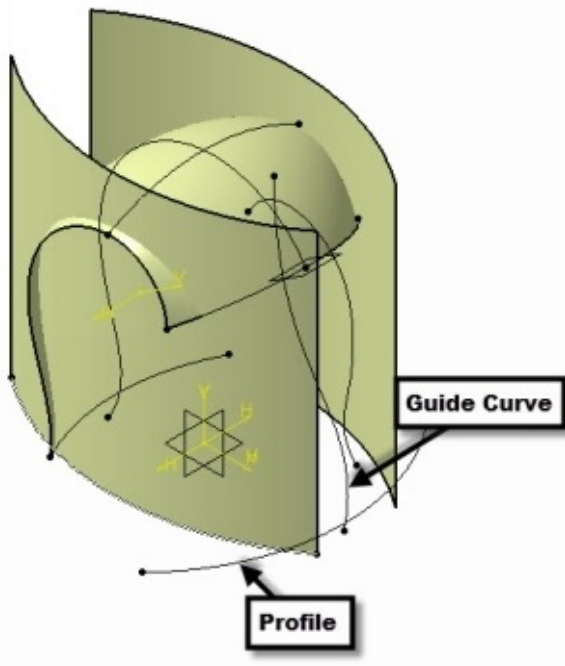


Creating the Back surface

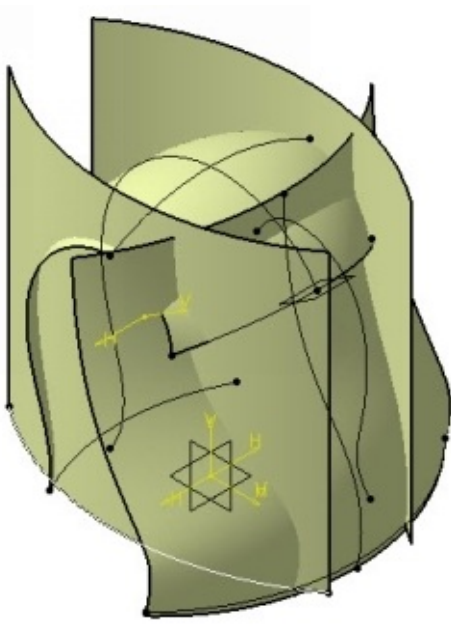
1. Create an arc on the XY plane. Exit the sketch.




2. On the **Surfaces** toolbar, click the **Sweep**  button (or) click **Insert** > **Surfaces** > **Sweep** on the Menu bar.
3. Select the profile and guide curve.

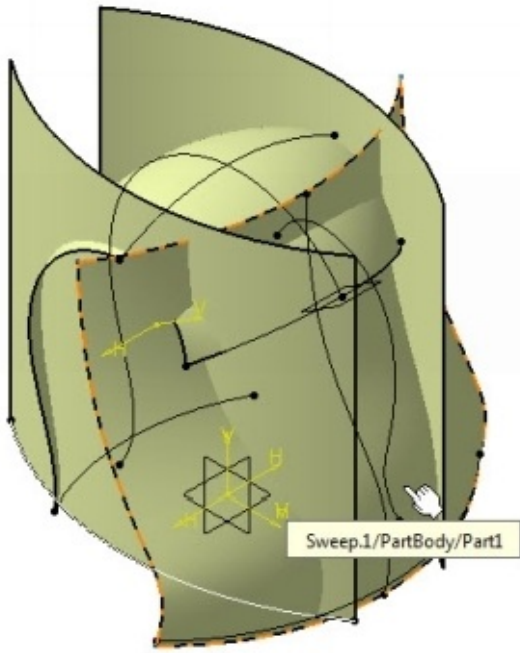


4. Click **OK** to create the surface.

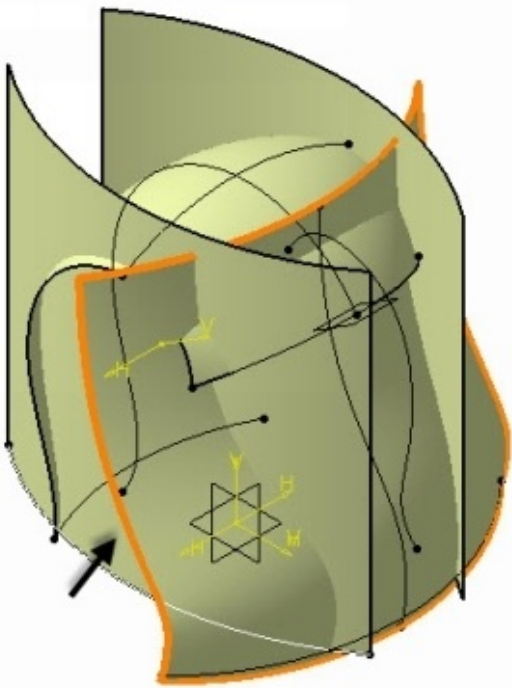


Trimming the Unwanted Portions

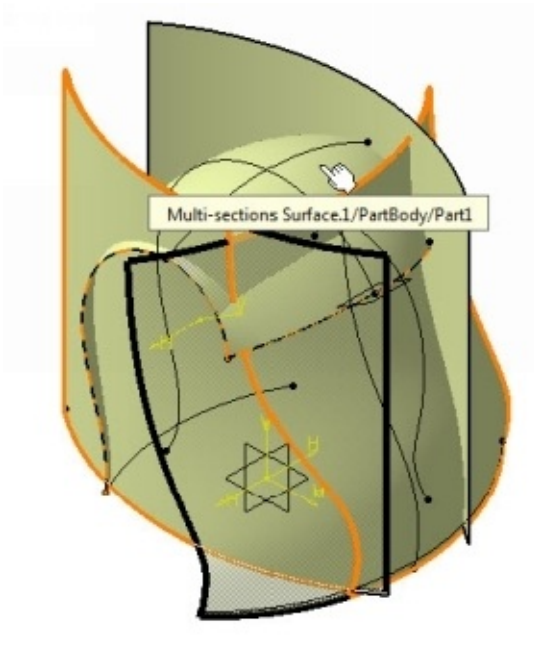
1. On the **Operations** toolbar, click **Split-Trim** drop-down > **Trim**  (or) click **Insert** > **Operations** > **Trim** on the Menu bar.
2. On the dialog, select **Mode** > **Pieces**.
3. Click on the portion of the sweep surface, as shown below.



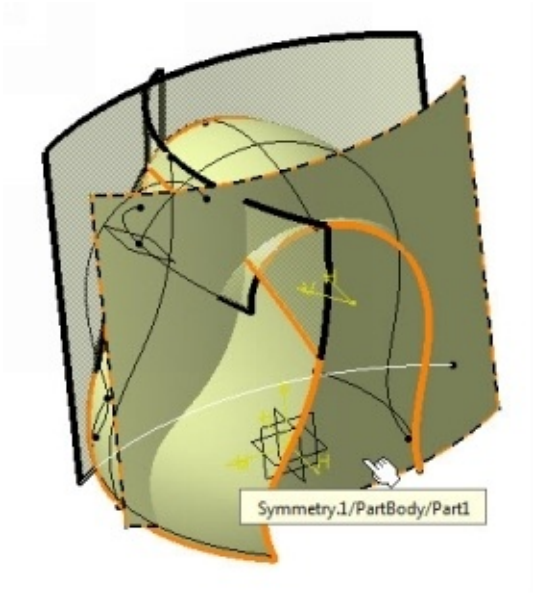
4. Click on the portion of the extrude surface, as shown next.



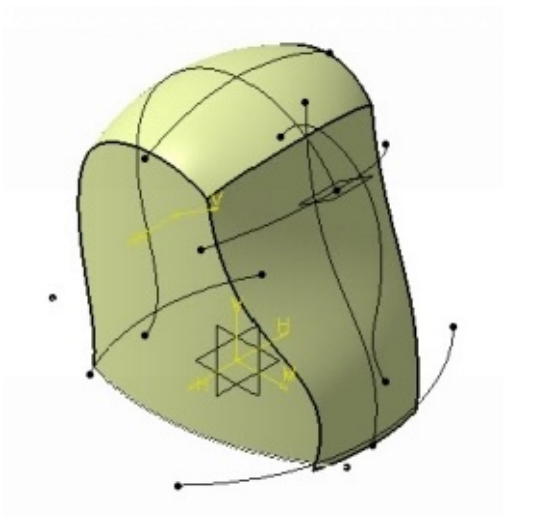
5. Click on the portion of the multi-section surface, as shown next.



5. Rotate the model and click on the symmetry surface, as shown next.

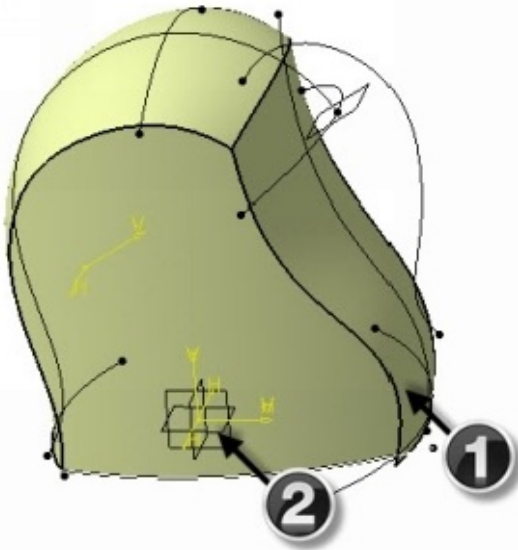


7. Click **OK** to trim the unwanted portions.



Trim the Sweep Surface using the Split command

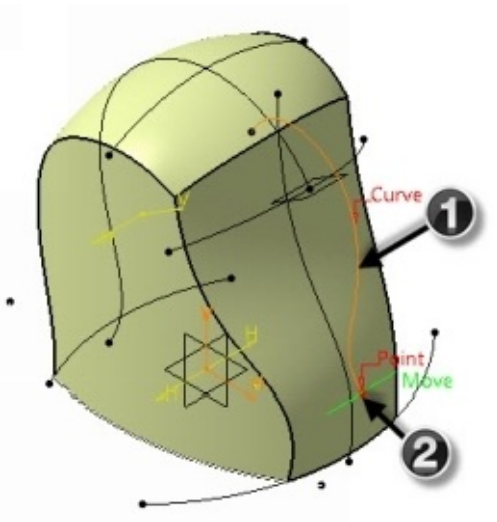
1. On the **Operations** toolbar, click **Split-Trim** drop-down > **Split**  (or) click **Insert** > **Operations** > **Split** on the Menu bar.
2. Click on the surface and the xy plane.



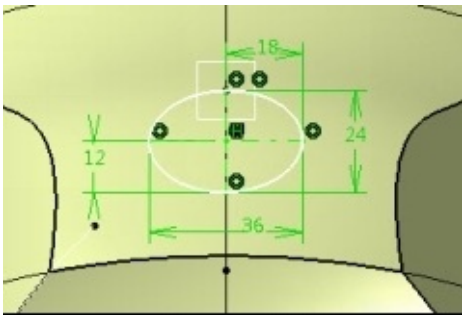
3. Click **OK** to trim the surface.

Creating the Handle Surface

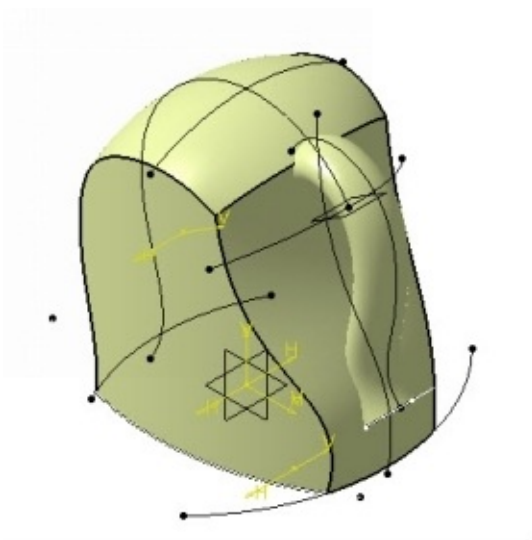
1. Activate the **Plane** command and click on the spline and its lower end-point, as shown below. Click **OK** to create the plane normal to the spline.




2. Start a sketch on the plane normal to the spline.
3. Activate the **Ellipse** command and create an ellipse on the sketch plane.
4. Make the upper quadrant point of the ellipse coincident with the end-point of the spline.
5. Add dimensions and constraints to the sketch. Exit the sketch.

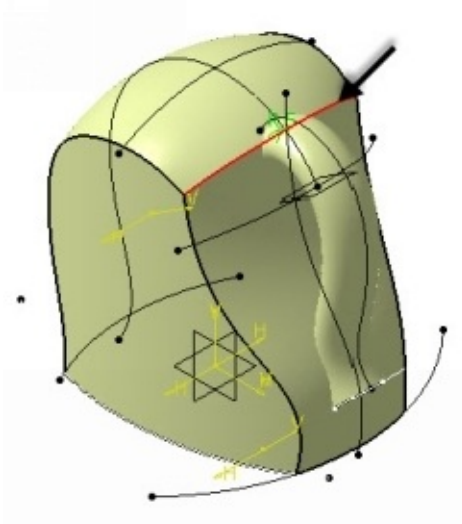


5. On the **Surfaces** toolbar, click **Sweeps** drop-down > **Sweep** (or) click **Insert** > **Surfaces** > **Sweep**. This selects the ellipses, automatically. If not, select the ellipse to define the profile.
7. Select the spline to define the guide curve.
3. Click **OK**.

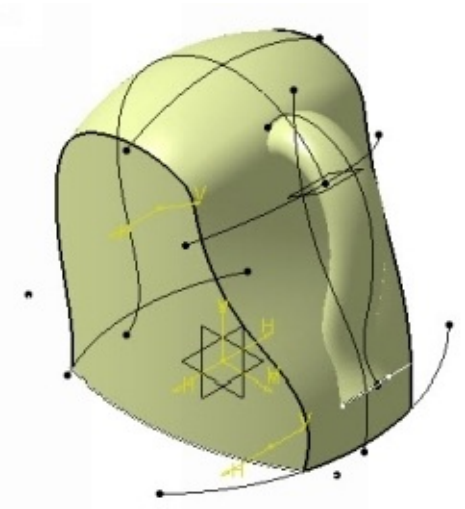


Blending the Front and back faces

1. On the **Operations** toolbar, click **Fillets** drop-down > **Edge Fillet**  (or) click **Insert** > **Operations** > **Edge Fillet** on the Menu bar.
2. Click on the edge connecting the front and back faces.

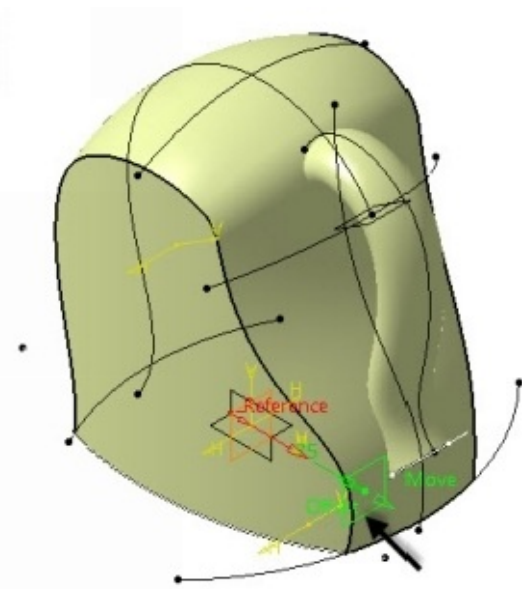



3. On the dialog, type-in 25 in the **Radius** box, and then click **OK**.

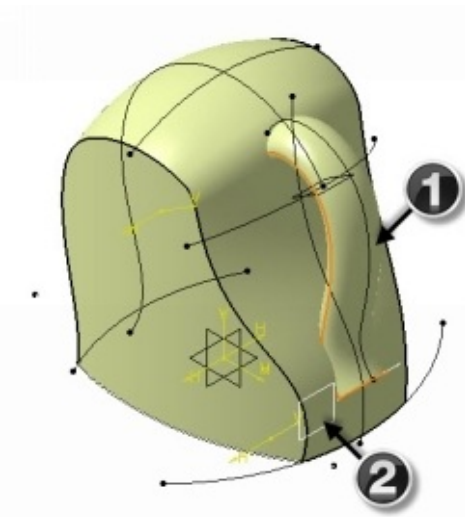


Trimming the Handle

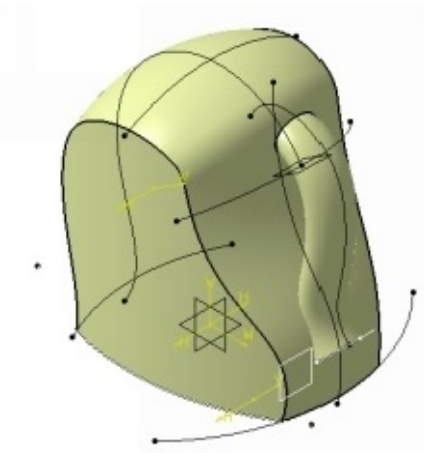
1. Create a plane offset from the zx plane. The offset distance is 75 mm.



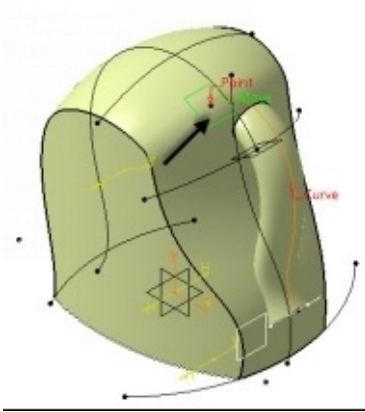
2. On the **Operations** toolbar, click **Split-Trim** drop-down > **Split**  (or) click **Insert** > **Operations** > **Split** on the Menu bar.
3. Click on the sweep surface, and then the offset plane.



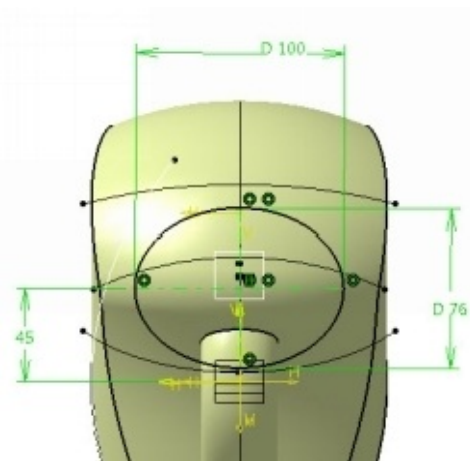
4. Click **OK** to trim the sweep surface.




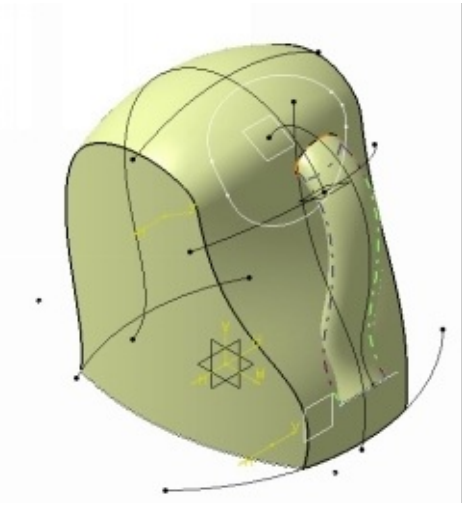
5. Create a reference plane, which is normal to the spline and located at the top end-point.




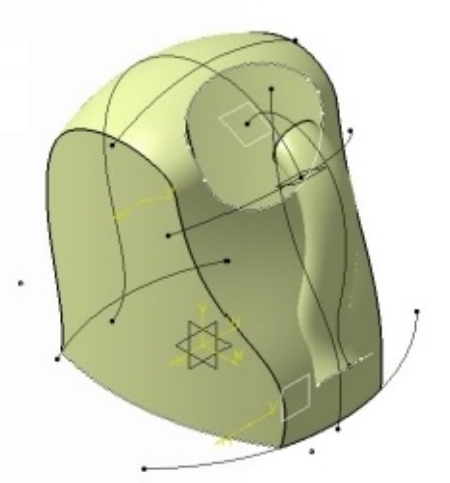
6. Start a sketch on the plane normal to the spline and draw an ellipse. Add dimensions to position the ellipse, and then Exit the workbench.




7. On the **Wireframe** toolbar, click **Project-Combine** drop-down > **Projection**  (or) click **Insert** > **Wireframe** > **Projection** on the Menu bar.
8. On the **Projection Definition** dialog, click **Projection type** > **Along a direction**.
9. Click in the **Projected** selection box and select the ellipse.
10. Click on the main surface to define the support.
1. Click on the plane normal to the handle spline. This defines the projection direction.
2. Select **Tangency** under the **Smoothing** section.
3. Click **OK** to project the sketch on to the main surface.
4. Hide the sketched ellipse to avoid confusion.

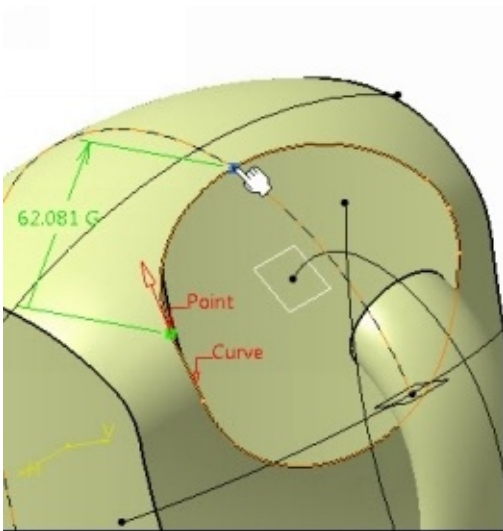


5. On the **Operations** toolbar, click **Split-Trim** drop-down > **Split**  (or) click **Insert** > **Operations** > **Split** on the Menu bar.
6. Click on the main surface and projected curve.
7. Click **OK** to trim the main surface.

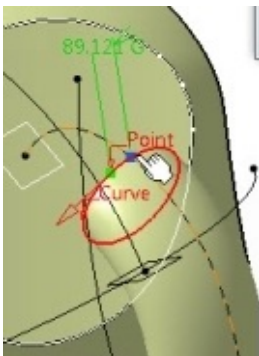



Blending the Top handle

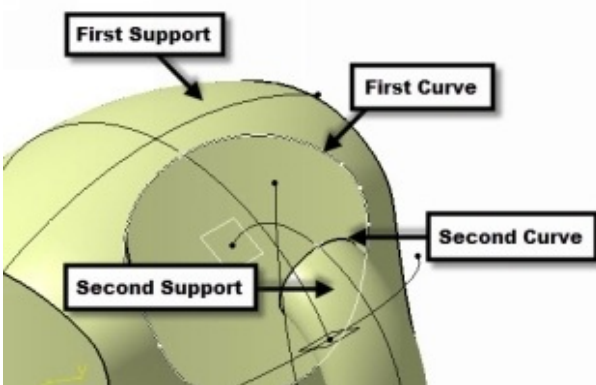
1. On the **Wireframe** toolbar, click **Points** drop-down > **Point**  (or) click **Insert** > **Wireframe** > **Point** on the Menu bar.
2. Click on the projected curve.
3. Click on the intersection point between the projected curve and sketch1.



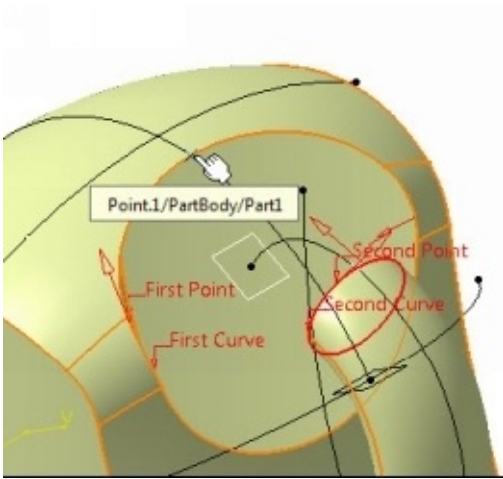
4. Click **OK** to create a point at the intersection.
5. Likewise, create a point on the handle edge, as shown below.



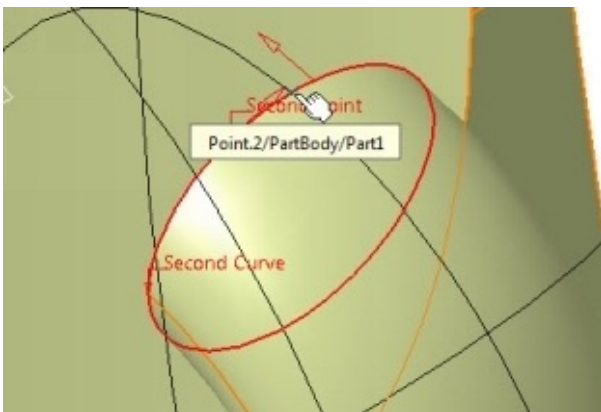
6. On the **Surfaces** toolbar, click the **Blend**  button (or) click **Insert > Surfaces > Blend** on the Menu bar.
7. Click on the projected curve to define the first curve.
8. Click on the main surface to define the first support.
9. Click on the top edge of the handle to define the second curve.
10. Click on the handle to define the second support.



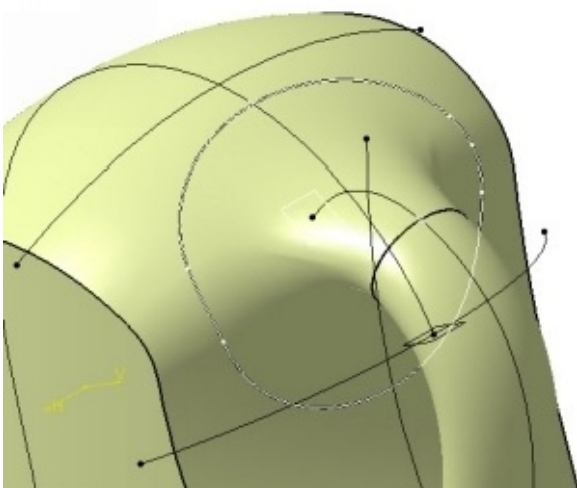
1. On the **Blend Definition** dialog, click the **Closing Points** tab, and then click in the **First closing point** selection box.
2. Click on the point intersection point on the first curve, as shown below.



3. Click in the **Second closing point** selection box and select the intersection point on the second curve.

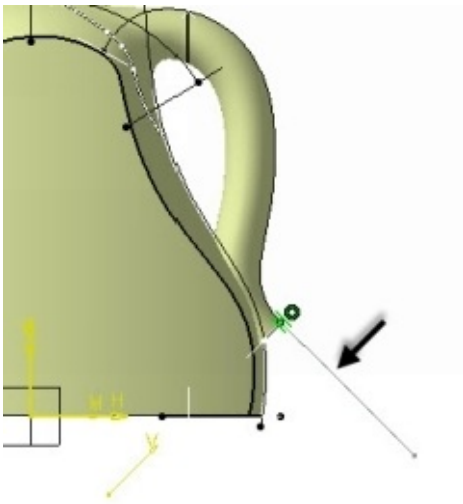


4. Ensure that the arrows on the first and second curves point in the same direction. If not, double-click on them to change the directions.
5. Click **OK** to blend the top portion of the handle with the main surface.

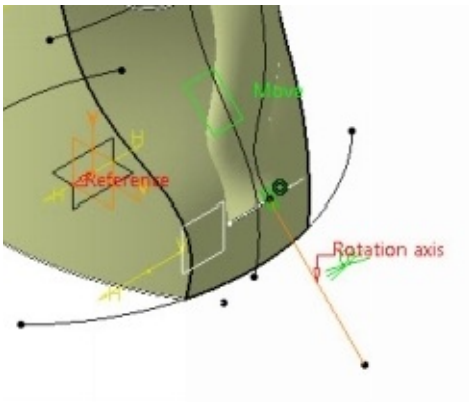


Blending the Bottom handle

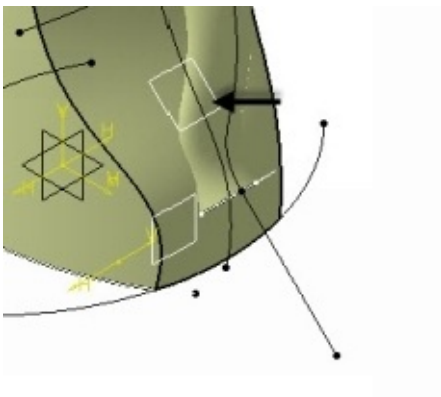
1. Start a sketch on the YZ Plane and draw a tangent to the spline of the handle. Exit the sketch.



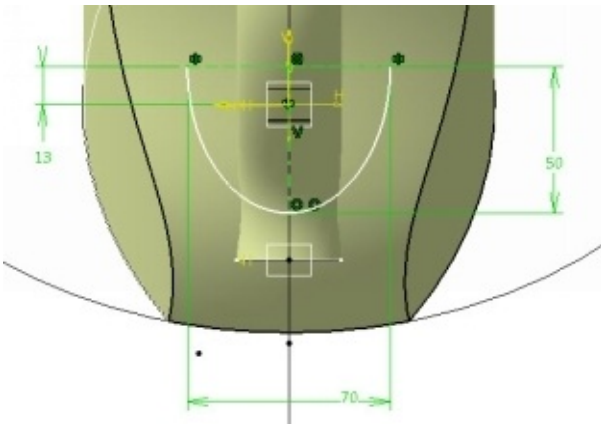
2. Activate the **Plane** command and select **Plane type > Angle/Normal to plane**.
3. Click on the tangent line and the YZ plane.



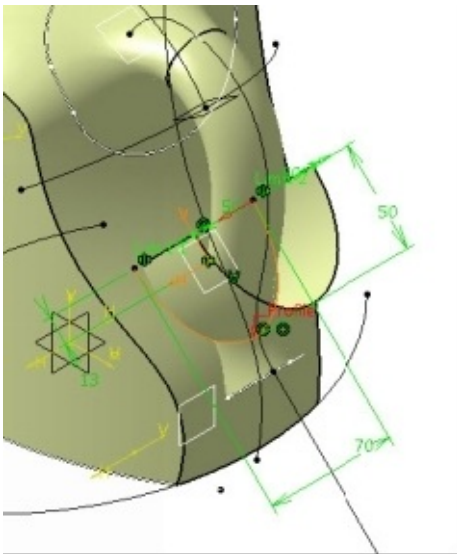
4. Click the **Normal to plane** button, and then click **OK** to create the plane.




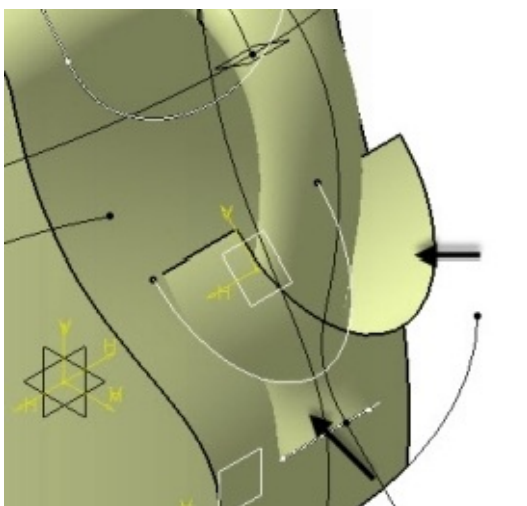
5. Create an ellipse on the new plane and trim it by half. Exit the sketch. Ensure that the sketch lies inside the handle surface.



5. Extrude the sketch up to an arbitrary distance in both the directions.

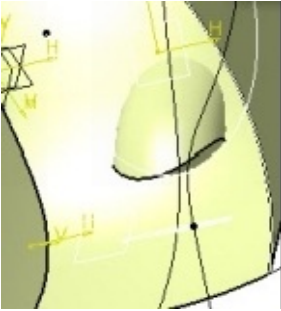
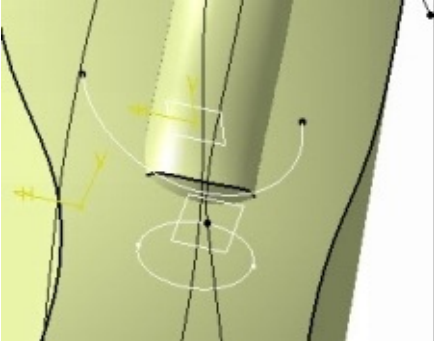



7. On the **Operations** toolbar, click **Split-Trim** drop-down > **Trim**  (or) click **Insert** > **Operations** > **Trim** on the Menu bar.
3. Select **Mode** > **Standard** on the dialog.
2. Click on the portion of the handle and extrude surface, as shown below.

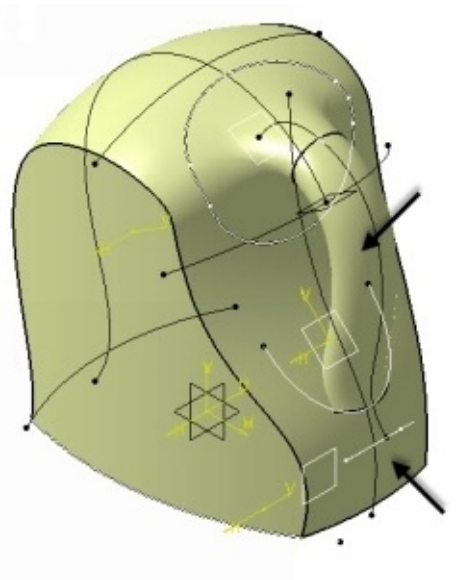


0. Click the **Other side/next element** and **Other side/previous element** on the dialog.

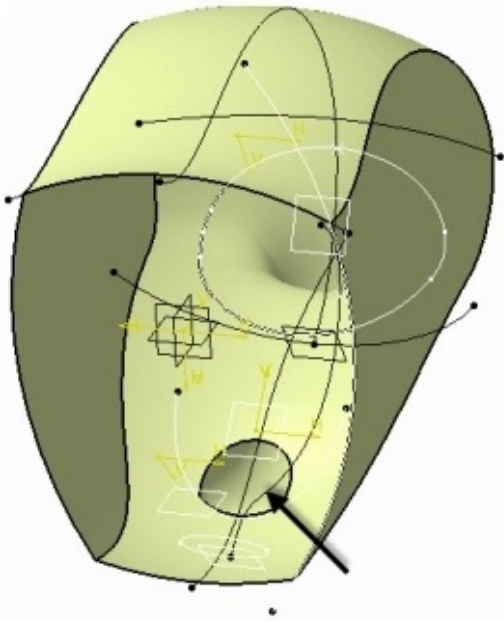
1. Click **OK** to create trim the handle.




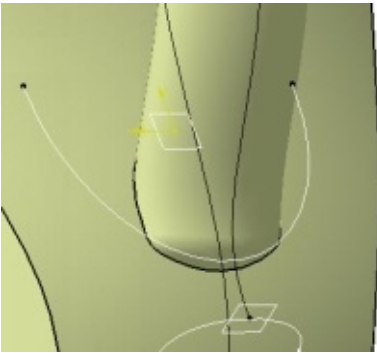
2. On the **Operations** toolbar, click **Split-Trim** drop-down > **Trim**  (or) click **Insert** > **Operations** > **Trim** on the Menu bar.
3. Click on the portion of the handle and main surface, as shown below.



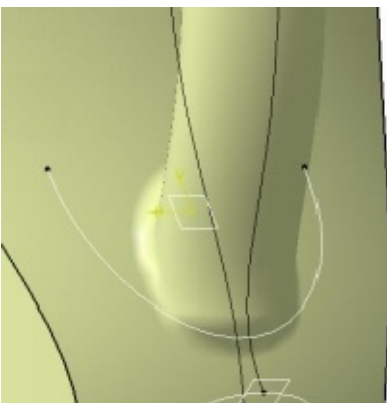
4. Click **OK** the trim the inside portion of the handle.




5. Activate the **Edge Fillet**  command and fillet the edge of the handle. The fillet radius is 6 mm.

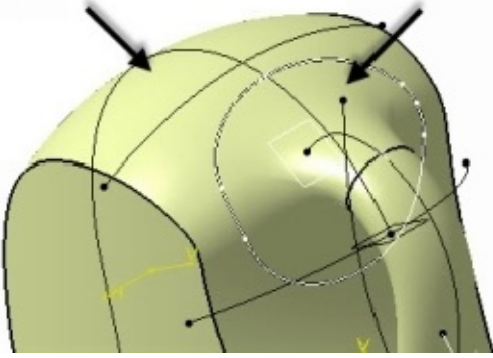


6. Fillet the intersection between the main surface and handle. The fillet radius is 5 mm.

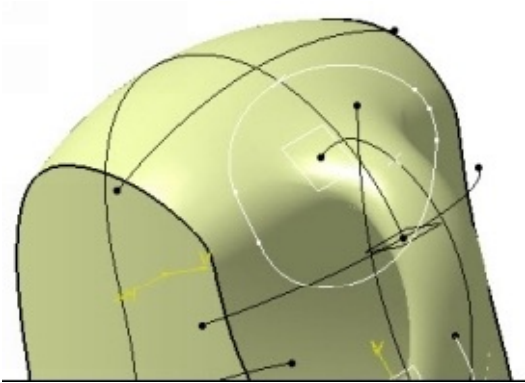


Joining the Surfaces

1. On the **Operations** toolbar, click **Join-Healing** drop-down > **Join**  (or) click **Insert** > **Operations** > **Join** on the Menu bar.
2. Select the main surface and blend surface.

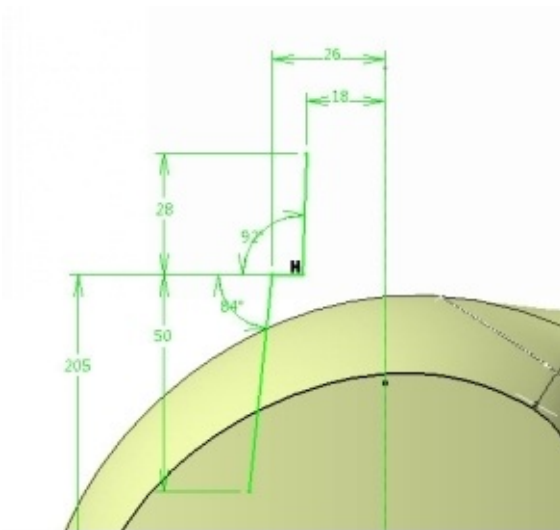


3. Click **OK** to join the surfaces.



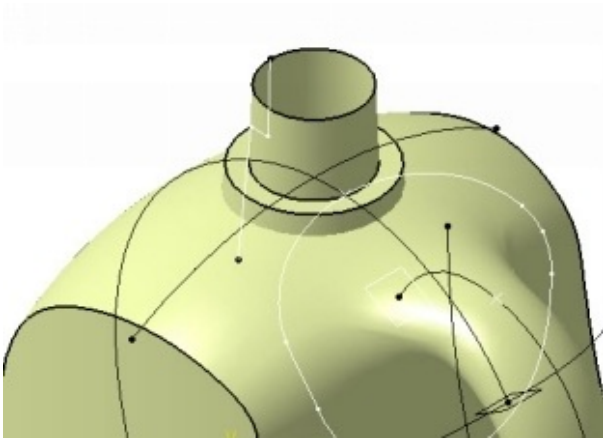
Creating the Neck and Spout


l. Start a sketch on the YZ Plane and draw the sketch for the revolved surface. Exit the workbench.

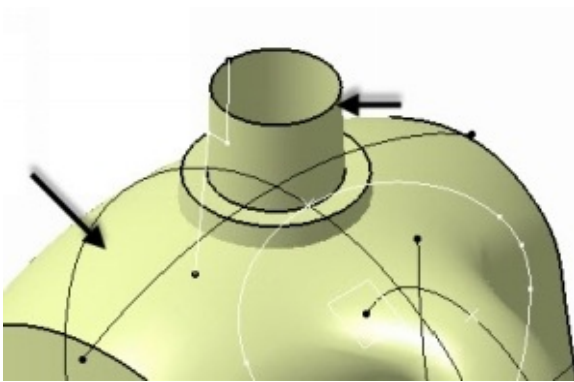


0. On the **Surfaces** toolbar, click **Extrude-Revolution** drop-down > **Revolve**  (or) click **Insert** > **Surfaces** > **Revolve** on the Menu bar.

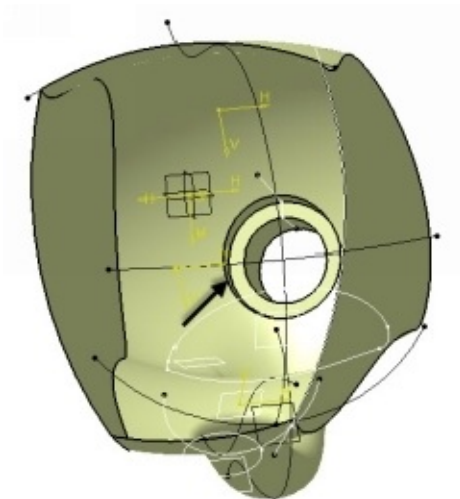
1. Type-in 360 in the **Angle 1** box and click **OK**.




2. On the **Operations** toolbar, click **Split-Trim** drop-down > **Trim**  (or) click **Insert** > **Operations** > **Trim** on the Menu bar.
3. Click on the portion of the neck and main surface, as shown below.

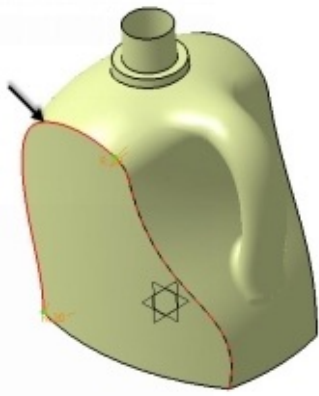



4. Click **OK** to trim the unwanted portion.



Creating the Variable Fillet

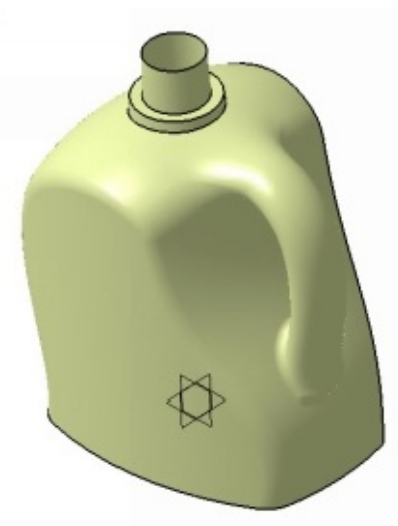
1. On the **Operations** toolbar, click **Fillets** drop-down > **Edge Fillet** (or) click **Insert** > **Operations** > **Edge Fillet** on the Menu bar.
2. On the **Edge Fillet Definition** dialog, click the **Variable**  icon.
3. Select **Propagation** > **Minimal** on the dialog.
4. Click on the edges of the label surface, as shown below.



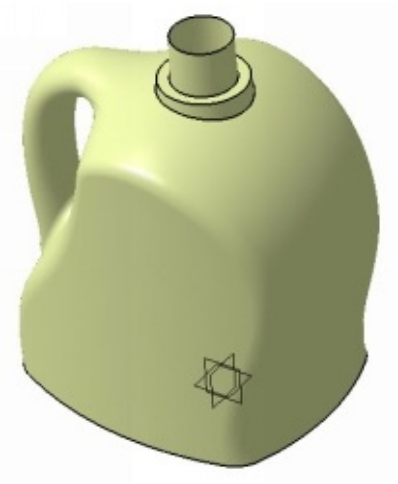
5. Click the  icon next to the **Radius** box. The **Fillet values** dialog appears.
6. Click on the radius value located at the midpoint of the selected edge.



7. Type-in 10 in the **Current value** box, and then click **OK**.
8. Click **OK** to create the variable fillet.

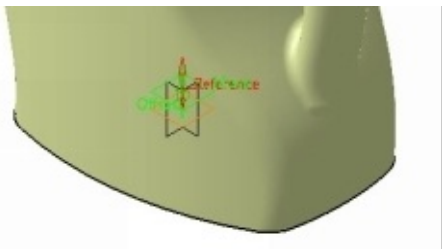


9. Likewise, create a variable fillet on the other label face.

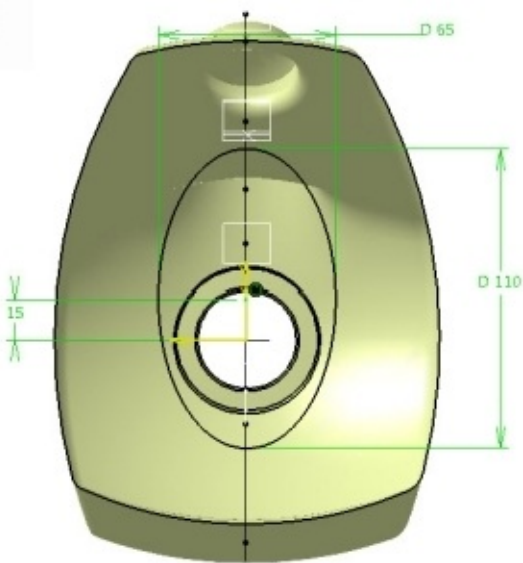



Creating a bump at the bottom

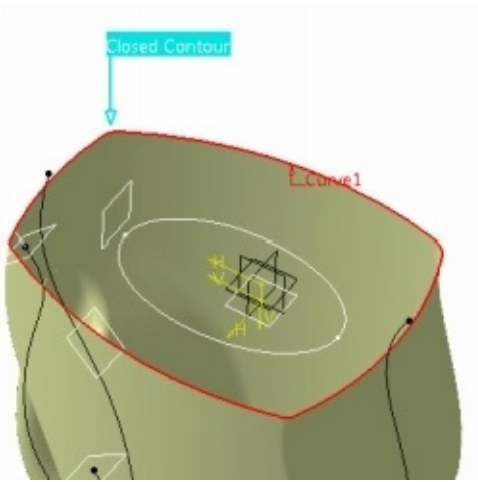
1. Create an offset plane from the XY Plane. The offset distance is 10 mm.



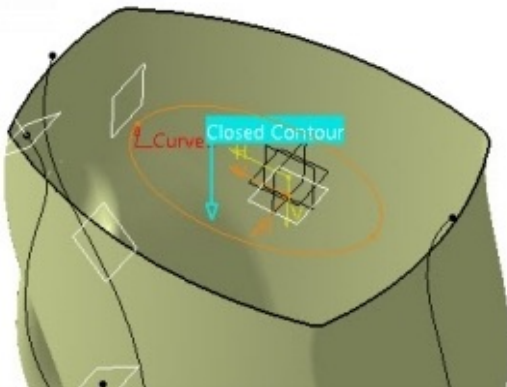
2. Start a sketch on the offset plane.
3. Create an ellipse, and then exit the workbench.



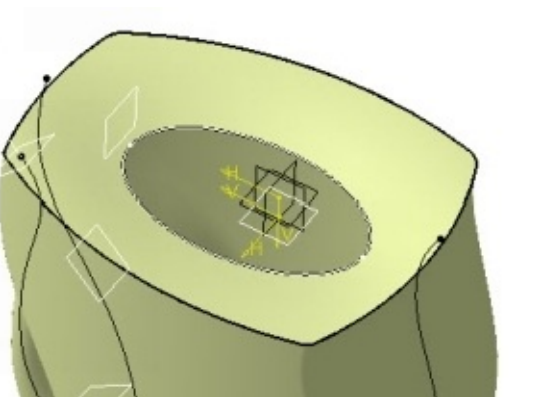
4. On the **Surfaces** toolbar, click the **Fill**  button (or) click **Insert** > **Surfaces** > **Fill** on the Menu bar.
5. Rotate the surface model and click on the outer edges.



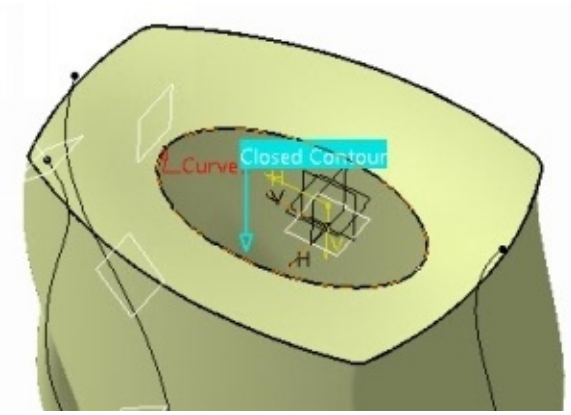
5. On the **Fill Surface Definition** dialog, click the **Inner Boundaries** tab and select the inner loop.



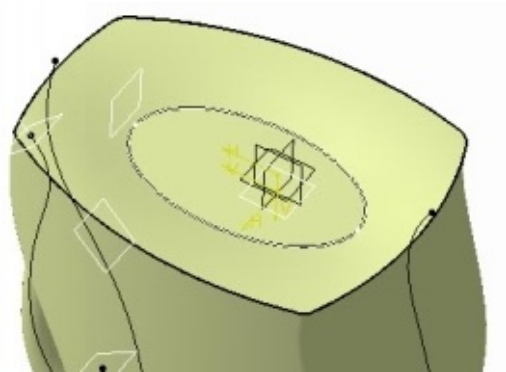
7. Click **OK** to create the fill surface.




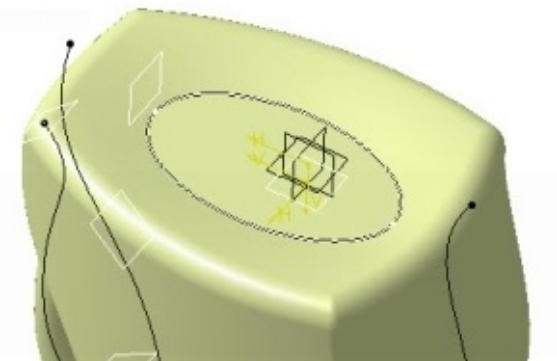
3. Activate the **Fill** command and select the inner loop.



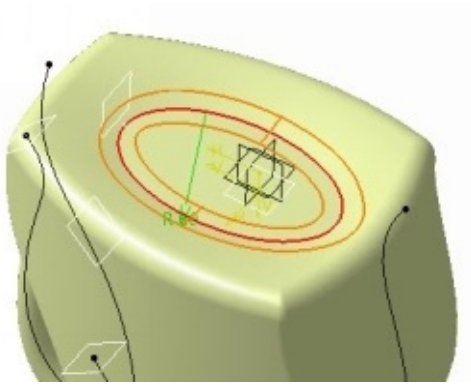
9. Click **OK** to create the fill.



10. On the **Operations** toolbar, click **Join-Healing** drop-down > **Join**  (or) click **Insert** > **Operations** > **Join** on the Menu bar.
11. Click on the main surface and two fill surfaces.
12. Click **OK** to join the surfaces.
13. Fillet the outer edges of the fill surface. The fillet radius is 10 mm.




14. Fillet the inner edges of the fill surface. The fillet radius is 60 mm.



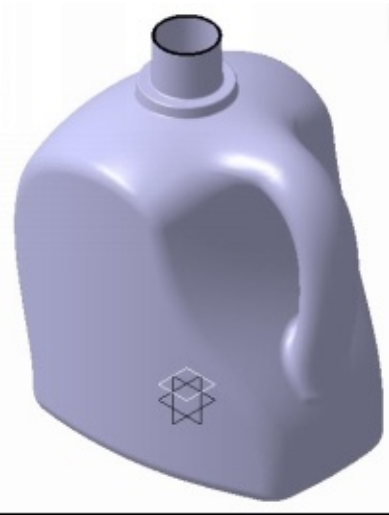
0. Fillet the sharp edges of the neck and spout. The fillet radius is 1 mm.



Adding thickness to the model

1. On the Menu bar, click **Start > Mechanical Design > Part Design**.
2. On the **Surface-Based Features** toolbar, click the **Thick Surface**  button (or) click **Insert > Surface-Based Features > Thick Surface** on the Menu bar.
3. On the **Thick Surface Definition** dialog, type-in 1 in the **First Offset** box, and then click on the surface model.
4. Click **OK** to thicken the surface.
5. Hide the **Join** surface to view the solid model.





5. Save and close the file.

Questions

1. What is the use of the **Join** command?
2. Why do we use the **Fill** command?
3. What are the commands that can be used to delete the openings on a surface?
4. Which commands can be used to bridge gap between two surfaces?
5. Name the command that can be used to trim and join the surfaces.
6. How do you add thicknesses to a surface body?
7. What is the command used to extend surfaces from an edge?
8. How do you split a solid body?
9. What is the command used to offset face?

