

Quicktour

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Quick Tour

Part

Quick Tour

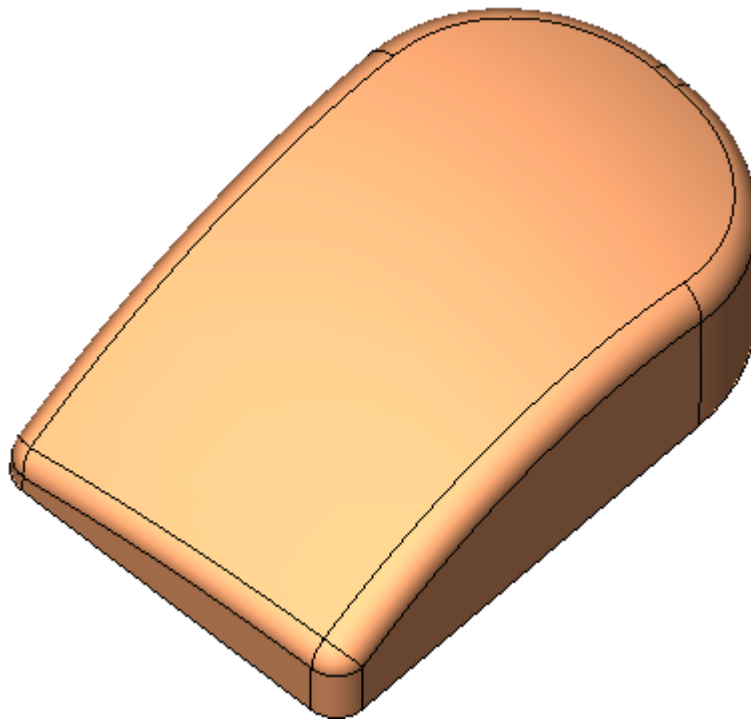
Introduction

The following Quick Tour is a step-by-step basic tutorial, which guides you through the main applications of **3DXpert** 13.0.

Creating the part.

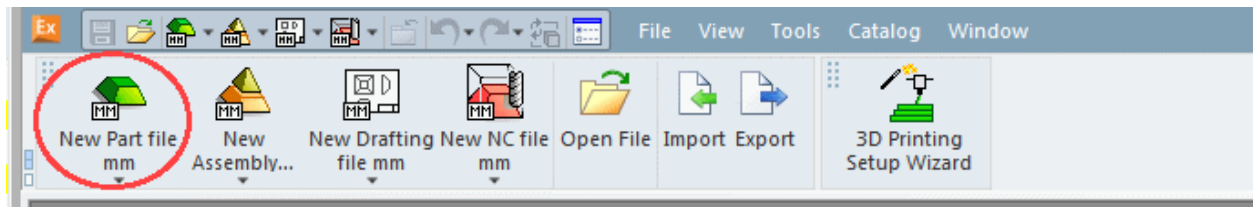
Part Description

In this first chapter we are going to create the cover of a mouse.



Step 1 - Create a New Part File

☞ From the initial toolbar menu, select **New Part File**.



☞ You may also access this option via the Quick Access Commands, or the file menu.



Step 1a - The 3DXpert13.0 Environment

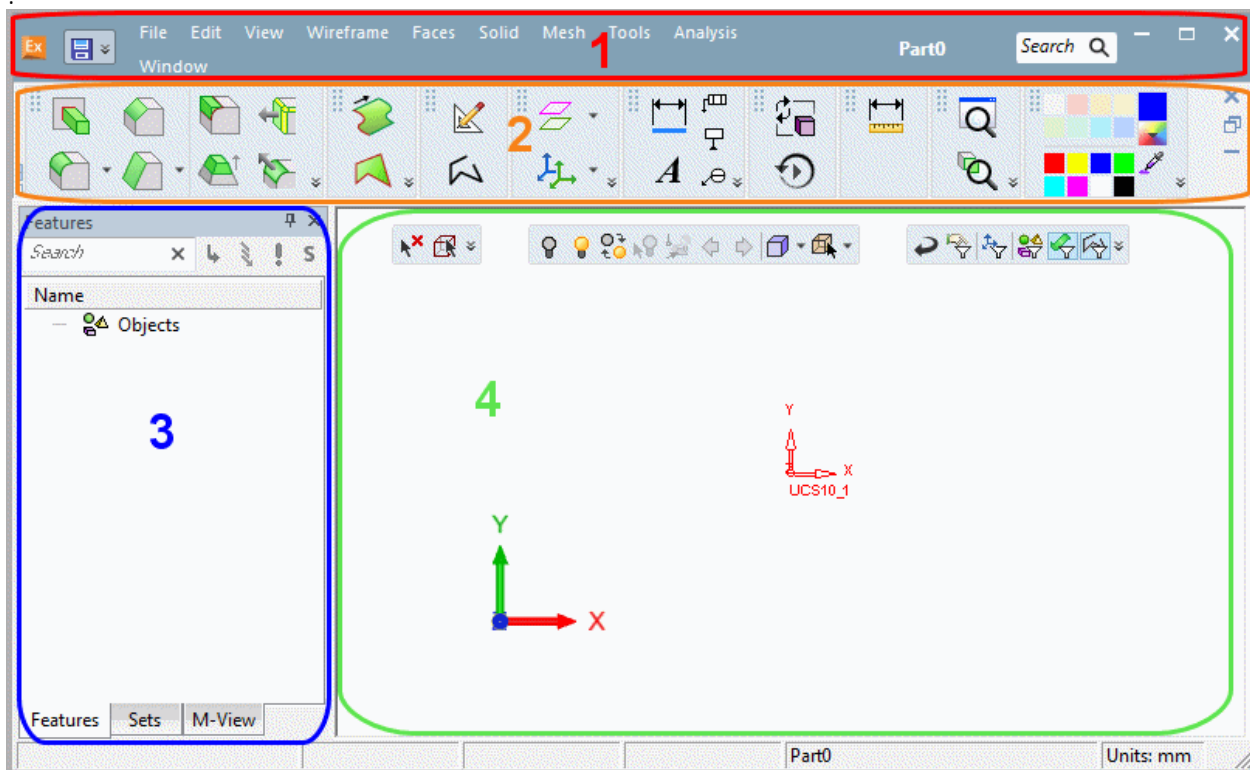


In the picture below the **Menu Bar (1)**, **Icon Toolbar (2)**, **Features Tree (3)** and **Display Area (4)** are displayed.

Display area of all parts and features. In addition, options for many function steps are displayed here.

This tree is specific to the 3DXpert application and to the file you have opened. In Part environment it shows operations and features that were created which can be managed directly from the data trees. Sets and M-Views data also available in this area.

The **Menu Bar** contains all commands specific to the type of **3DXpert** file that you have opened



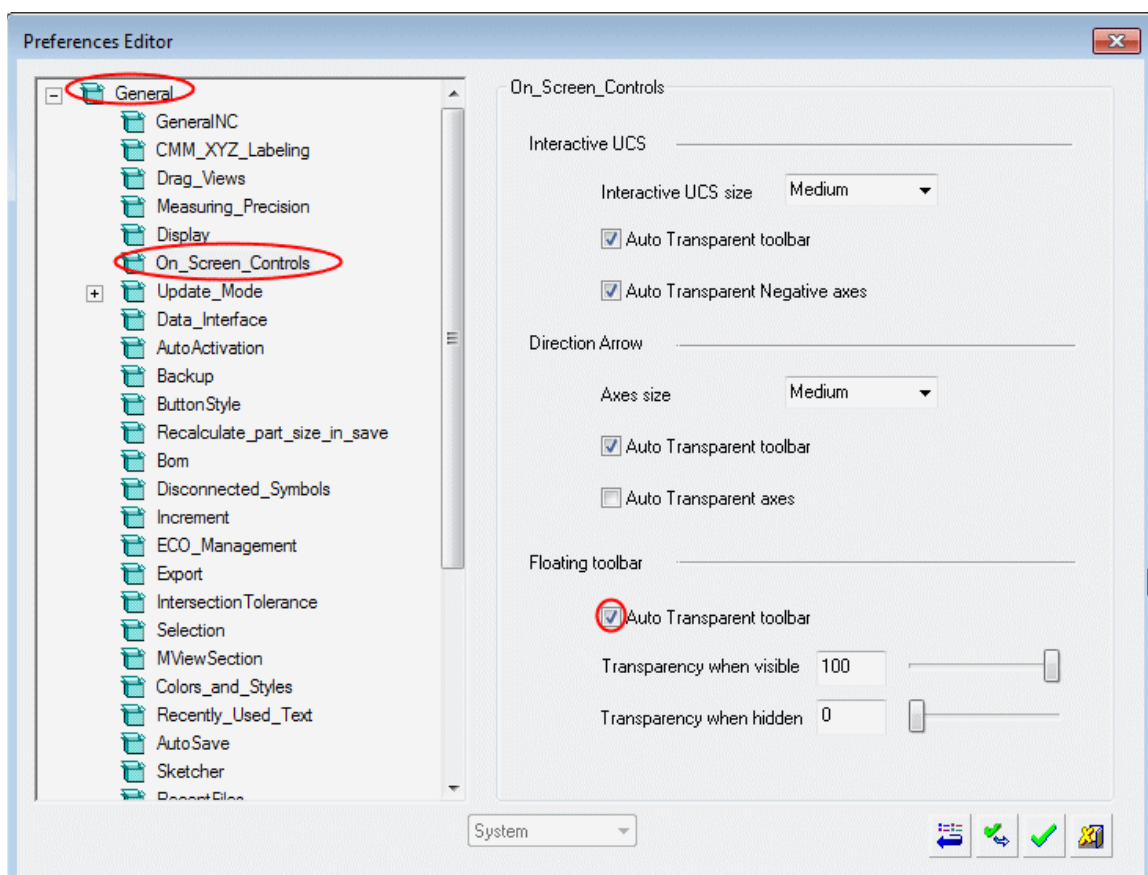
Toolbars

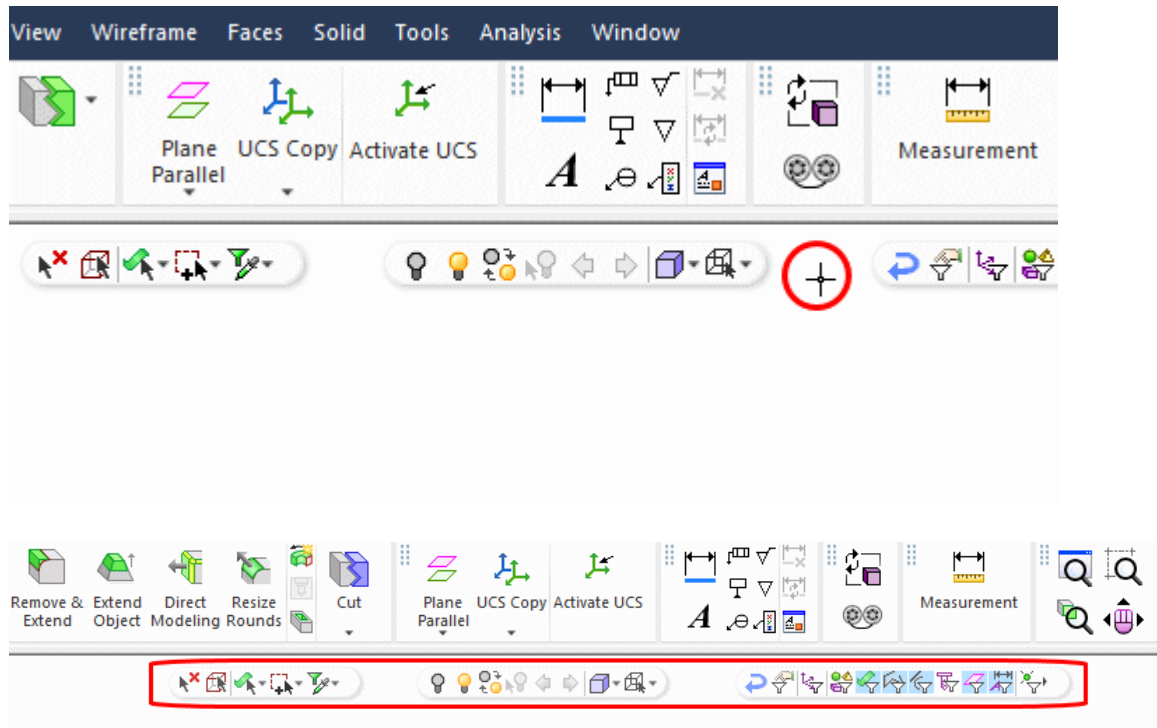
Default toolbars are initially displayed. Some of these toolbars are specific to the Design application or file you have opened.

There are several Toolbars in the 3DXpert environment, three of which are discussed below:

1. *Floating menu*

The *Floating toolbar* can be either transparent unless the user goes *near it*, or opaque in any case. The control of each case and of the transparency percentage can be controlled through the *preferences option*. the floating toolbar is separated into 3 sections:





1. **Filters** *These options enable you to quickly filter entities by geometry. For example if the Faces Filter is turned ON, The selection of faces is enabled.*



2. **Visibility**



Manage visibility and render mode for components.

3. **Selection**



Various filters to quickly select or unselect entities.



Select All.



Clear Selection.



Add Mode selects all entities within a box.



Remove Mode unselects all entities within a box.


















Visible faces Only selects only visible faces.



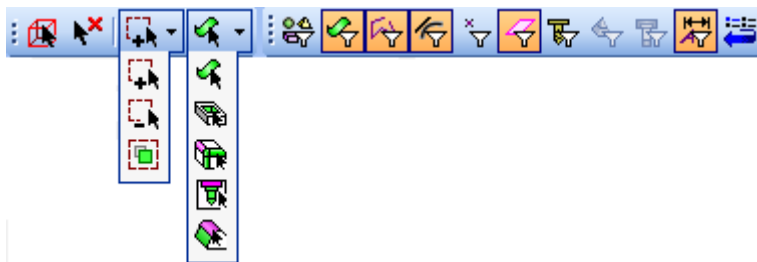
Picked Face Only selects only the picked face



Expand to pocket selects all the faces belongs to the pocket

	Expand to round selects the entire rounded area automatically
	Expand to Hole extends the selection to include a hole.
	Expand to Smooth extends the selection to include all smooth faces
	Objects.
	Faces.
	Sketches and Composites.
	Edges and Curves.
	Points.
	Datum.
	Threads
	Component. (In Assembly environment)
	UCS manager
	Enable/Disable keep filters
	Symbols (PMI)
	Reset

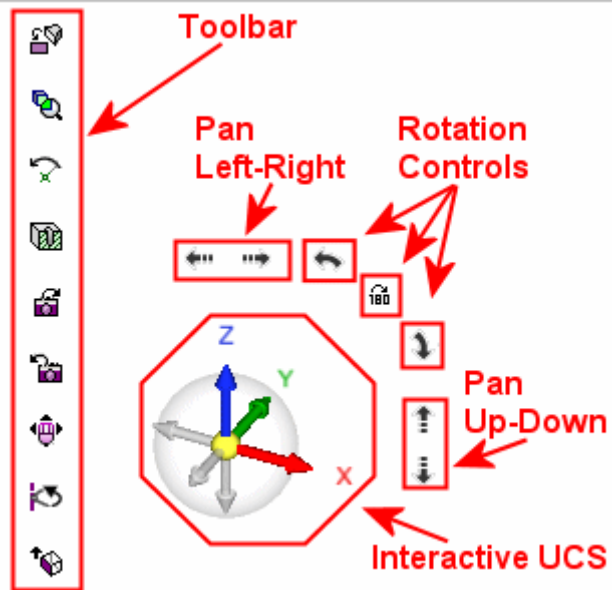
2. Quick Access toolbar



The Quick access menu is a set of commands such as Save, New document, Undo/Redo often used.

3. Interactive UCS

The Interactive UCS and associated toolbar are displayed at the bottom left corner of the 3DXpert display area (when a file is open). They are used for easy control over rotation, orientation and the direction arrow. The interactive UCS is explained in detailed in the Environment tutorial step 9.



For further information about the **3DXpert** toolbar, access the OLH.



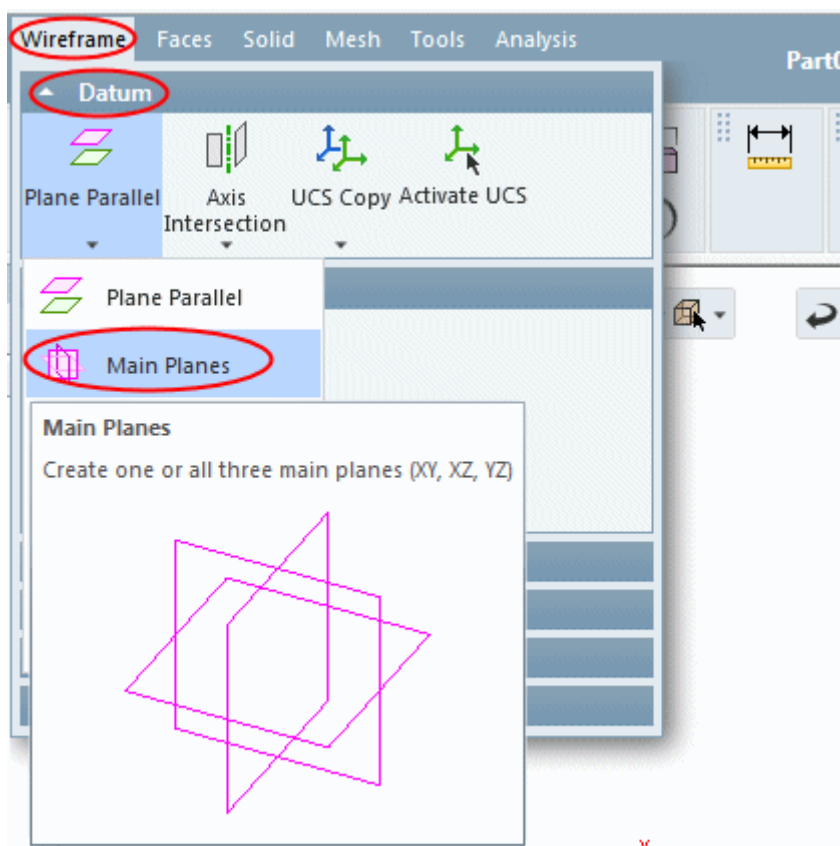
	Zoom All		Edges
	Zoom Selection		Global Render (Shade)
	Zoom Window		Wireframe
	Zoom/Pan and Rotate		Hidden Lines
			Partial Hidden Lines
			Mixed Render Mode
			Local Render (Shade)
			Wireframe
			Transparent
			Render Mode Dialog

Step 2 - Create a Sketch

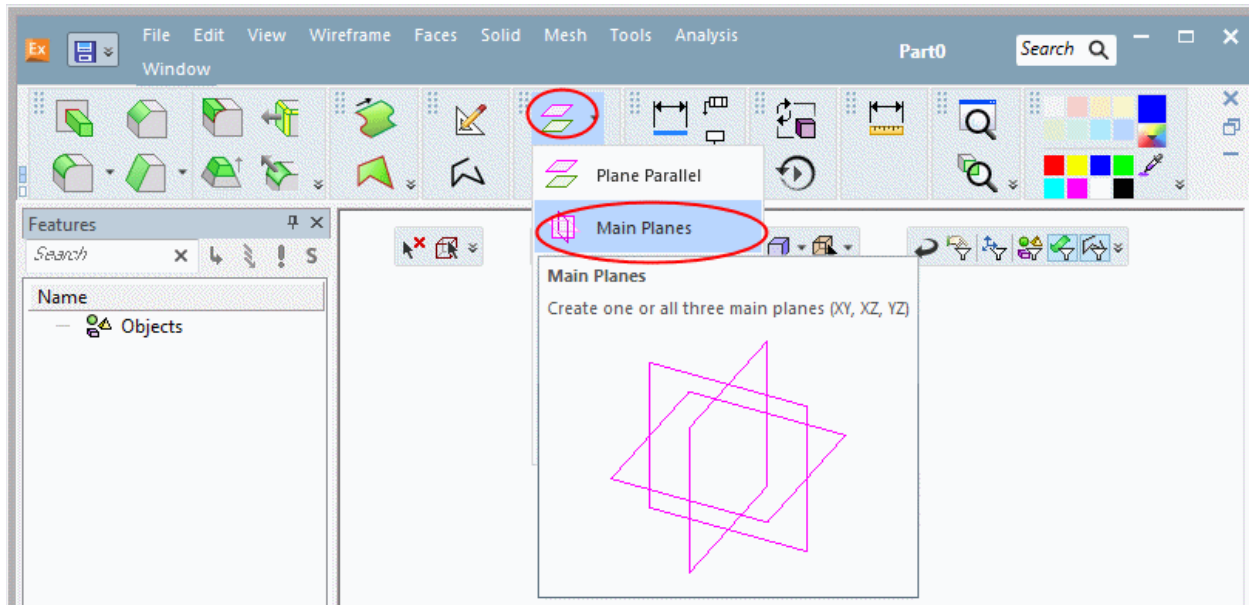
Before creating the first sketch we will define main planes for the model.


 From the ribbon toolbar select the **Plane Parallel/Main Planes** option. You may also access this option via *Wireframe/ Datum menu*.


Select the **Plane Parallel/Main Planes** option.





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 Pick the displayed UCS.



 At the bottom-left of the Cimatron window the *appropriate prompt* for the function is displayed.

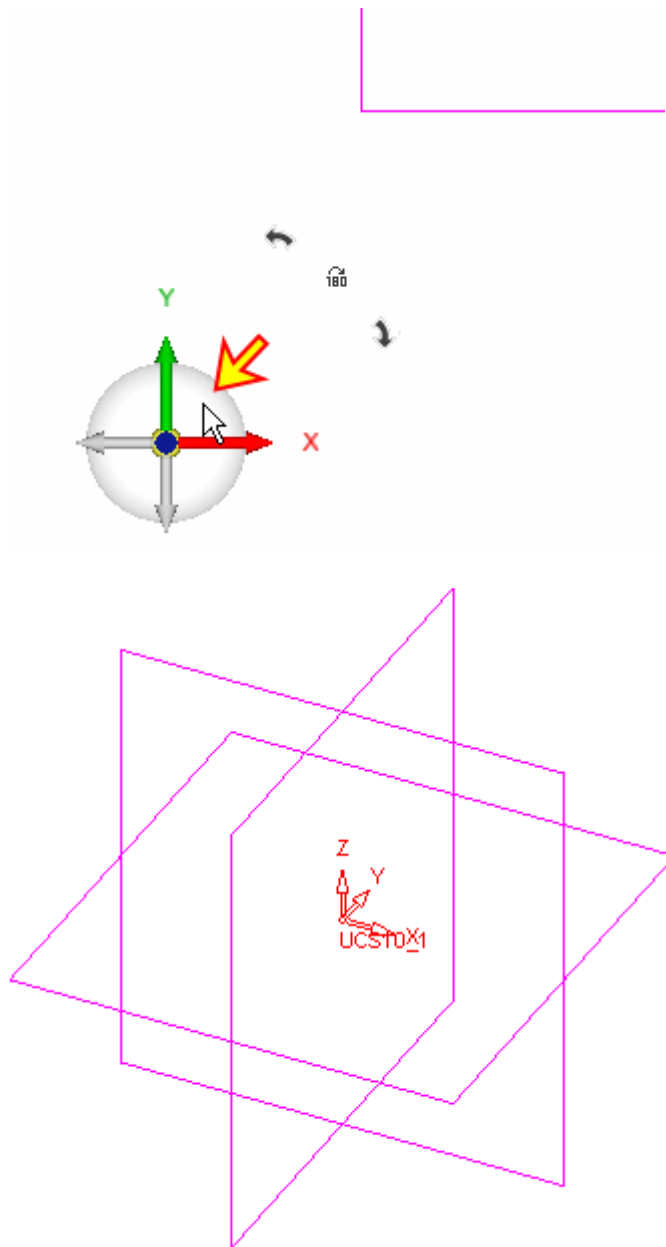


 To approve the operation, click the **OK**  button in the Feature Guide in the upper-left part of the screen.

 You may also press the right mouse button to display the Feature Guide in the display.

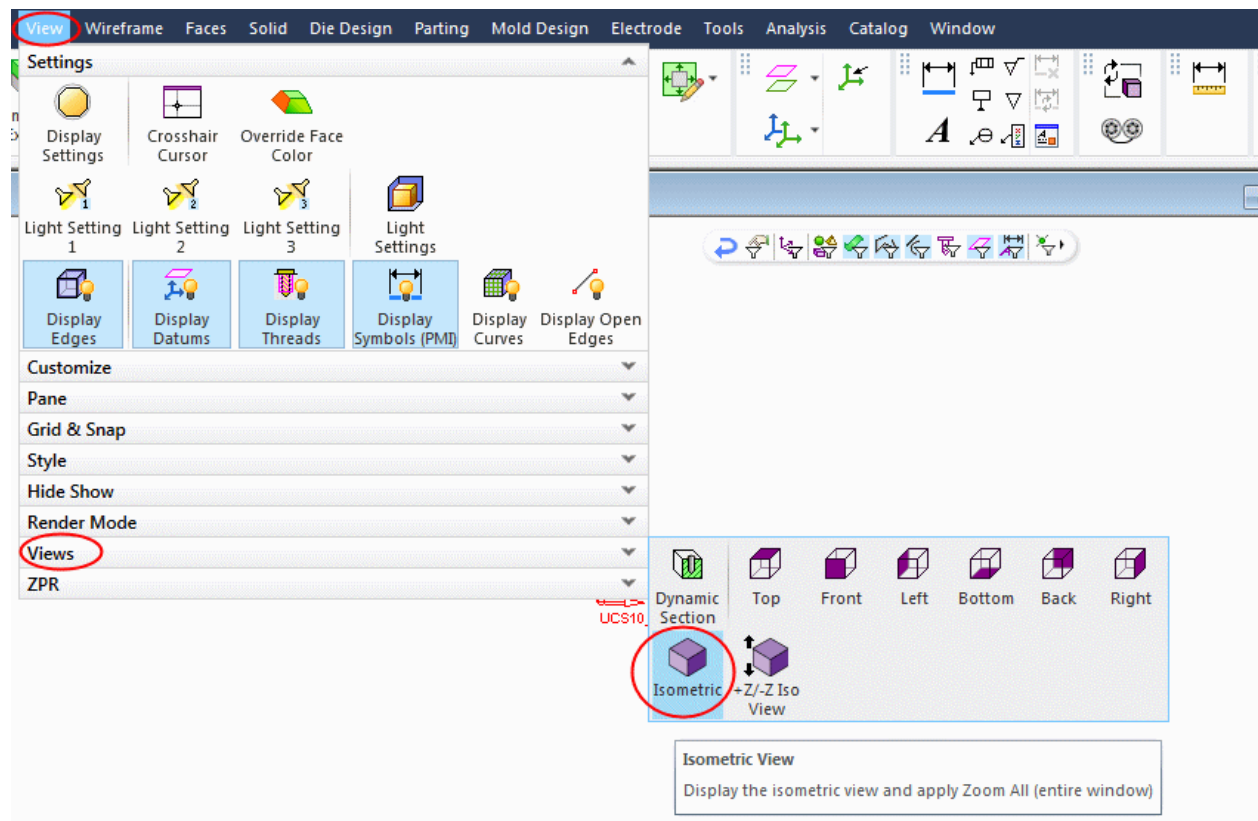


 Move the mouse close to the Interactive UCS. Click somewhere inside the bubble to display the isometric view. For further information of Interactive UCS and displaying views look at the help .



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Note that you may also use *Isometric View icon* to display the following view of the main planes. Right click the icons toolbar and select to display the main views:



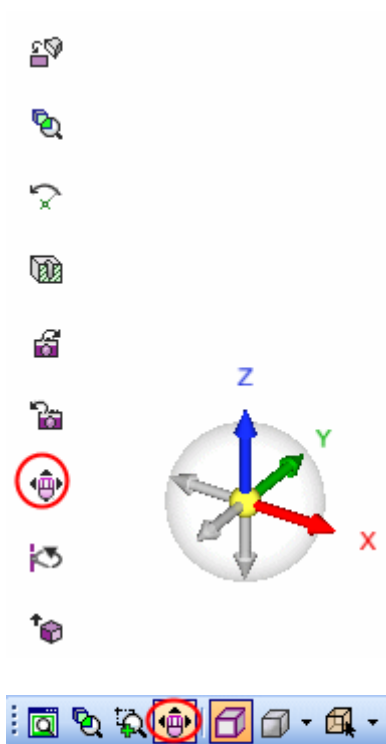
Practice manipulating the Cimatron work area by using the three manipulation options as describe below:

The **Zoom**, **Pan** and **Rotate** can be operated in several ways:

Zoom in Zoom out:

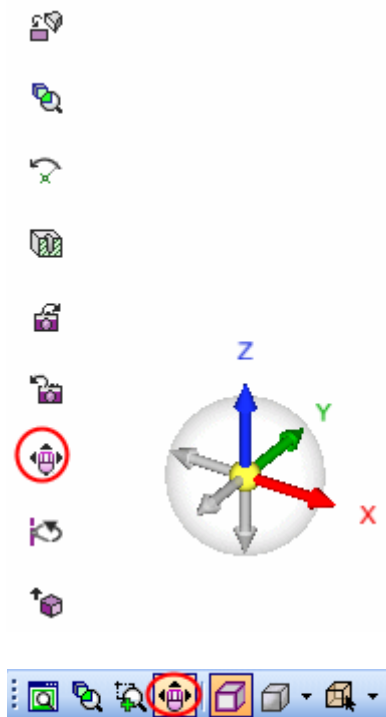
1. Press the **Ctrl + Right** mouse button mouse and Move the mouse up to **zoom in** and down to **zoom out**.
2. Press the **Ctrl + mouse wheel** and move the wheel forward to **zoom in** and backwards to **zoom out**.

3. Press the **Dynamic ZPR mode** either from the *toolbar icons* or place the mouse button right to the interactive UCS, and press the *Dynamic ZPR* button. Press the right button mouse and Move the mouse up to **zoom in** and down to **zoom out**.



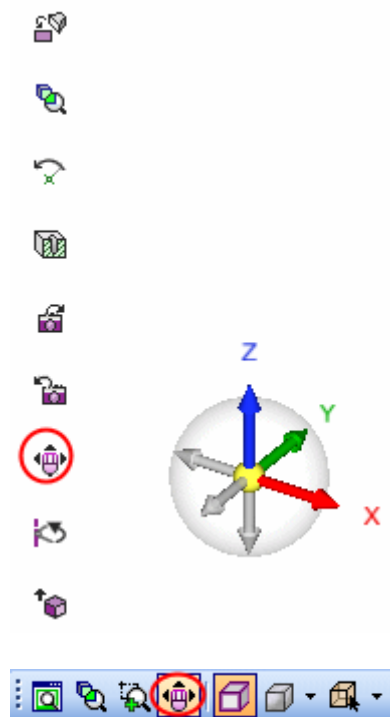
Pan:

1. Press the **Ctrl + Middle** mouse button and move the mouse to dynamically **pan**.
2. Press the **Dynamic ZPR mode** either from the *toolbar icons* or place the mouse button right to the interactive UCS, and press the *Dynamic ZPR* button. Press the middle mouse button and move the mouse to dynamically **pan**.



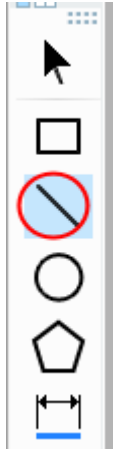
Rotate:

1. Press the **Ctrl + Left** mouse button and move the mouse to dynamically **rotate** around the center of geometry.
2. Press the **Dynamic ZPR mode** either from the *toolbar icons* or place the mouse button right to the interactive UCS, and press the *Dynamic ZPR* button. Press the **left** mouse button and move the mouse to dynamically **rotate** around the center of geometry




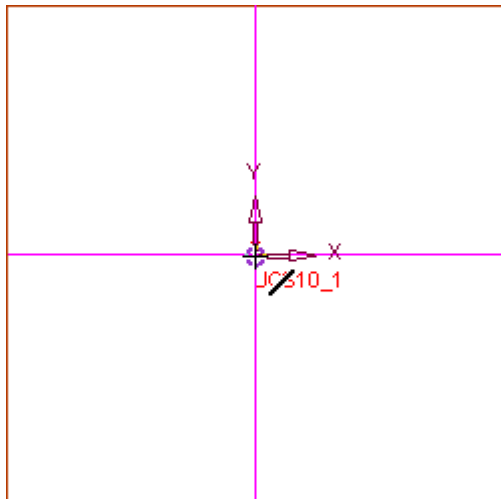
Step 2b - Create a Sketch


In the Sketcher toolbar the **Line** option is selected as the default drawing tool.



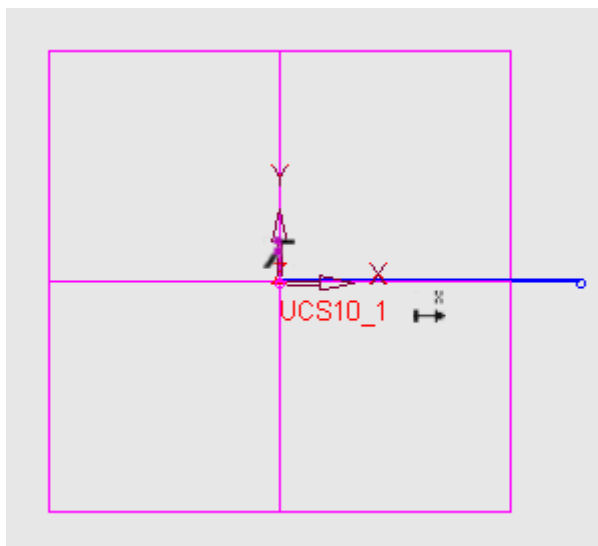
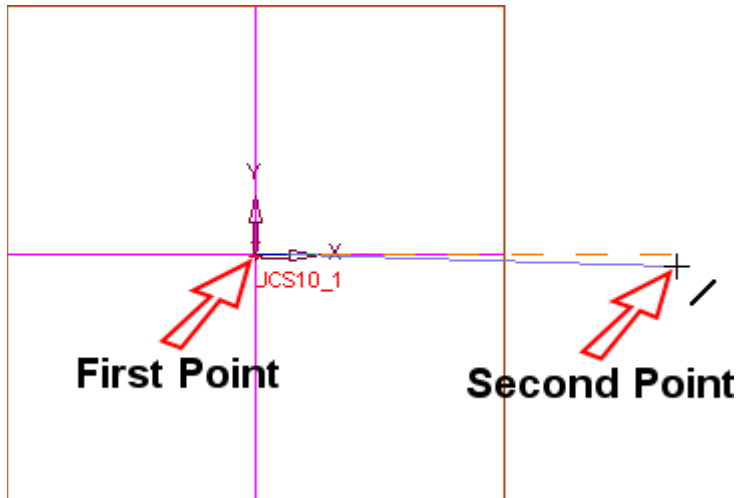
To create the first line, two end points have to be defined.

 To define the first end point, move the mouse over the *UCS origin*, point and click it.




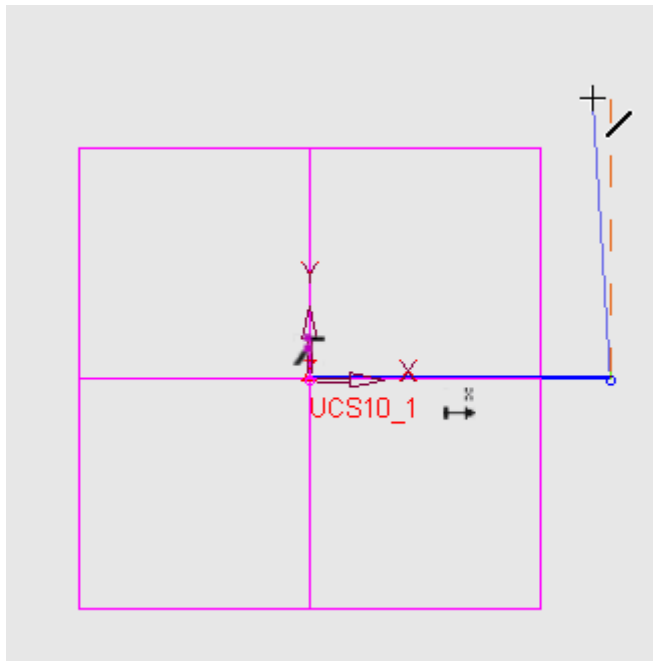
 Move the mouse to the right, make sure that the dashed red horizontal line i.e. horizontal **constraint** appears, and then pick the second end point.

When creating geometric entities, lines or points are highlighted to help you align entities relative to the Sketcher UCS or other entities.

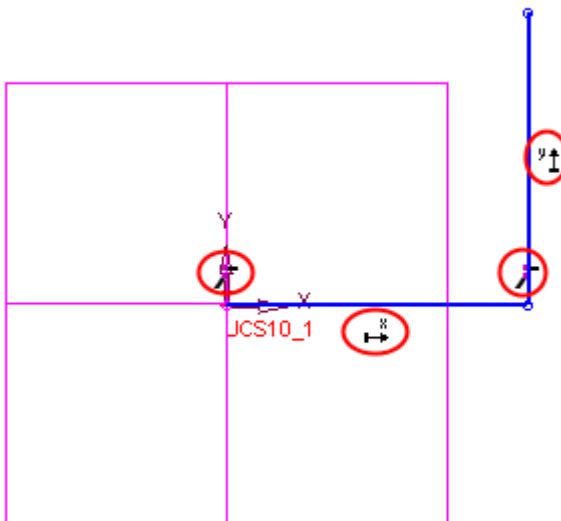


Step 2c - Create a Sketch

 Drag the mouse to the third point as shown in the figure below and **click it**.

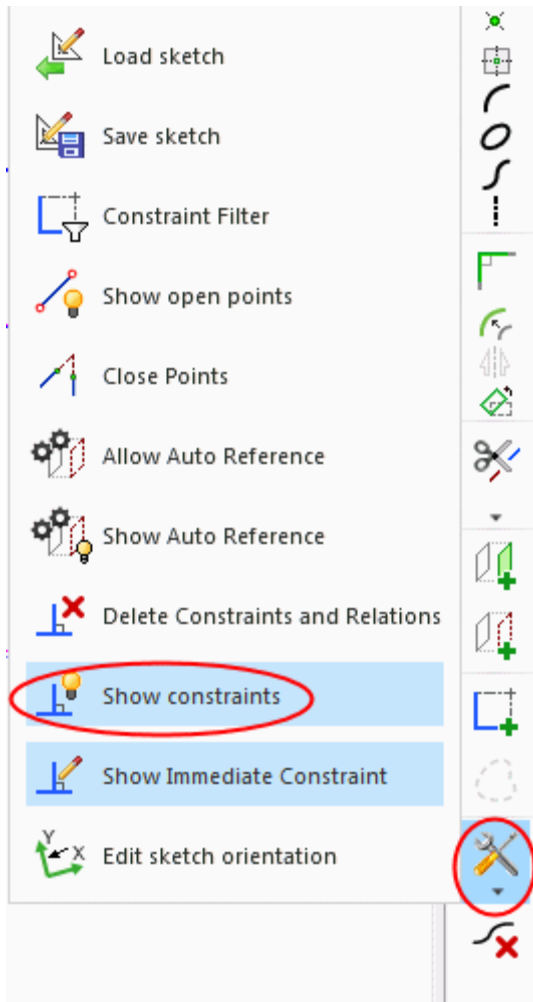


You may work with the **Show Constraints Constantly** mode to see the constraint while creating the sketcher. By default, this tool is turned on when you enter sketcher, enabling you to view all the constraints in the sketch.




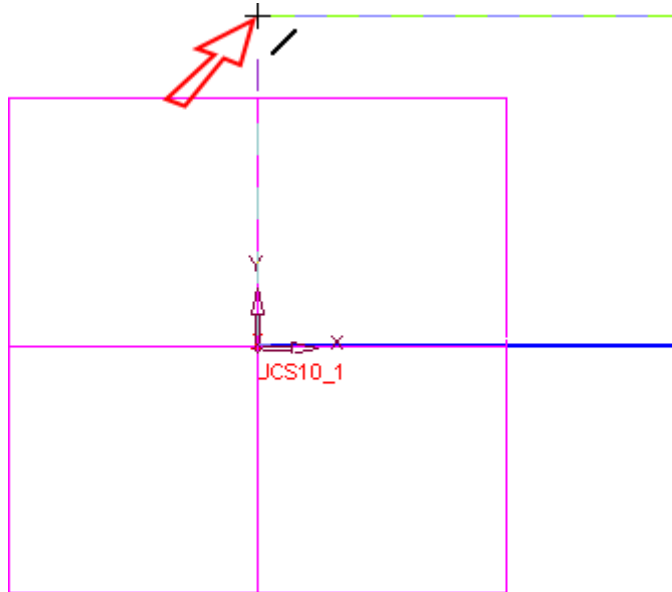
 Meanwhile turn the **Show Constraints Constantly** off, by selecting the **Sketcher**


Tools icon.  and then press the **Show Constraints** icon to deactivate it.



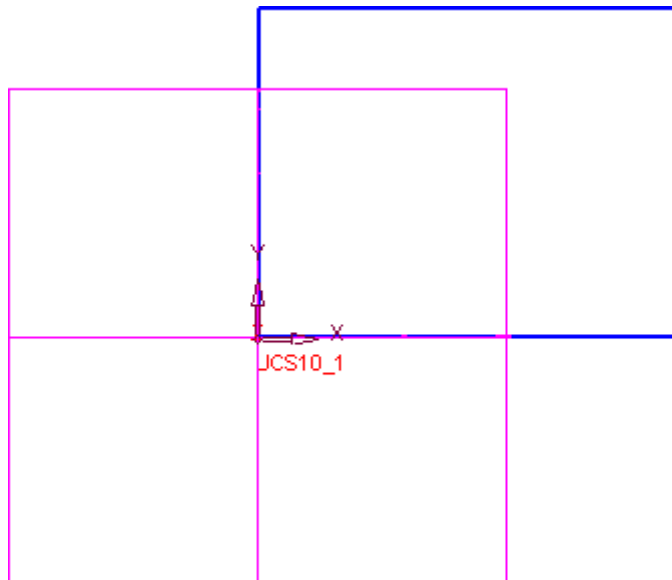
Step 2d - Create a Sketch

 Drag the mouse to the left until both the X horizontal & Y vertical constraints (dashed lines) appear. Click to define the end point of the third line.




 Drag the mouse downwards and pick the UCS origin point to complete the creation of the last line.

 **Exit** the line function by pressing the Middle Mouse Button.





Step 2e - Create a Sketch

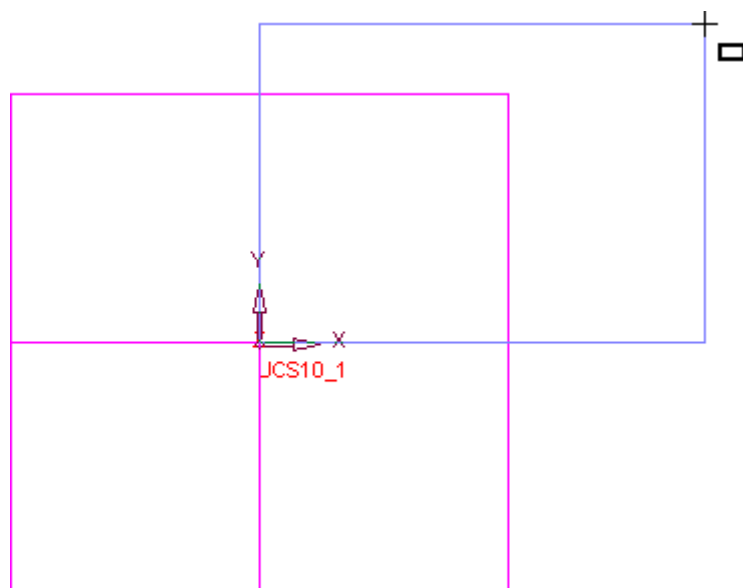
Now that you have learned the basic sketch features (line and constraints), we will recreate the sketch using the rectangle option.

 To undo the sketch created, either press the *Undo icon* four times, OR press the left and middle mouse buttons together four times.





Sketch a rectangle


 Press the *rectangle icon*  in the Sketcher Toolbar. Pick the UCS origin as the first point, move the mouse to the right and upwards and Pick the second corner point i.e. top right of the rectangle.

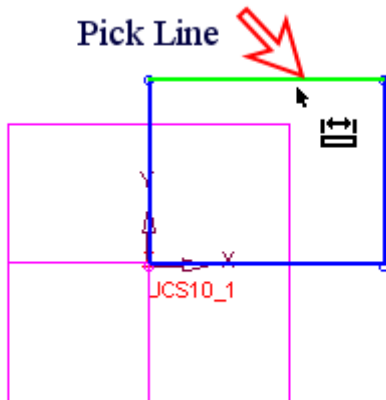


Step 2f - Create a Sketch

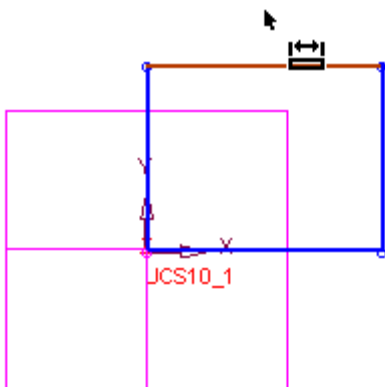
Set Dimensions for the sketch


 Click the dimension icon  from the Sketcher Toolbar.

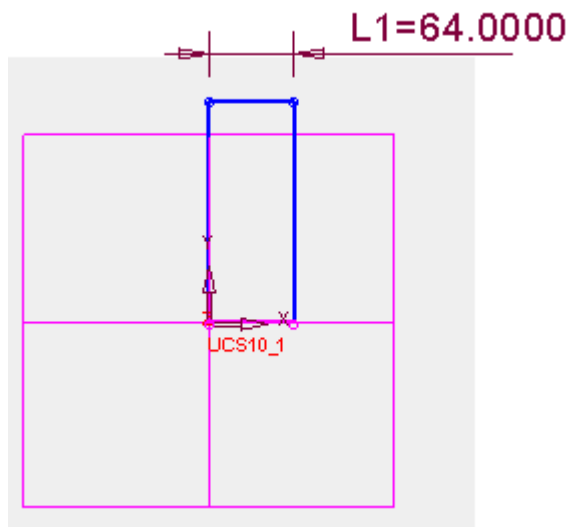
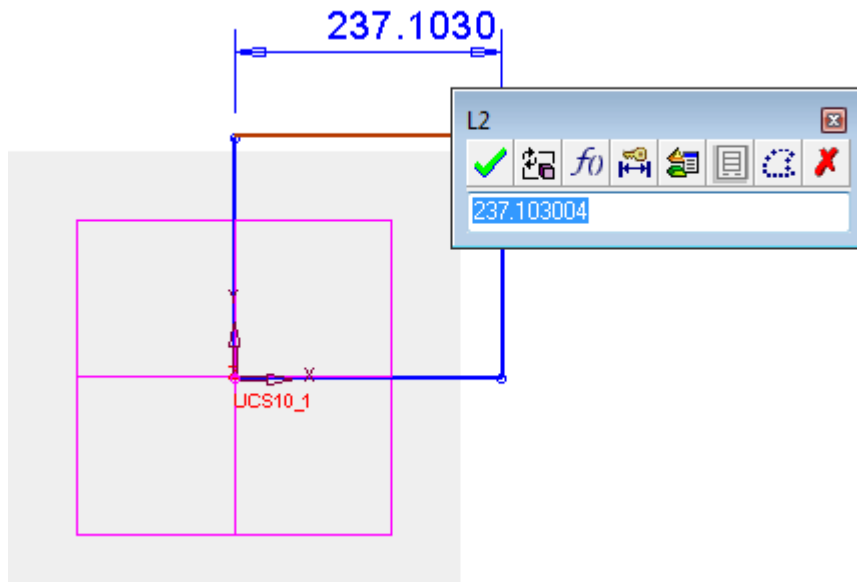
 Pick the line as shown below:




 Move the mouse upward and click a point above the line, to locate the dimension.

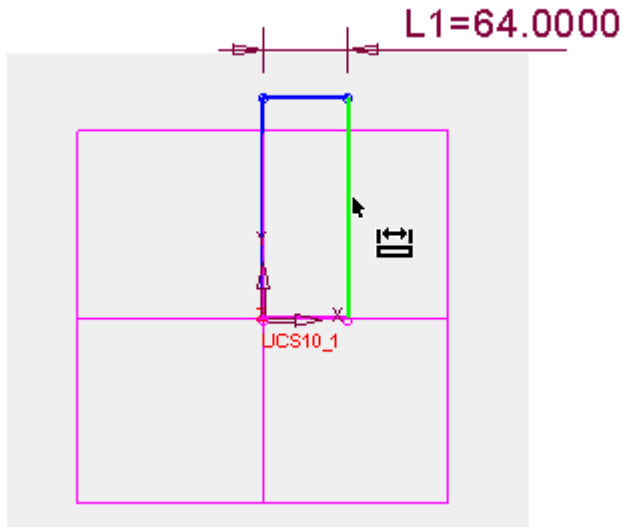



 In the dialog, enter the value **64** and press Enter.

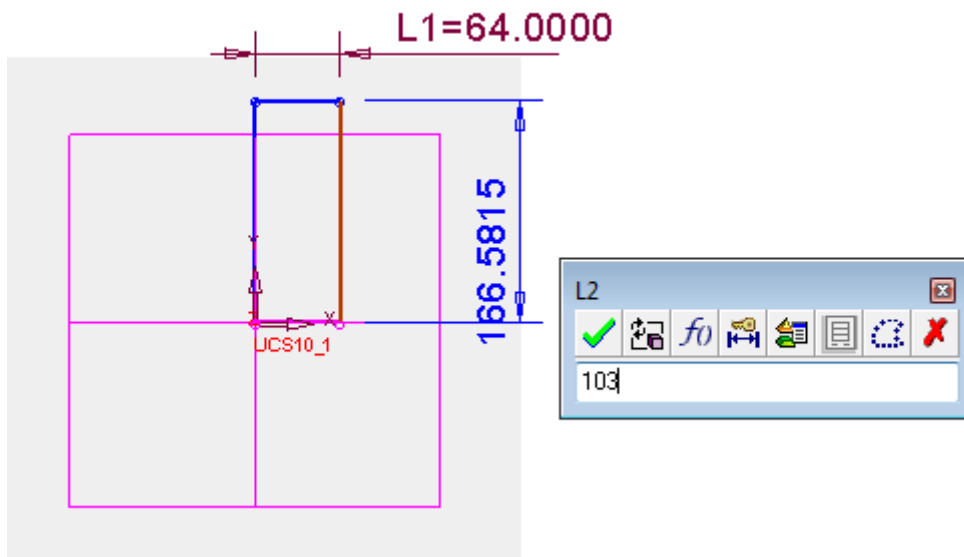


Step 2g - Create a Sketch

 Pick the second line, as indicated by the cursor:



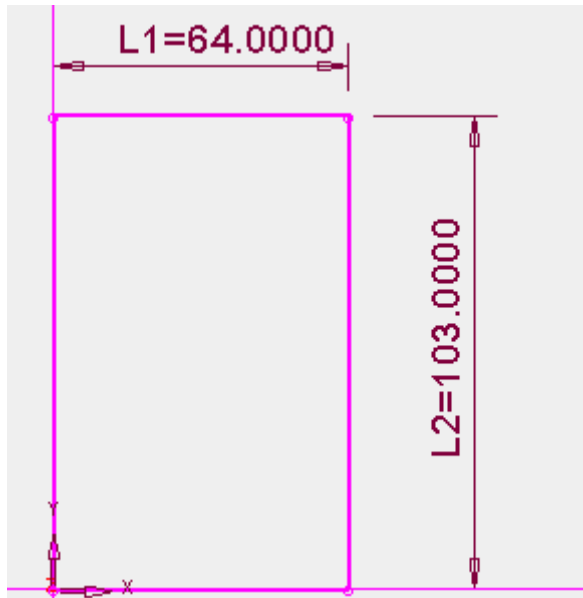
 Place the dimension as shown and set its value to **103**.



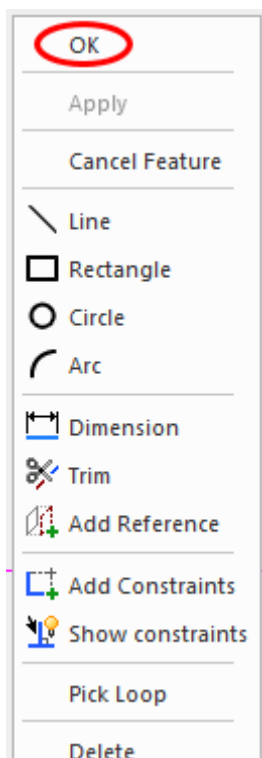
💡 Click on the Zoom All icon



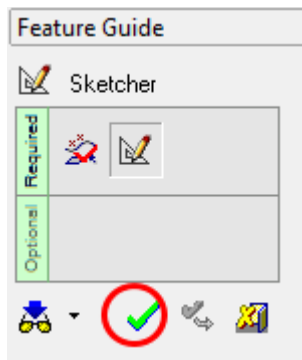
so that the sketch fits the entire display.




👉 Approve the Sketcher by clicking the **OK** icon in the feature guide, , or by right clicking in the display area and selecting the option, **OK**.




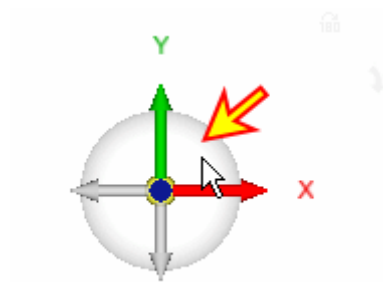
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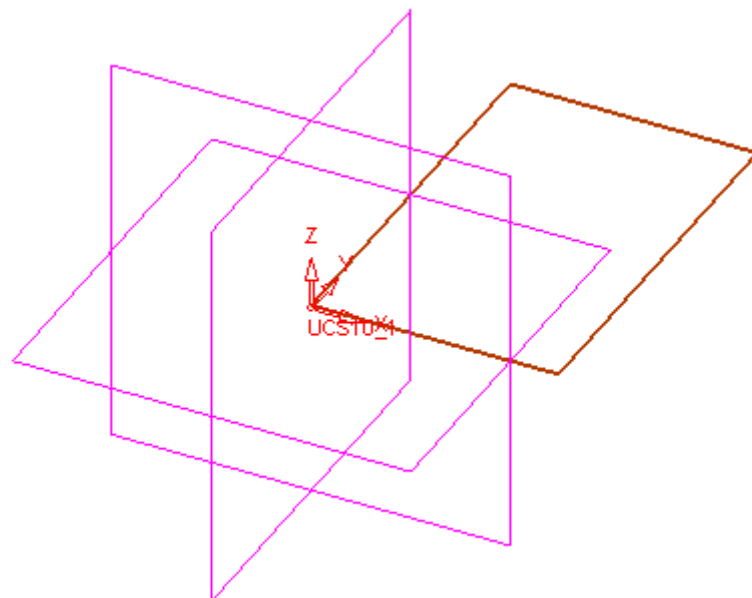
 In the above example the dimensions were updated automatically. There is also the option to enter a dimension and manually update it at a later stage. Click on the icon below for a further explanation of the automatic and manual update options.

Step 3 - Extrude the Sketch

 Change to **ISO** view.

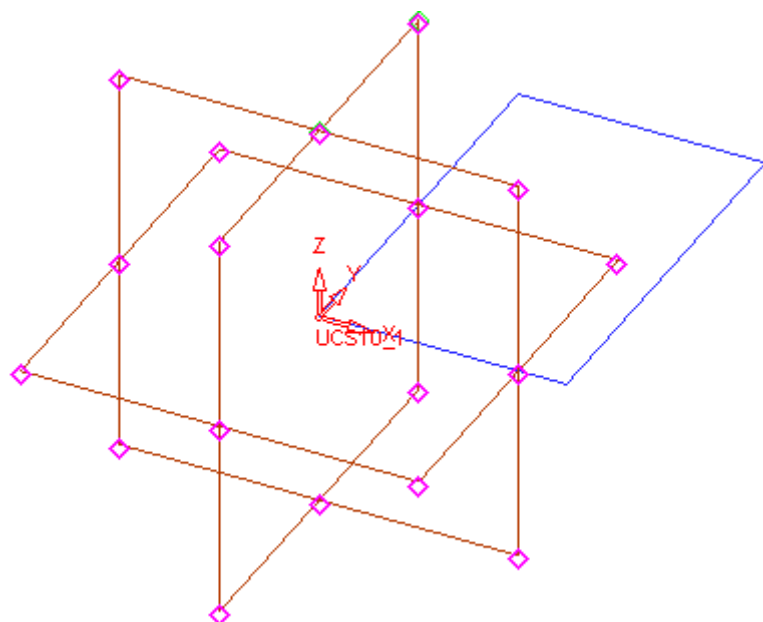



 After exiting the Sketcher the sketch is automatically selected (highlighted).

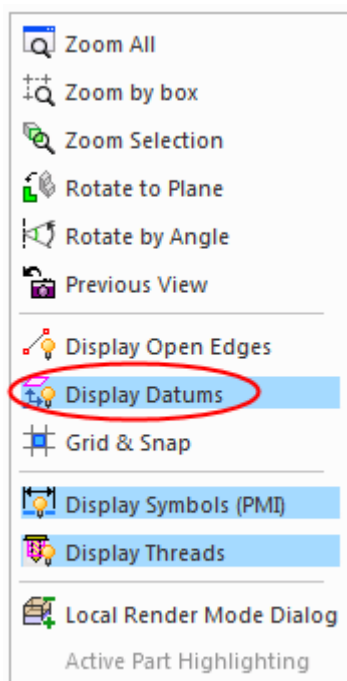


See the text below about hiding the planes to get a clearer display:


 Unpick the sketch (by clicking on it), click the planes and then click the hide icon .

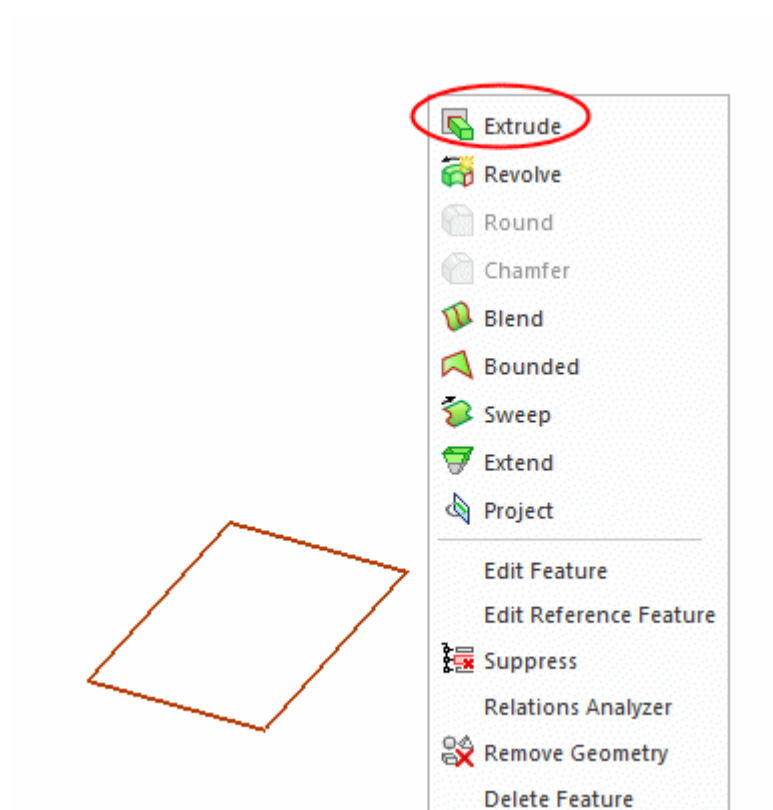


 To hide ALL the existing and future Datums, press the Right and Middle Mouse Buttons together, and unselect **Display Datums**.



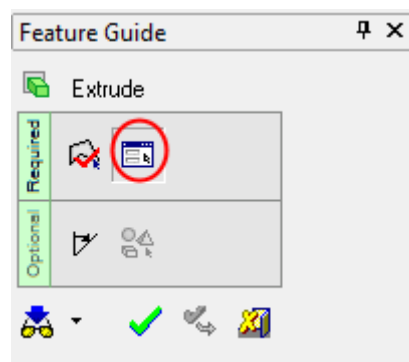
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 Pick the sketch, Right Click in the display area and select New Extrude.

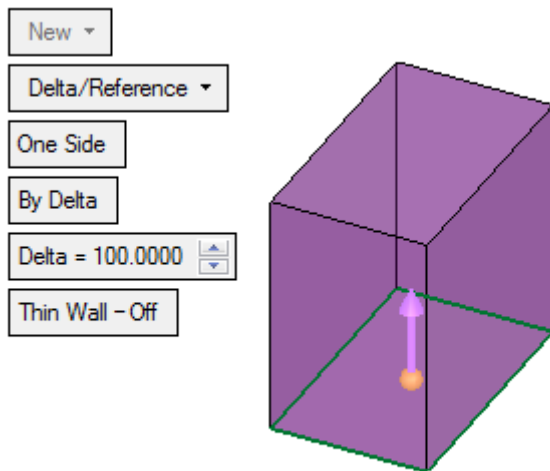


 You may create the extrude by clicking **Solid/Extrude** from the main menu or from the Icon Toolbar.

Note that the first required step in the feature guide has a check mark. This is because the selection of the contour is already completed. The current required step is the second step where you are required to set the extrude parameters.




 Accept the default parameters.



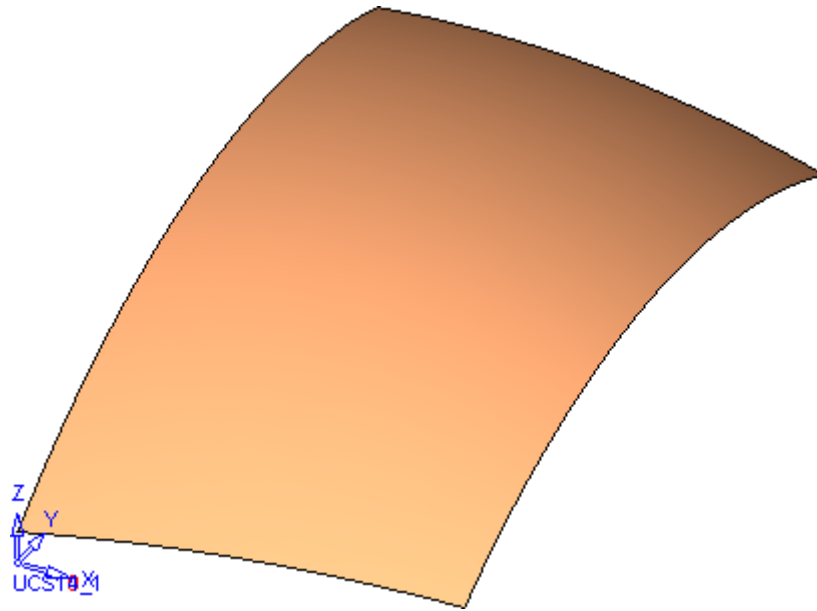
 From the Feature guide, click **OK**  to approve the operation.

 Click on the Zoom All icon  so that the model fits the entire display.

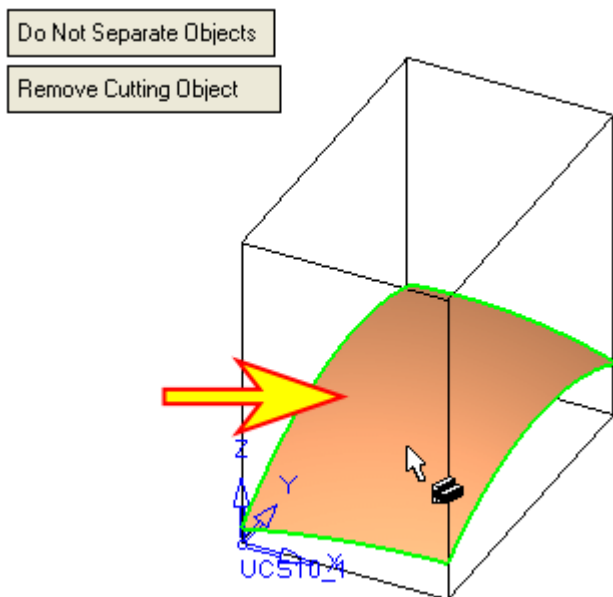
Step 4 - Create a Blend Face

 3DXpert is a hybrid environment, which can work with closed, open or partially open solid objects. Solid, Surfaces and Wire-Frame tools can be applied on the same object.

We will now create a blend face.



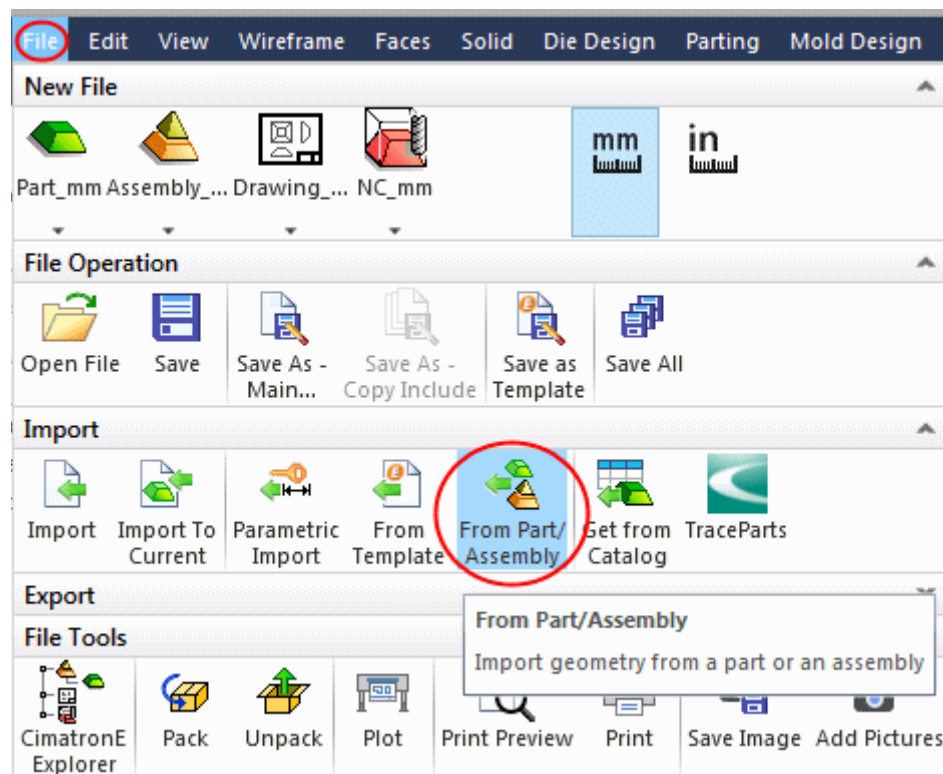
We will then cut the object with the blend face



💡 We will now import predefined contours. Instead of receiving these contours another option would be to create the blend contours via the sketcher.

💡 A part file will be imported from the 3DXpert **Explorer**.

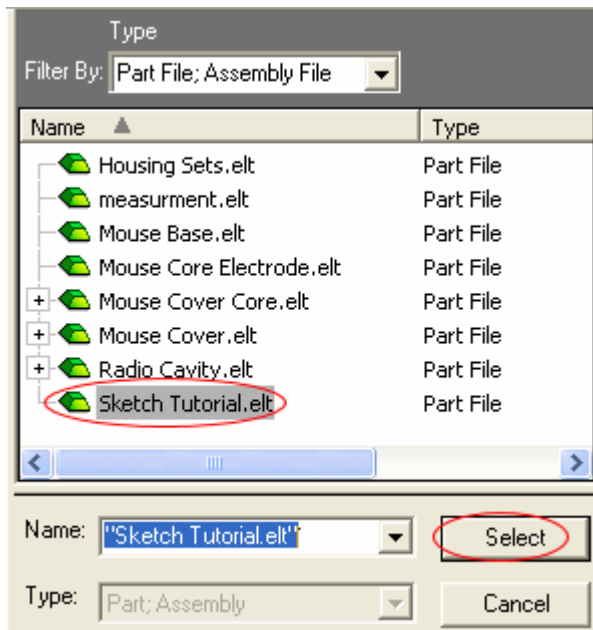
👉 From the File menu, select the **Import/From Part/Assembly** option.




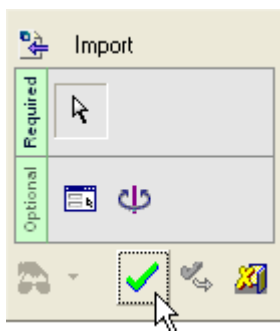
👉 From the 3DXpert **Explorer**, select the **Sketch Tutorial.elt** file (located in the \\Tutorials\3DXpert13.0\WorkFiles\Getting Started directory).

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 Click the **Select** button.

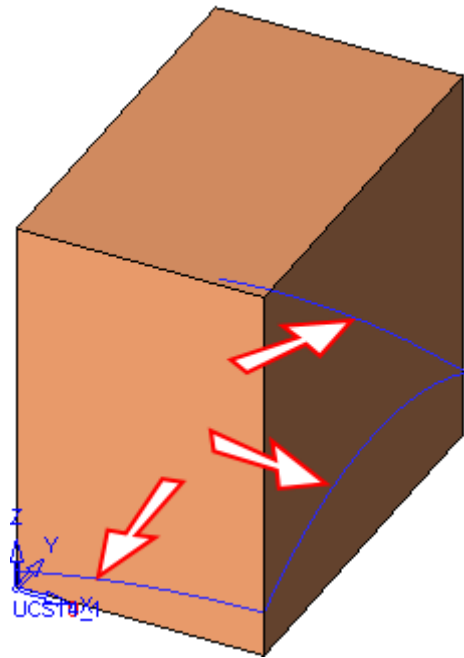


 Click **OK** in the Import Feature Guide.

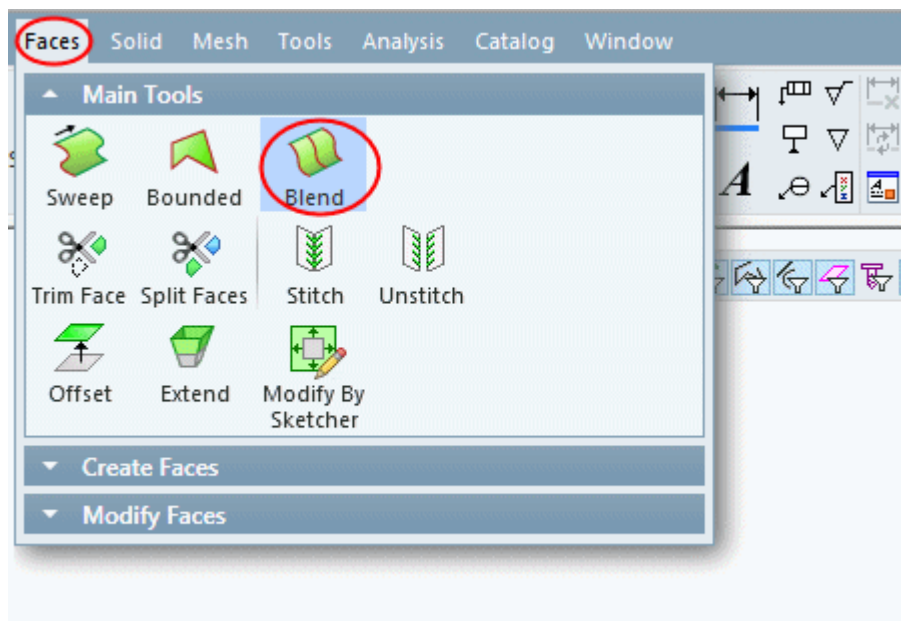


Step 4a - Create a Blend Face

💡 Note the **three** imported curves indicated by the arrows in the picture below:




👉 From the **Faces** menu, select the *Blend* function.



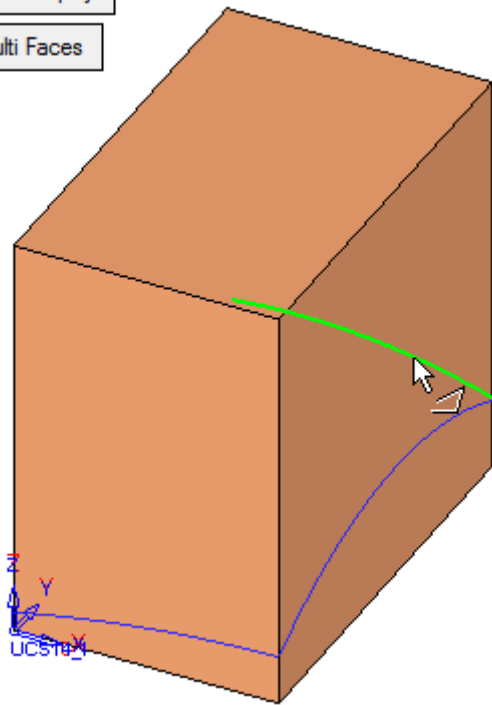
💡 Use the **Dynamic ZPR mode** or press the **CTRL** key together with the **Left Mouse Button** to rotate the part.


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 Pick the first edge of the blend face as indicated by the cursor below:

With Simplify

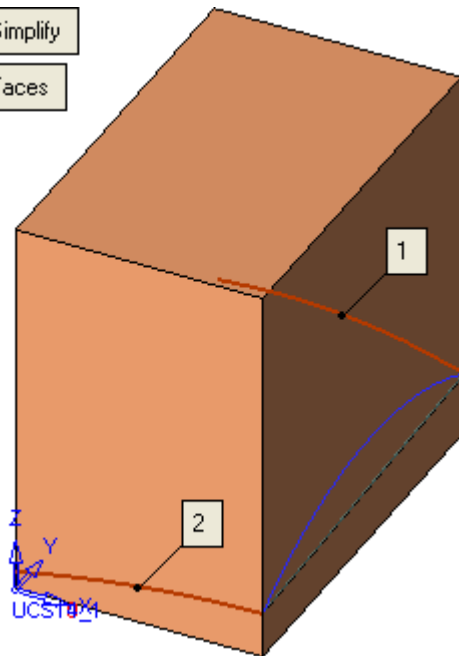
Multi Faces




 Pick the second edge.

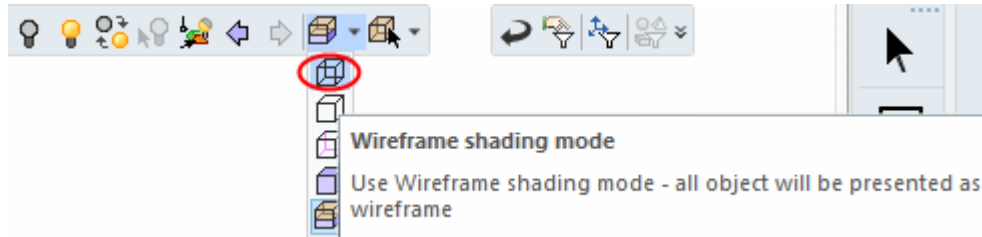
With Simplify


Multi Faces

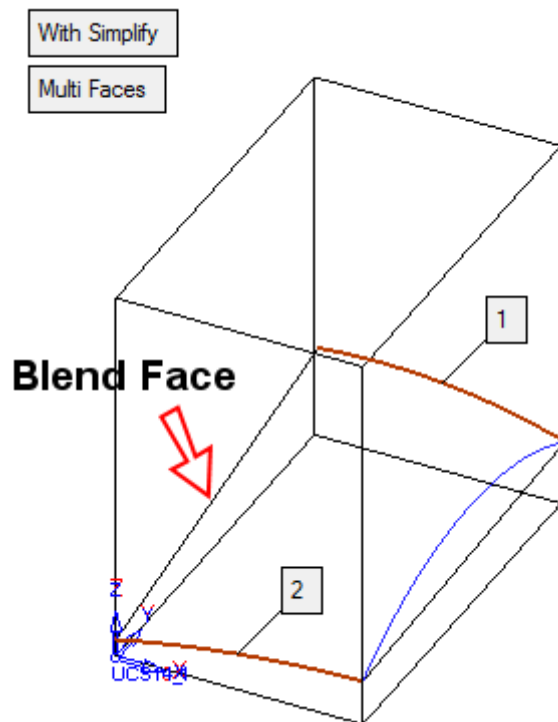



Step 4b - Create a Blend Face

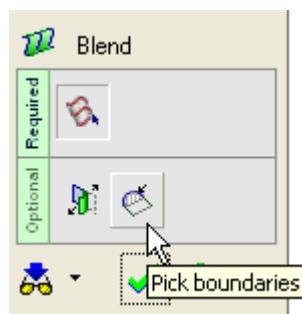
 Change the display to Wireframe mode, by clicking the Global Render mode icon and selecting Global Wireframe.



 Create a blend face.





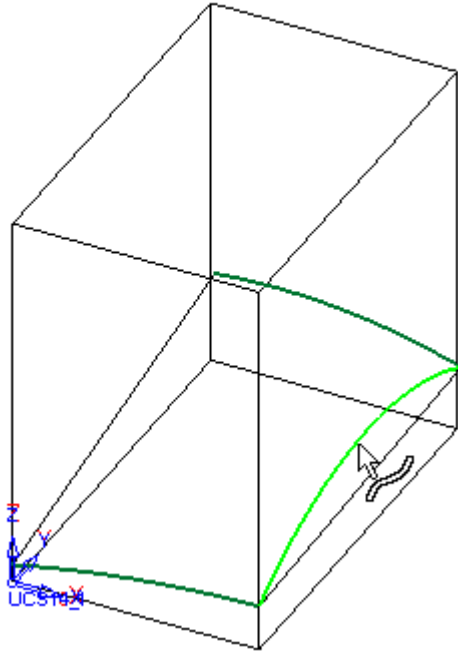
 Define the boundaries of the blend by clicking the **Pick boundaries** option.




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 Pick the contour indicated below:

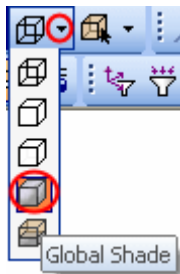
Distance Tol. = 0.1000  



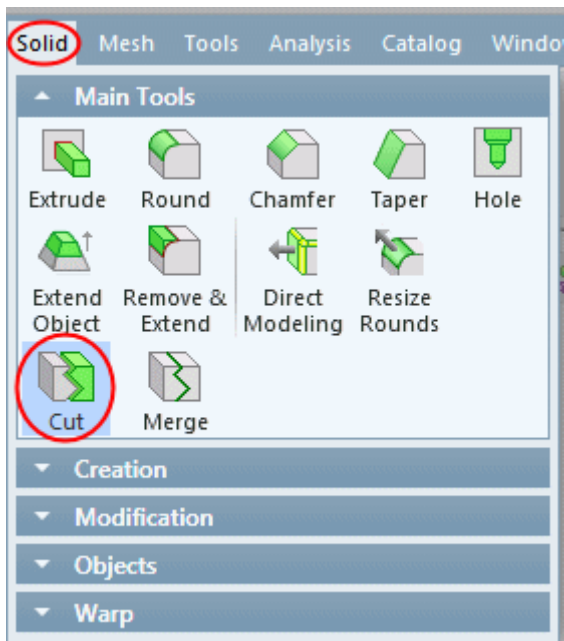
 Click the **OK** button in the Blend Feature Guide.

Step 5 - Solid/Cut


Return to the **Global Shade** mode.

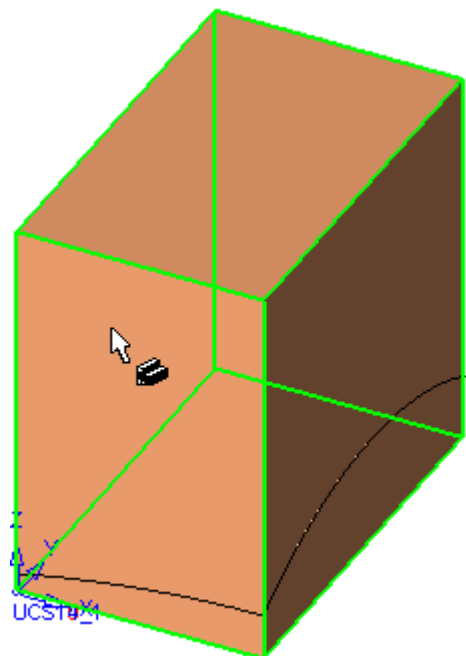


From the **Solid** menu, select the *Cut* function.



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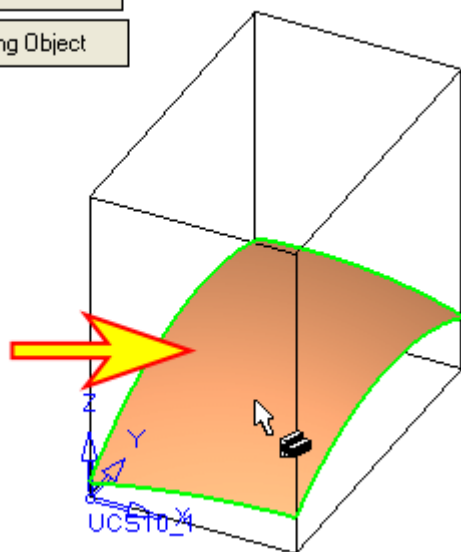
 Pick the solid object you have created as the object to be cut and exit using the Middle Mouse Button.




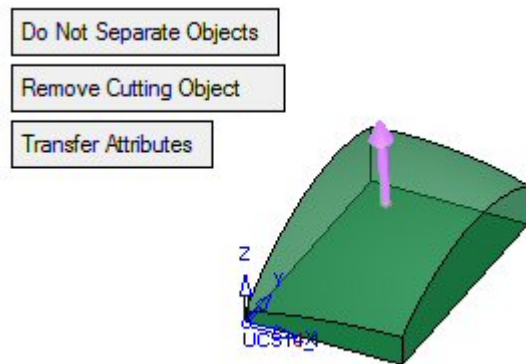
 Pick the blend face as the cutting object.

Accept the default parameter: **Remove Cutting Object**. When using this option, the system automatically removes the cutting object (blend face) when approving the Cut operation.

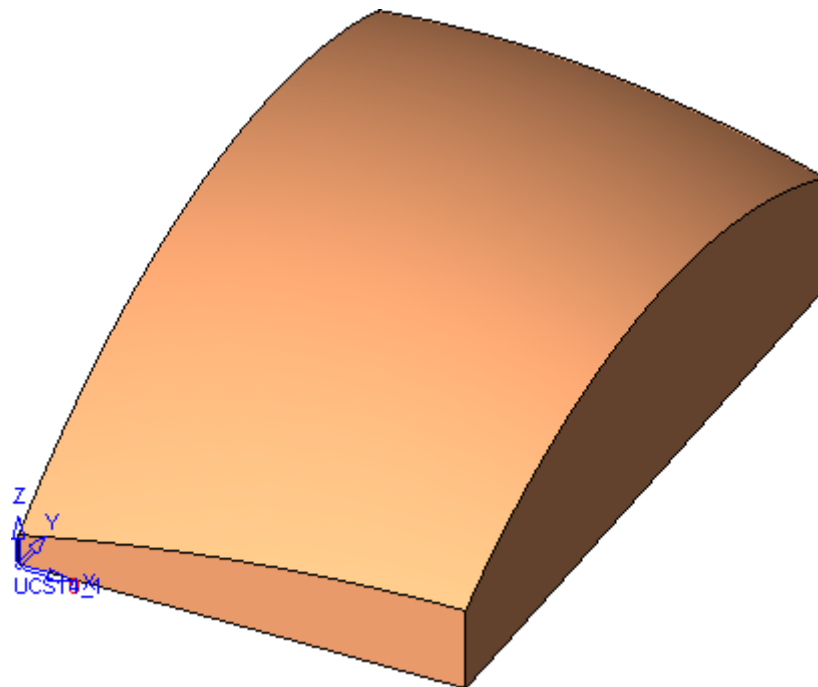
Do Not Separate Objects
Remove Cutting Object




 Accept the cutting direction (blue arrow) as shown in the picture below:



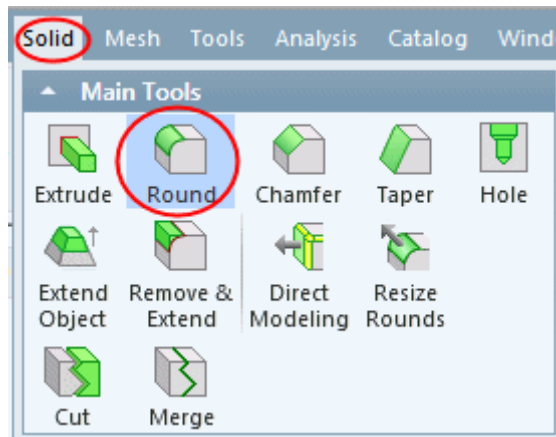
 Click **OK** in the Solid Feature Guide.



Step 6 - Apply Rounds

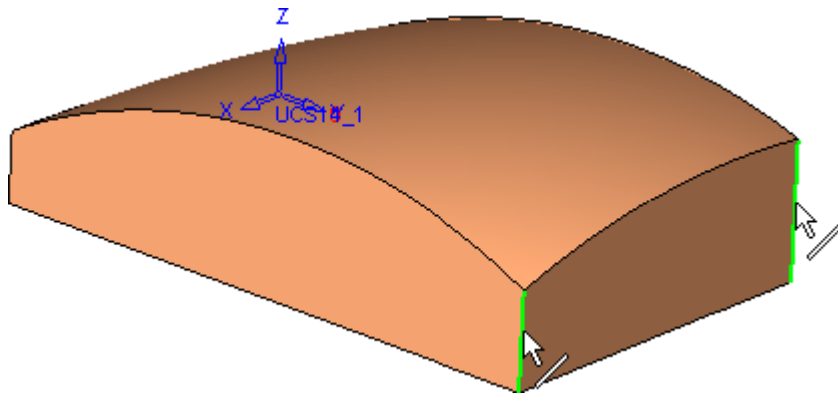
 From the Menu Bar, select **Solid/Round**.


Either from the icons (ribbon menu) 1, or from the Toolbar menu (Solid \ Round) 2.



 Pick the edges to be rounded, indicated in the picture below:

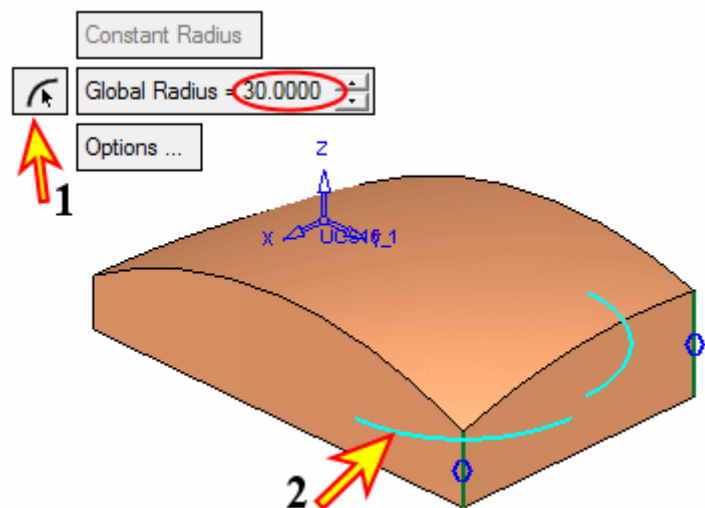
 Use the **Dynamic ZPR mode** or press the **CTRL** key together with the **Left Mouse Button** to rotate the part.



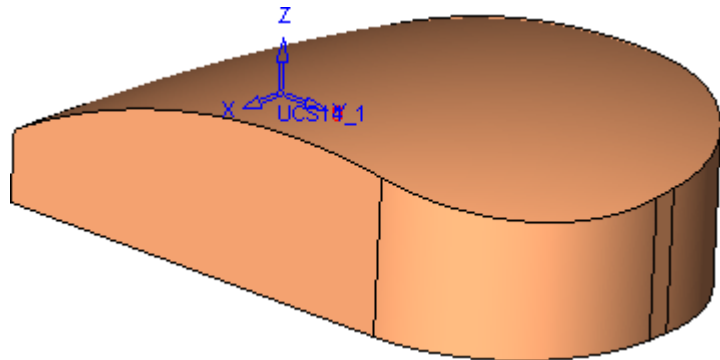
 **Exit** (Middle Mouse Button).

 Apply the following values: **30**


Note that you may also define the radius value by picking an arc, cylindrical face, torus face or a point. (The button is indicated by arrow 1 in the picture below). The selected edges are marked with a preview symbol that reflects the size of the given radius (As indicated by arrow 2)



 Click **OK** to approve the round operation.



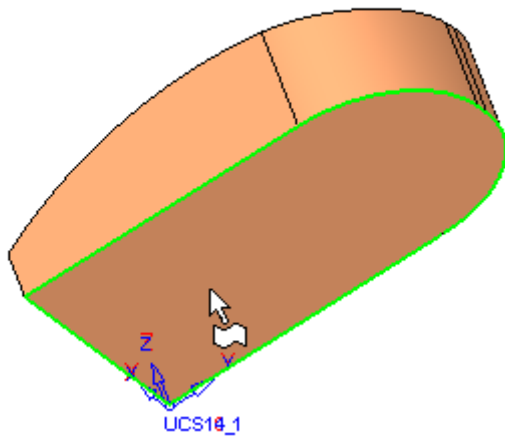
Step 7 - Create a Taper

 From the **Solid** Menu, or from the ribbon icons select **Taper**.


 Accept the default **Neutral Plane** option and pick the following plane:

Neutral Plane

Keep Size at Plane Level

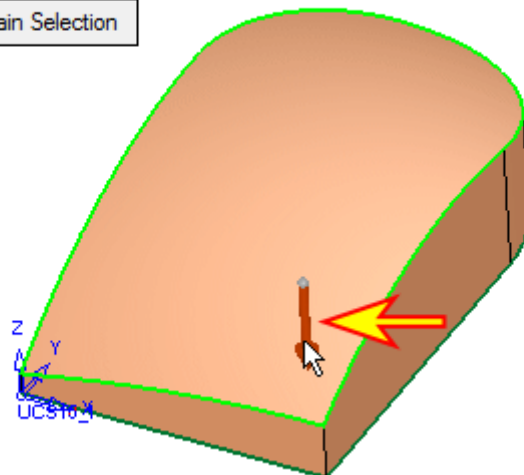



 Exit *via the Middle Mouse Button*.

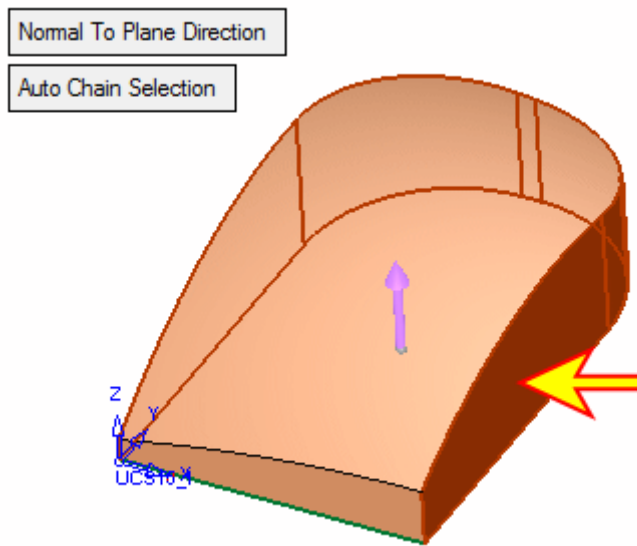
 Click the arrow as indicated in the picture below to flip the arrow direction.


Normal To Plane Direction


Auto Chain Selection



 Pick the face indicated by the cursor, as the face to be tapered:

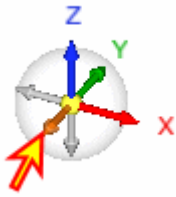



 All the connecting faces (apart from the front face) have been automatically selected.

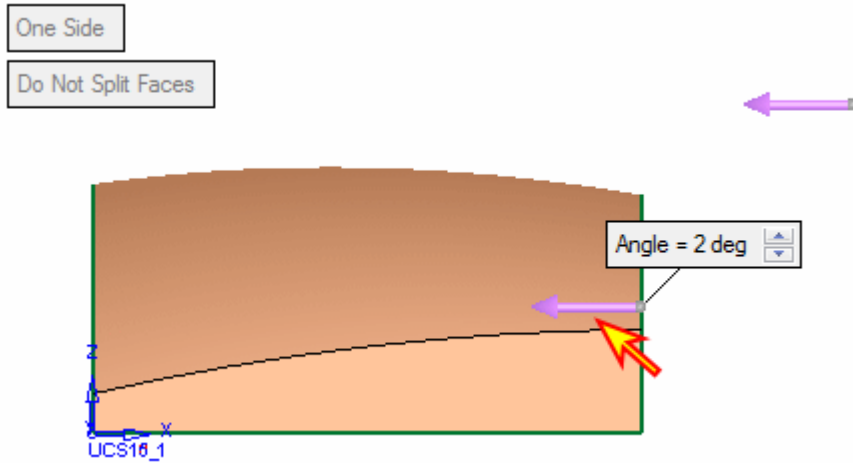
 Middle mouse click to accept these faces, set the **Taper Angle** and the flip the arrow as *necessary*:



 Change to front view  or press the **-Y** axis of the **Dynamic UCS**.

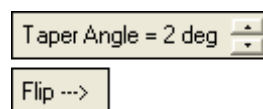
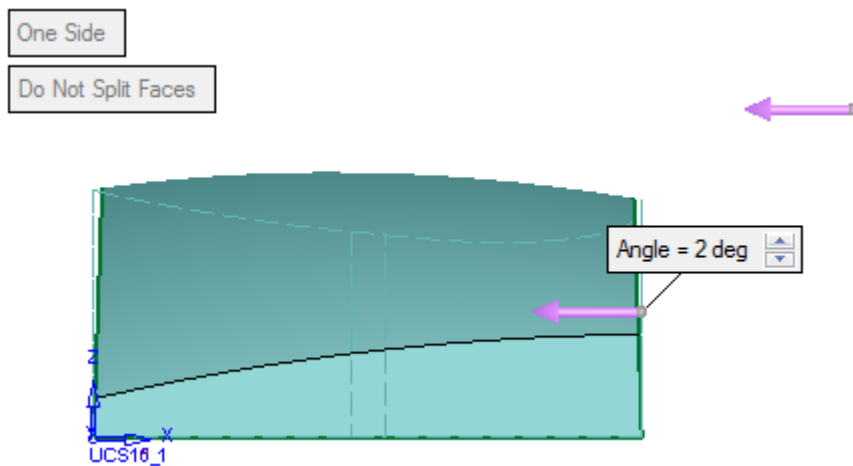
Quicktour




 If necessary, press the arrow to change its the default direction.




 Press the Preview button  in the Feature Guide to view the result before approving it.




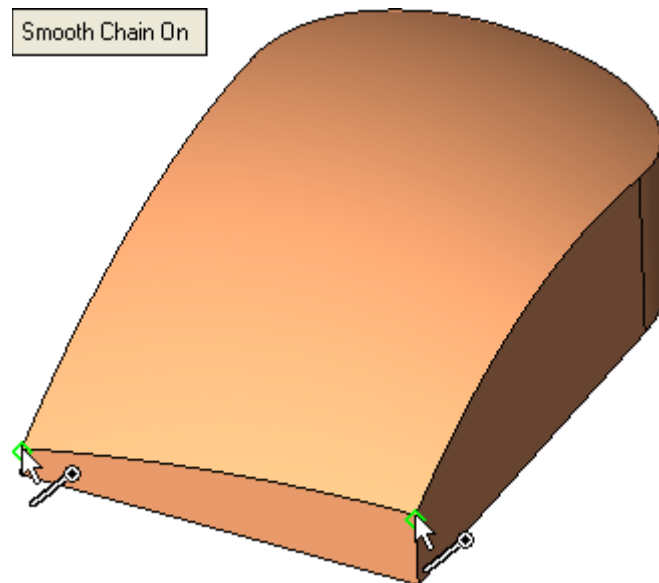
 Click **OK** in the Feature Guide.


Create the final rounds

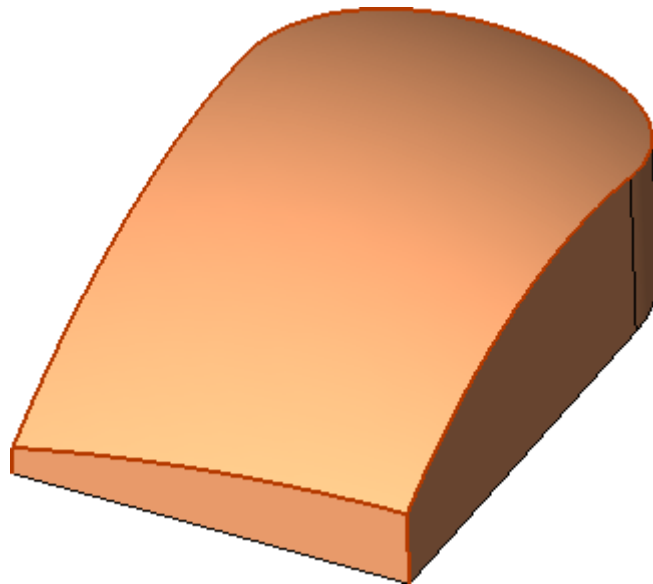
 **Solid/Round** or pick the **Round** icon from the ribbon icons.

 Pick the corners as shown in the picture below:


 Hide the UCS and zoom in for easier selection of the corner points.

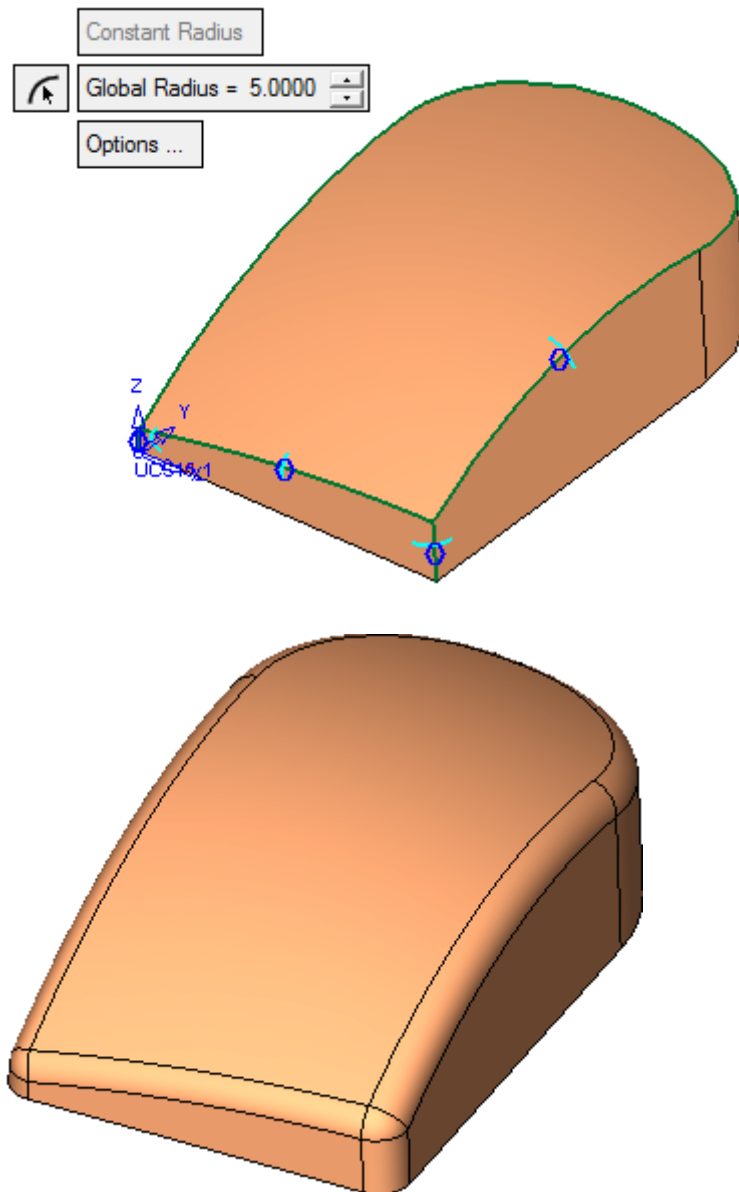


 Note that all the edges connected by this corner, are automatically picked.



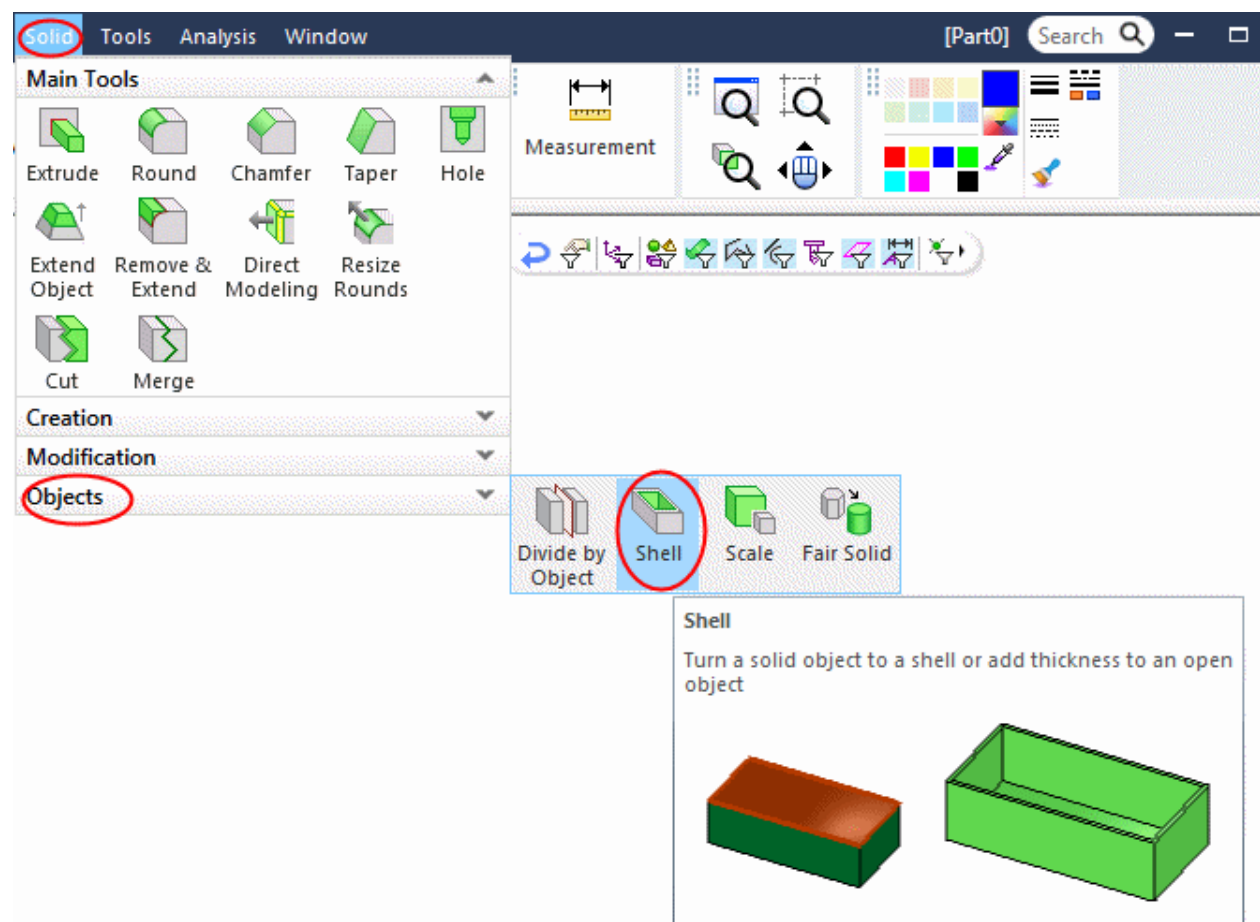
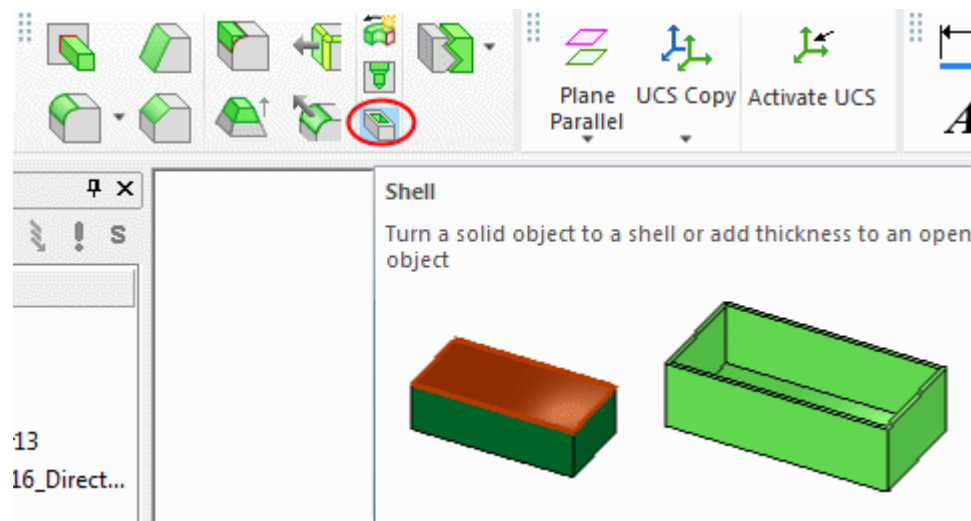
 Exit *via the Middle Mouse Button*.

 Apply rounds with a radius of 5.




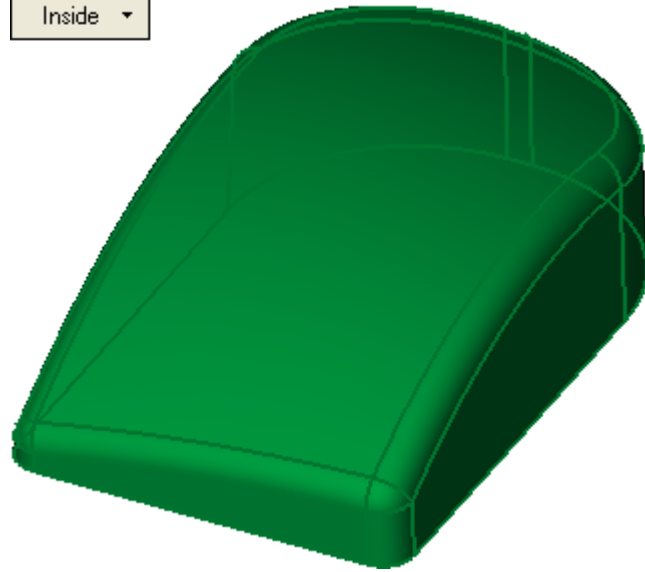
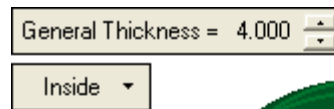
Step 8 - Create a Shell


From the **Solid** menu, select **Objects \ Shell** or select the *icon* from the menu.





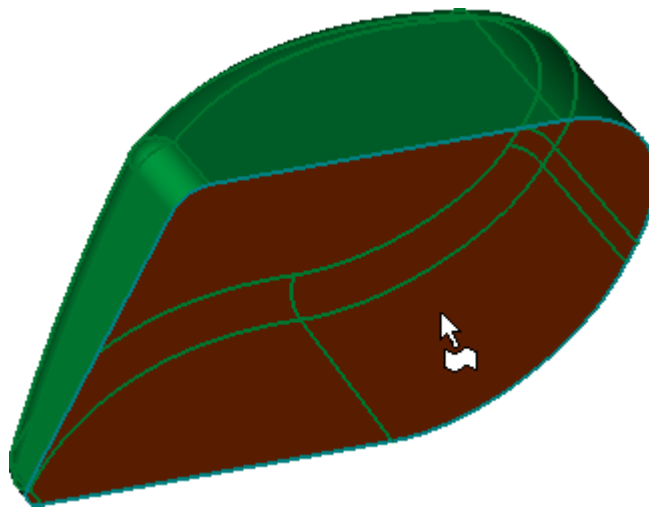
Quicktour

 Change the thickness to 4.

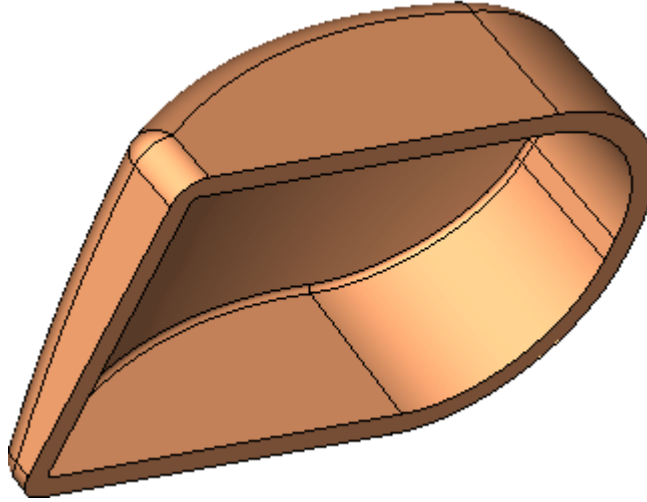


 Pick the bottom face as the open face.


 Use the **Dynamic ZPR mode** or **press the** Rotate icon  or press the **CTRL** key together with the **Left Mouse Button** to rotate the part.



 Click **OK** in the Shell Feature Guide.

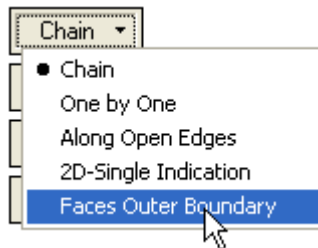


Step 9 - Create a Step

 Create a composite curve, by clicking the Composite curve icon or select it from the **Curves** menu.



 Change the option to **Faces Outer Boundary**.



Quicktour



See the text below about how to create a composite curve:



Pick the face shown:

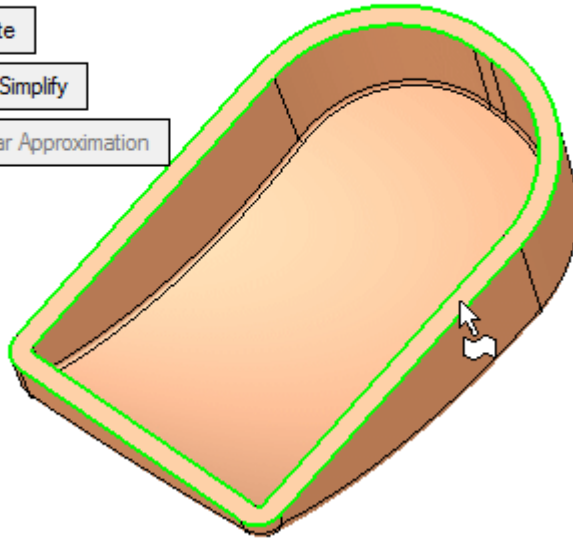
Faces Outer Boundary ▾

Max. Gap = 0.0100

Composite

Without Simplify

No Planar Approximation



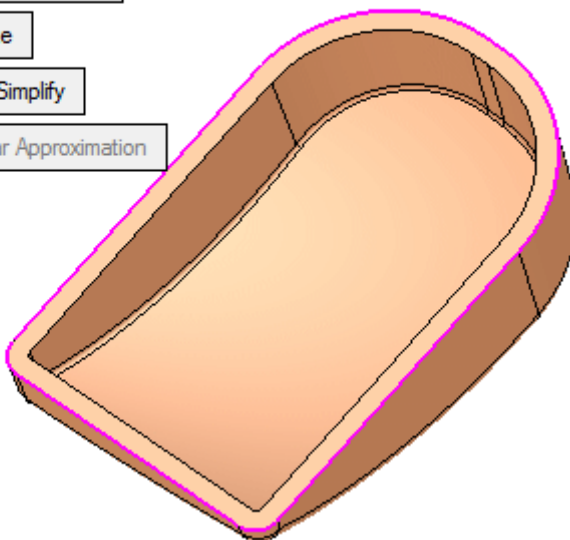
Faces Outer Boundary ▾

Max. Gap = 0.0100

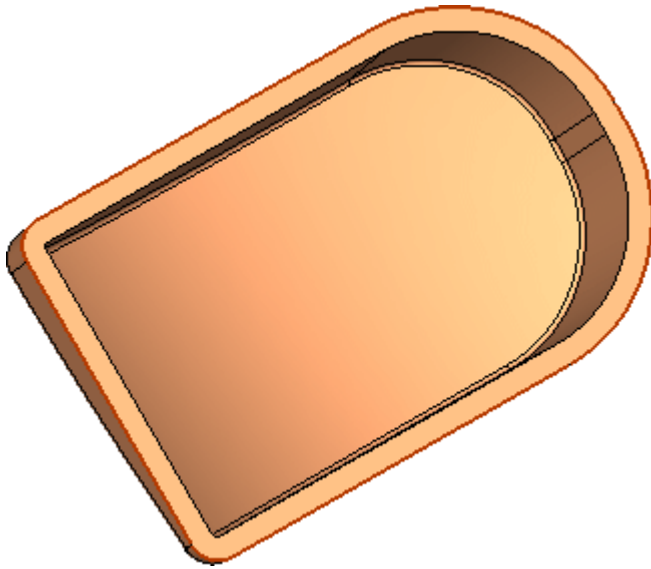
Composite

Without Simplify

No Planar Approximation

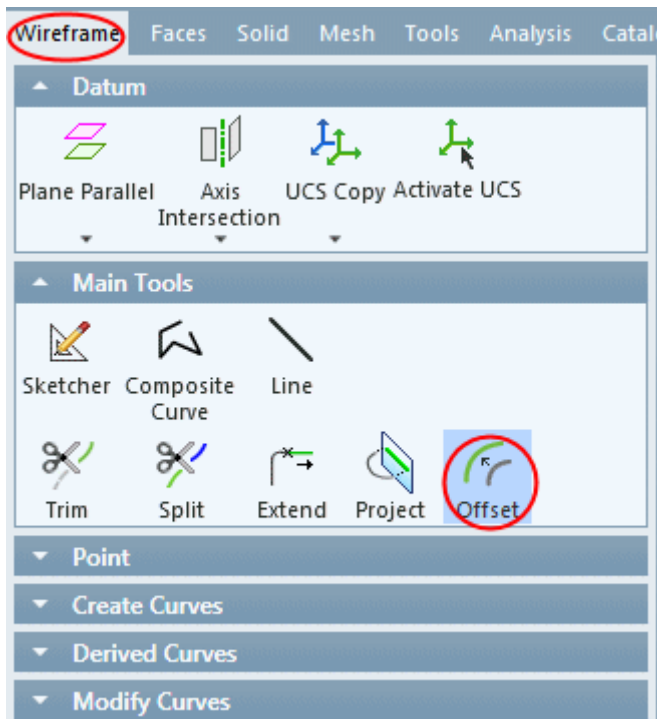


 Accept the result by clicking **OK** in the Composite Feature Guide.




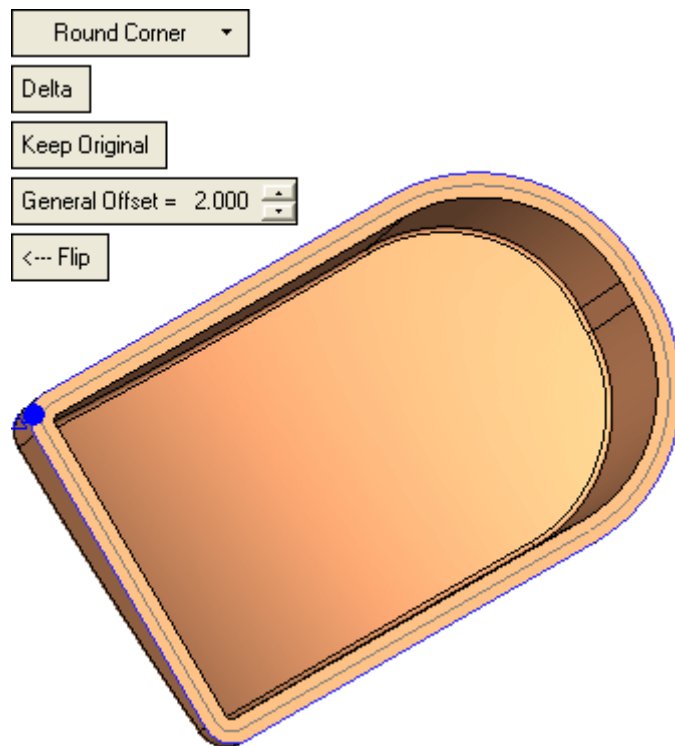
Create an offset curve

 From the **Wireframe** menu, select **Offset**.




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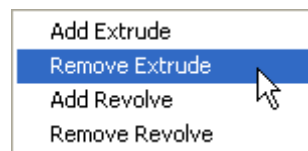
 The previously created composite curve is already selected, and if not select it and apply the following parameters:




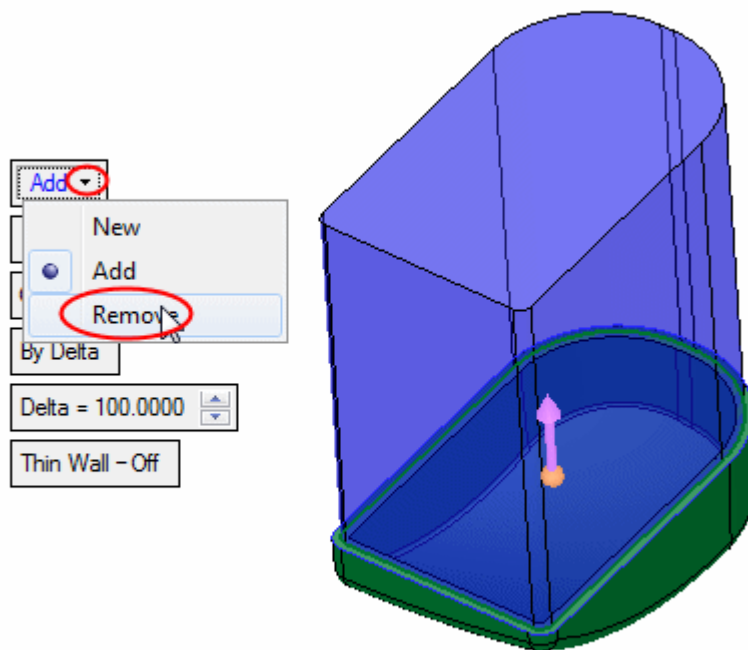
Click OK in the Offset Feature Guide.

Extrude remove the offset curve

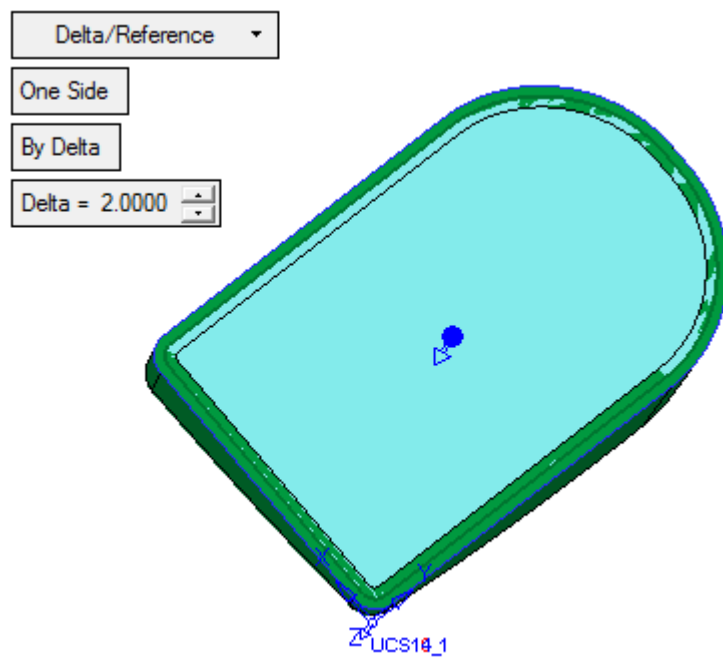
 Ensure that the Offset curve is selected,, right click and select *Extrude*.



 Select the option Remove. :

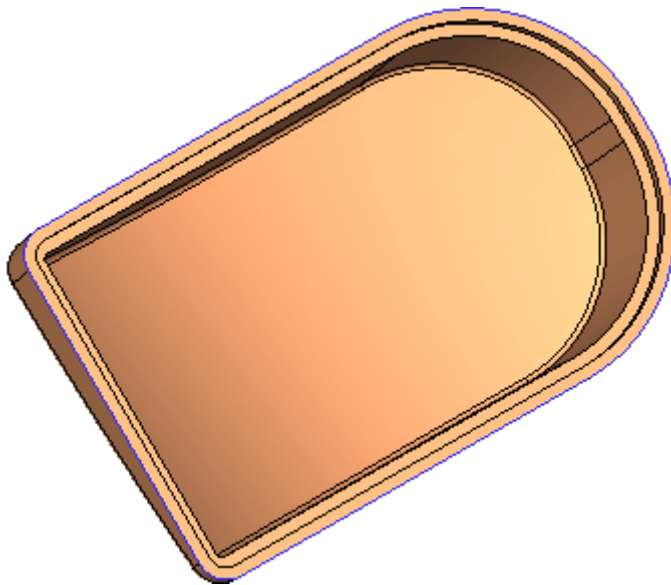



Set the following parameters:

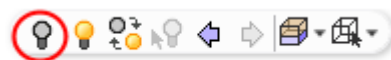


Quicktour

 Click **OK** in the Extrude Feature Guide.




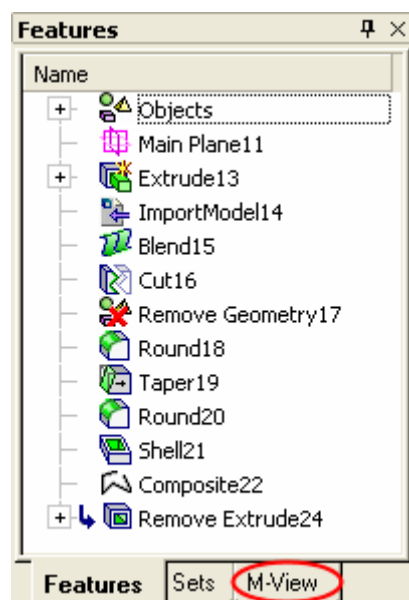
 Pick the previously created curve and hide it.



Step 10 - Create an M-View


A Modeling View (M-View) will now be created. The M-View is a user-defined view of entities which includes a profile of display parameters.


 At the bottom left of the screen select the M-View tab.

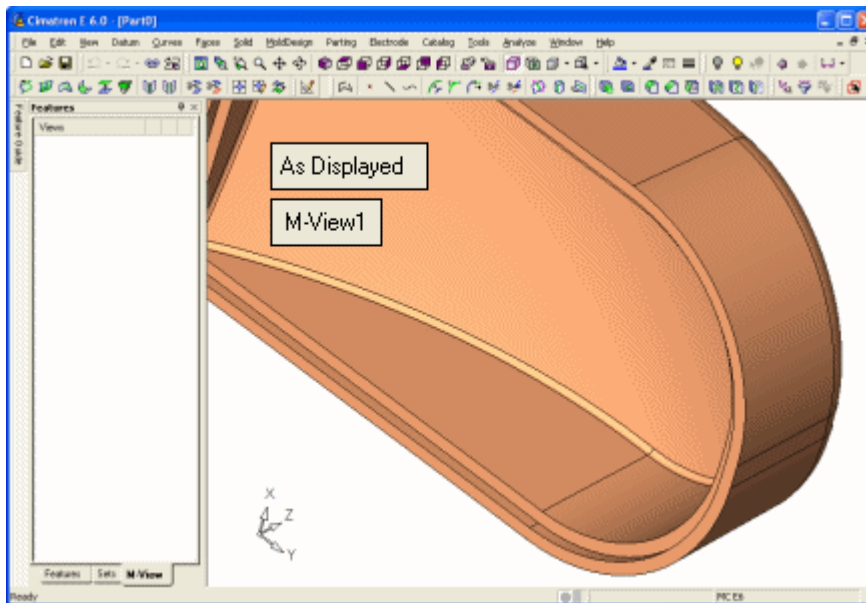


 Under the M-View tab, Right Click and select **New M-View**.



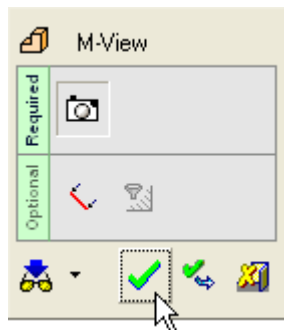
 Position the Part on the Screen as follows:

 Zoom, Pan and Rotate the part accordingly

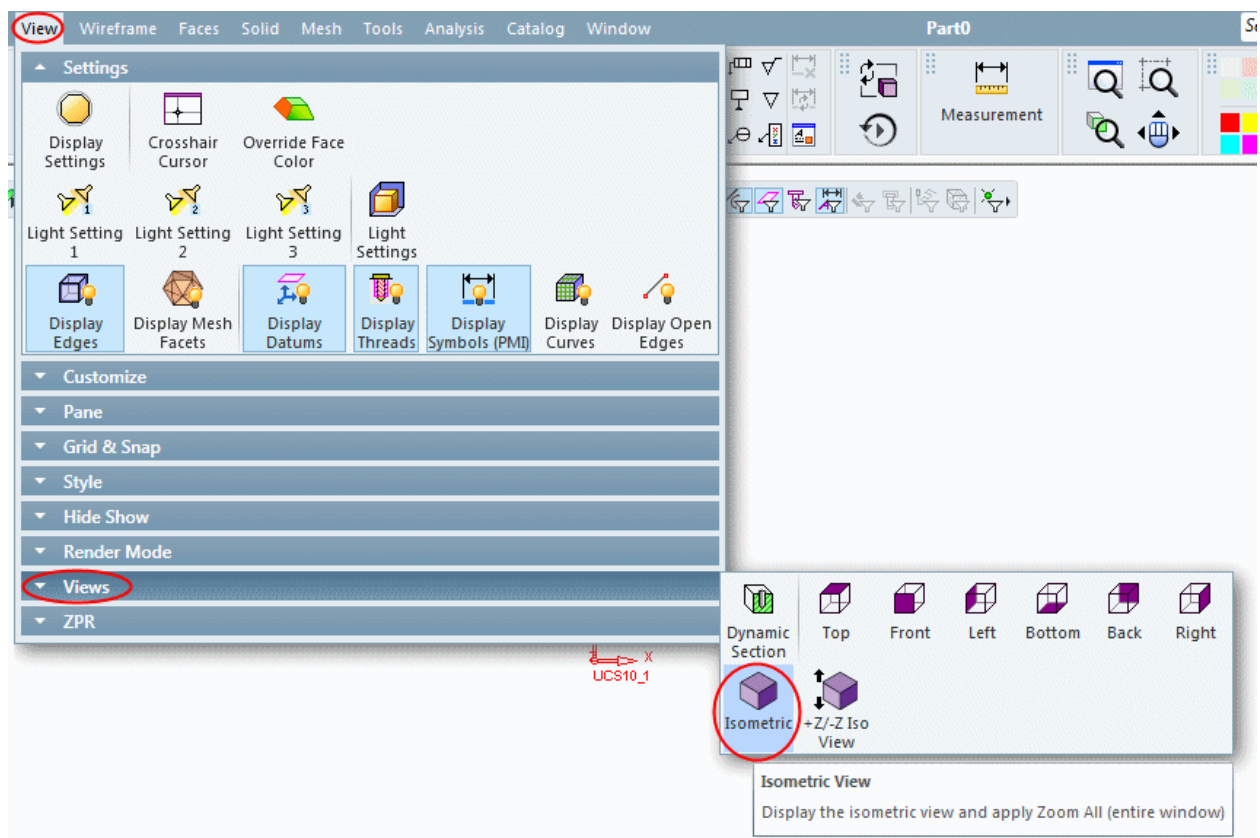


Quicktour

Click **OK** in the M-View Feature Guide.




Click on the *Isometric view icon*.

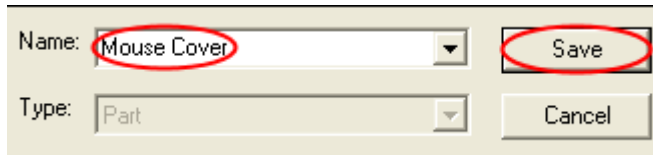
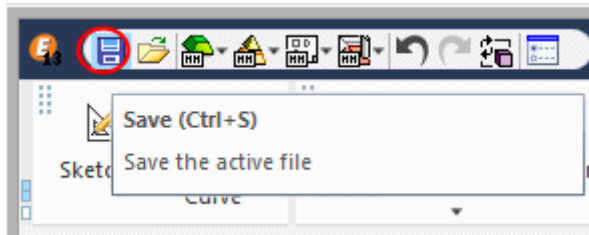


Under the M-View tab, Double Click on **M-View1**.

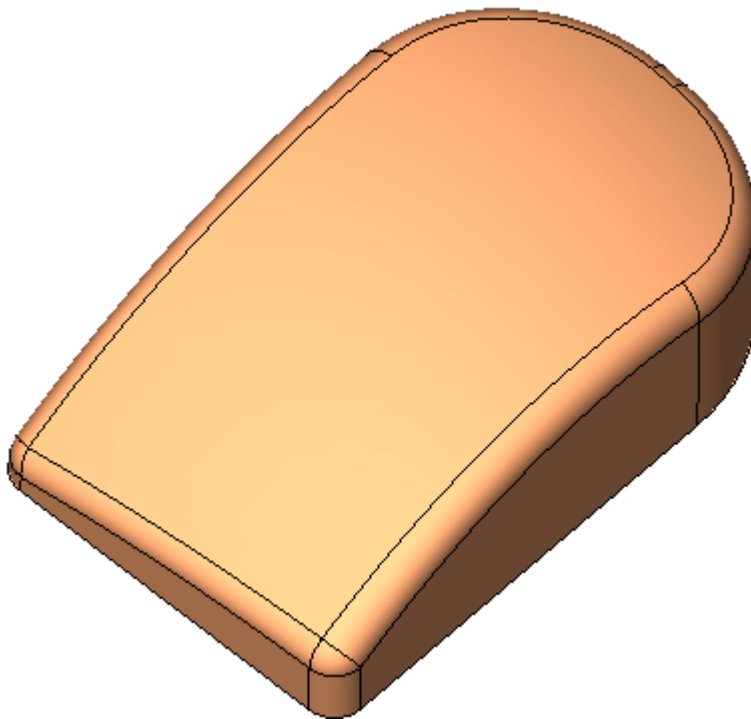



The Part has returned to it's original orientation and size.

 Select **File/Save** or select the *Save icon* from the Quick Access Commands and in the 3DXpert **Explorer**, save the file under the following name:



A new file named **Mouse Cover** is created.



 Close the part file.