Cast part modeling

In this web training task we will learn how to create solids, insert draft angles starting from different draft solids and joining them all together using Boolean commands. At the end we will create a new model using Model derived from current and see how to work on the Derived model.This casting part is used in an industrial truck.

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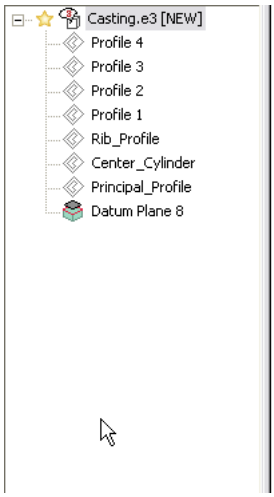
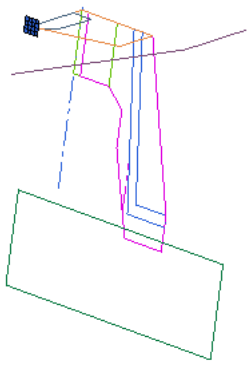
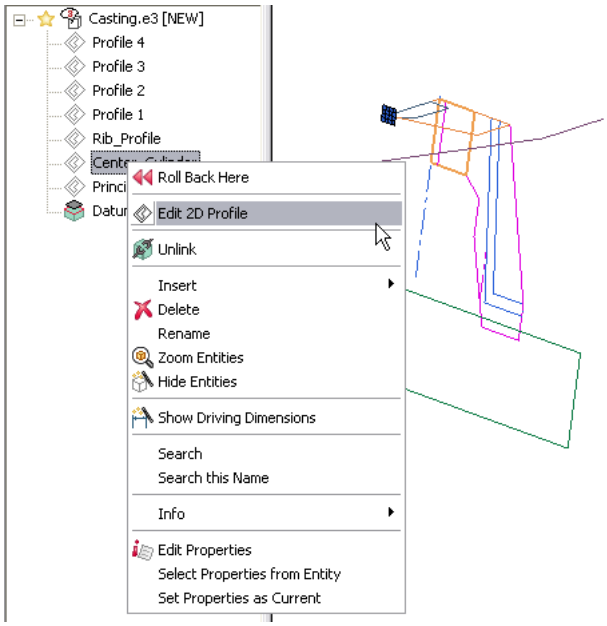
1. Step 1 : Profiles and basic features.

In this web training task, most of the profile creation will be skipped in order to show some advanced features and design tips. We have created several profiles before creating any solids and will activate the profiles as we need them. This is a useful method when we don't need solids for reference because it keeps the work area cleaner.





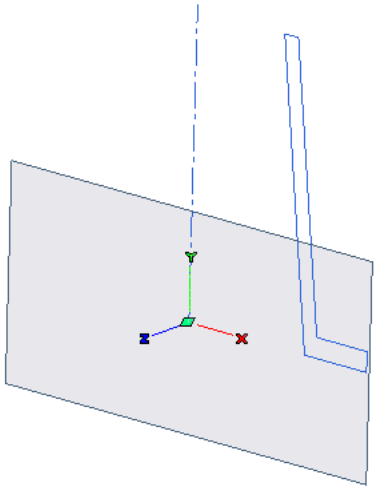
Right click on each profile and select **Edit Profile**. Take the time to study each profile and how it was created before you start modeling.



Note:

We are creating constraints with the Principal_Profile, reliable relations do not depend on topology changes. Half dimensions are used to enhance the drawing layout.

Once you are done studying the profiles, using the **Hide Entities** command, hide everything except for the Principal_Profile. We will unhide the remaining profiles as we need them.

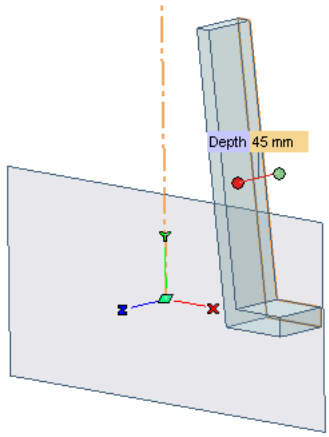


Let's start building the solids. Building separate solids will give us better control of the model when adding draft and fillets. Before creating each solid, change to a different **Color** to add contrast.

Create a **Linear Solid** using the Principal_Profile.

- Start the **Linear Solid** command.
- Select the Principal_Profile.
- Set the depth as Depth45

- Click OK.



Using the **Draft Angle** command, we will add the material, not remove it, so it is better to create the draft later.

Note:

There are 3 methods to create the draft angles, that are listed below:

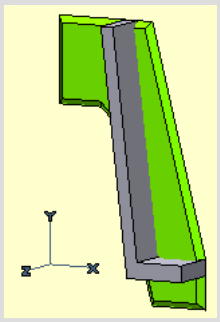
1. Drawn directly into the profile.
2. By angled extrusions.
3. Using the **Draft Angle** command.

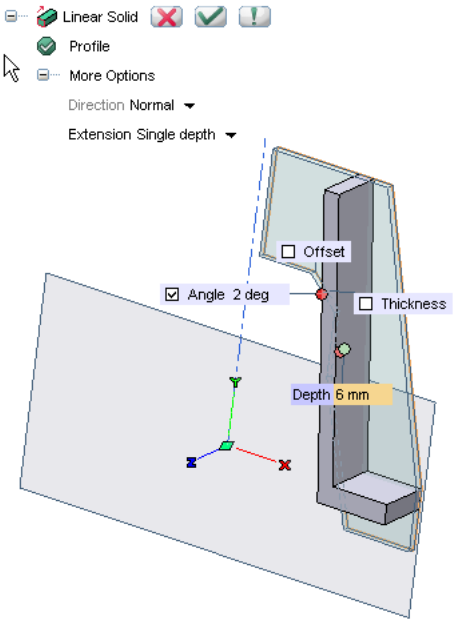
Now let's make another **Linear Solid** using the Rib_Profile.

- Using the **Unhide Entities** command, unhide the Rib_Profile.
- Start the **Linear Solid** command.
- Select the Rib_Profile.
- Set the depth as Depth6
- Expand More Options in the selection list to expose the Angle mini dialog box.
- Set the Draft Angle as Angle2, removing the material.
- Click OK.

NOTE:

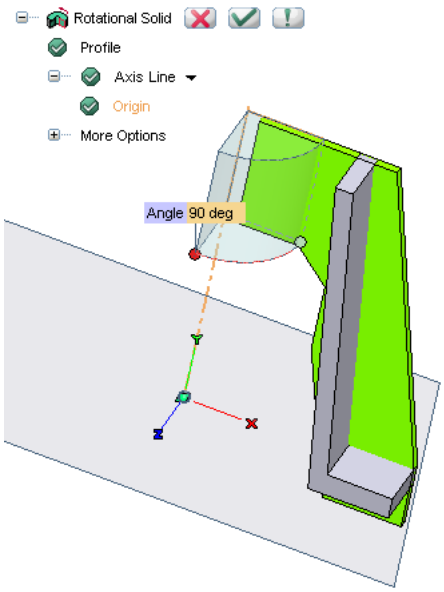
Make sure that the draft angle direction is set to remove material from the solid as shown in the exaggerated example image below.





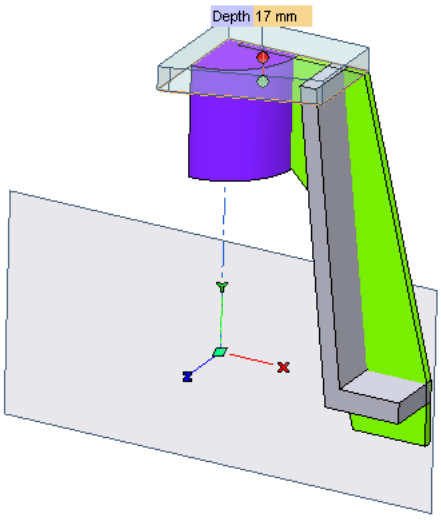
Create a **Rotational Solid** using the Center_Cylinder profile. We will add draft angle to this later.

- Using the **Unhide Entities** command, unhide the Center_Cylinder profile.
- Start the **Rotational Solid** command.
- Select the Center_Cylinder profile.
- Set the Axis to Line.
- Pick the left vertical line as the axis.
- Set the Angle to 90 Angle90
- Set the rotation in the direction shown in the image.
- Click OK.



Create a **Linear Solid** using Profile 1.

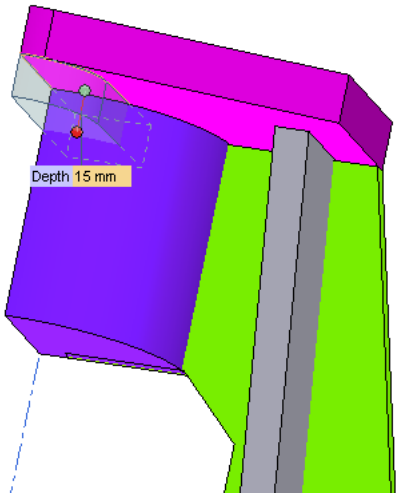
- Using the **Unhide Entities** command, unhide Profile 1.
- Start the **Linear Solid** command.
- Select Profile 1.
- Set the Depth as Depth17
- Uncheck the Angle, we do not want draft on this solid.
- Click OK.



Create the last **Linear Solid** using Profile 3.

- Using the **Unhide Entities** command, unhide Profile 3.

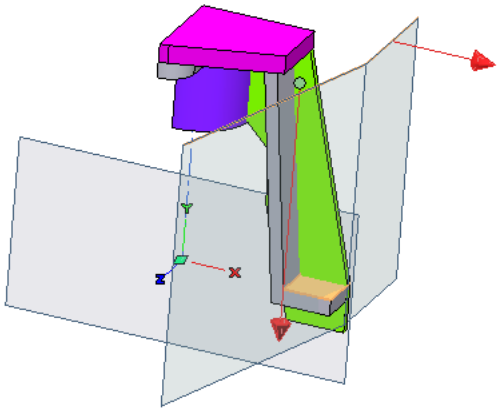
- Start the **Linear Solid** command.
- Select Profile 3.
- Set the Depth as Depth15
- If the solid preview is facing the wrong way, right click on the Depth15 mini dialog box and select Invert.
- Click OK.



2. Step 2 : More Basic features.

Make a **Linear Slot** through first solid using Profile 2.

- Using the **Unhide Entities** command, unhide Profile 2.
- Start the **Linear Slot** command.
- Select Profile 2.
- Set the Extension to Thru all.
- If the cutting arrow is pointing in the wrong direction, double click on it to invert it. It must be pointing as shown in the image below.
- Select the highlighted face to cut.
- Click OK.



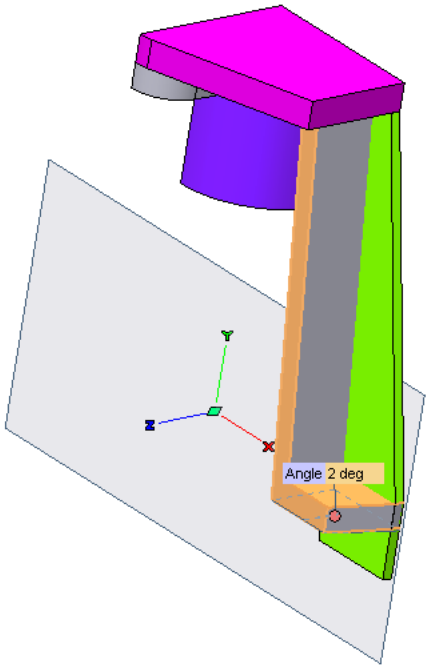
To clean up the view, using the **Hide Entities** command, hide all the profiles and any driving dimensions that are shown.

Let’s start creating draft angles. We are going to add material not remove it.

Note :

The same results can be reached by building the main profile over a datum plane, then creating a linear extrusion using the draft angle option.

- Start the **Draft Angle** command.
- Select the front surface as the Reference Plane.
- Select the 4 other side faces except the side face as shown in the image as the Face to be drafted.
- Set the draft Angle as Angle2 for adding material.
- Click OK.

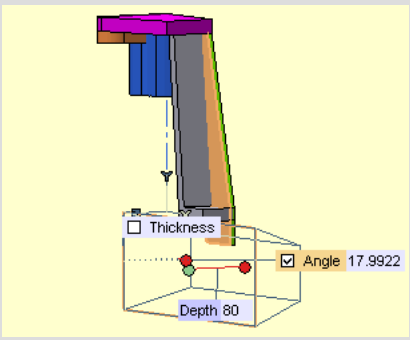


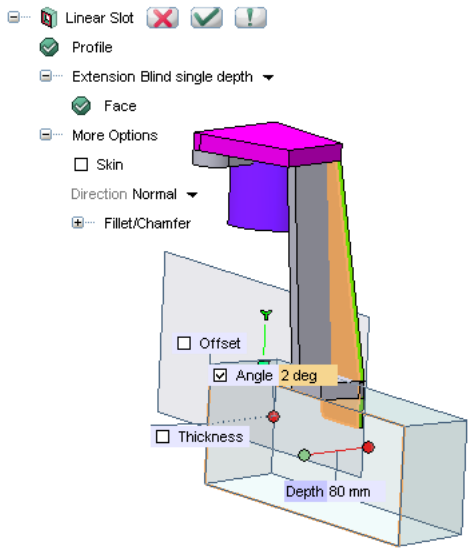
Now let's cut the bottom part of the Rib using Profile 4.

- Using the **Unhide Entities** command, unhide Profile 4.
- Start the **Linear Slot** command.
- Select Profile 4.
- Set the Extension to Blind.
- Select the highlighted face to cut.
- Set the Depth as Depth80
- Expand More Options in the selection list to expose the Angle mini dialog box.
- Set the draft Angle as Angle2
- Click OK.

NOTE:

Make sure that the draft angle direction is set as shown in the exaggerated example image below. To use the Draft Angle option, we must set the Extension to Blind.

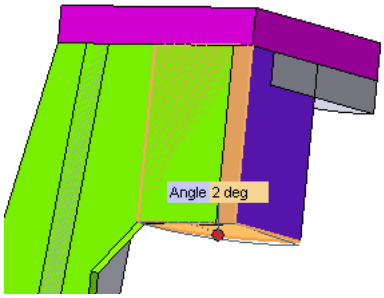




Using the **Hide Entities** command, hide the profiles.

Now we are going to add draft to the Cylinder solid.

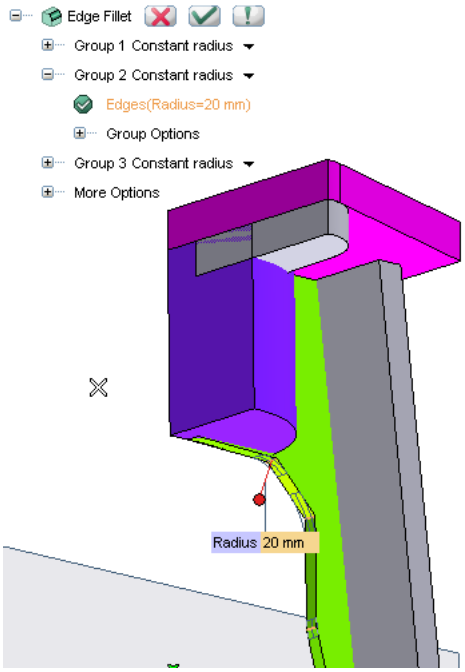
- Start the **Draft Angle** command.
- Select the rear surface as the Reference Plane.
- Select the bottom face as shown in the image below as the Face to be drafted.
- Set the draft Angle as Angle2 to remove material.
- Click OK.



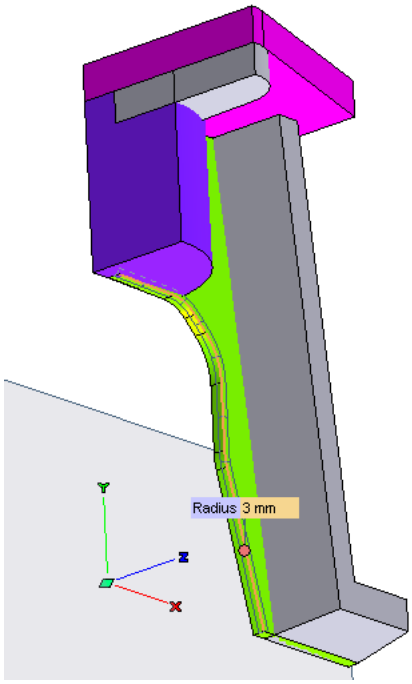
3. Step 3 : Filleting the Model edges.

Let's start adding fillets to the model using the **Fillet Edges** command.

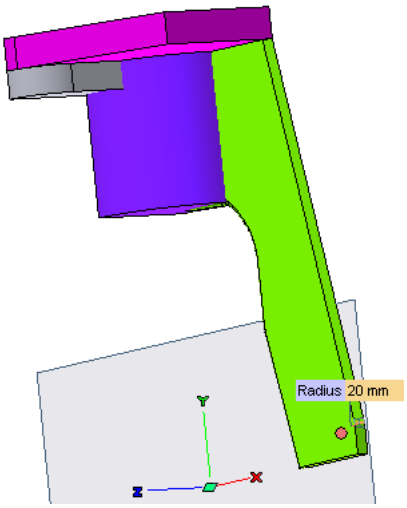
- Using the **Hide Entities** command, hide the Principal solid for better visibility.
- Start the **Fillet Edges** command.
- Add the Radius30 fillet to the 2 edges in Group 1 as shown.
- Expand Group 2.
- Add the Radius20 fillet to the edge as shown.
- Click Apply.



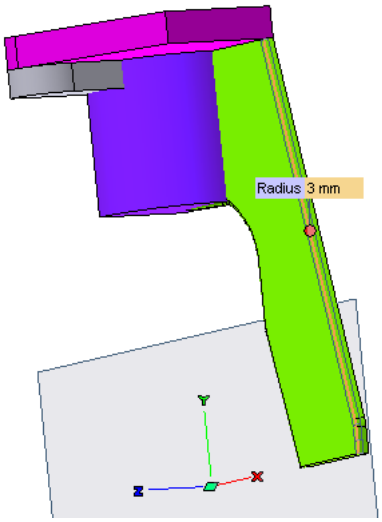
- Add the Radius3 fillet to the edge shown
- Click Apply.



- Add the Radius20 fillet to the edge shown
- Click Apply.



- Add the Radius3 fillet to the edge shown
- Click OK.

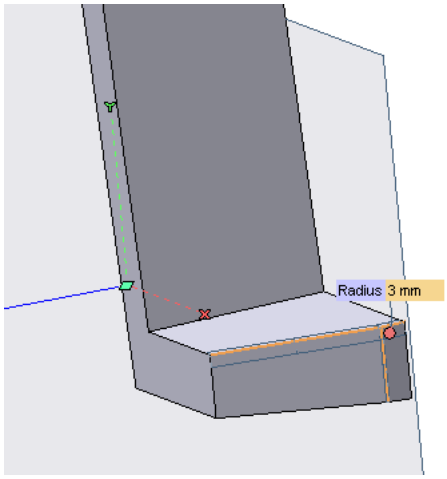


NOTE:

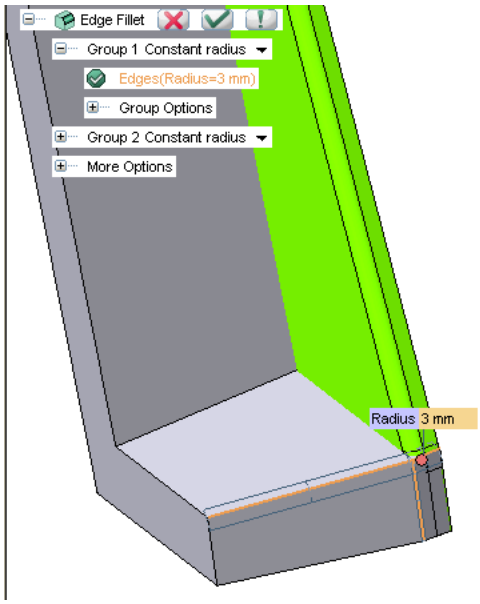
We have not joined the solids yet; all the solids are still separate.

Using the **Hide Entities** command, hide the fillet dimensions to clean up the view.

Using the **Unhide Entities** command, unhide the Principal solid and add fillets to it.



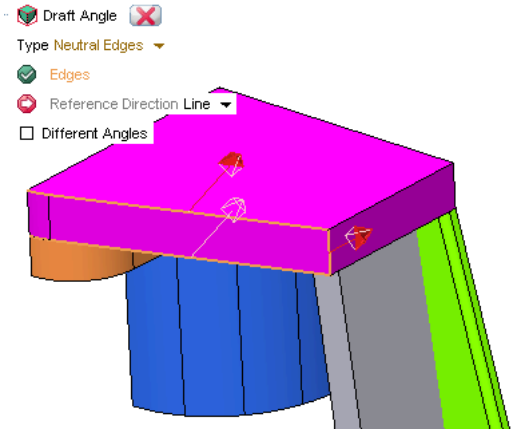
- Using the **Hide Entities** command, hide the Rib solid.
- Add the two Radius3 fillets as shown, in one operation.
- Click OK.



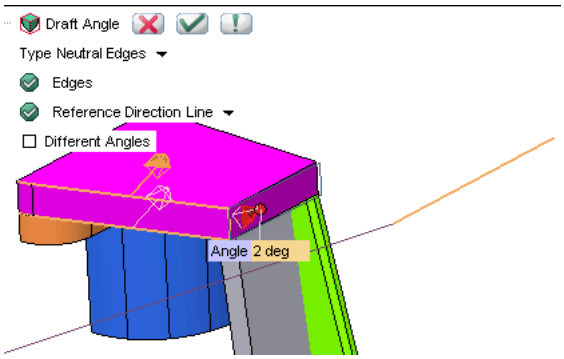
Create a **Draft Angle** on the top solid using the Neutral Edges option.

Here the goal is to add material to the top,bottom and right side surfaces. Same result can be achieved by removing material, by selecting the back side of the solid as the Reference Plane.

- Unhide the Profile2 using **Unhide Entities** command.
- Start the **Draft Angle** command.
- Set the Type to Neutral Edges.
- Select the 3 edges of the solid as shown in the image below.
- Set the arrow directions as shown in the image.



- Select the vertical line of the Profile2 as a reference direction line.
- Set the Draft Angle as Angle2, adding material.
- Click OK.



Using the **Unhide Entities** command, unhide the Rib solid.

And hide the Profile2 using **Hide Entities**.

4. Step 4 : Drafting and Union of solid features.

Now use the datum plane as the Reference Plane to add draft to the lower support. We are going to add material to it. Hide the datum plane using **Hide Entities**.

- Start the **Draft Angle** command.
- Change the Type to Neutral Plane.
- Unhide the datum plane and Select the same as Reference Plane.
- Select the bottom face as shown in the image below as the Face to be drafted.
- Set the Draft Angle as Angle2, adding material.
- Click OK.

Using the **Hide Entities** command, hide the datum plane.

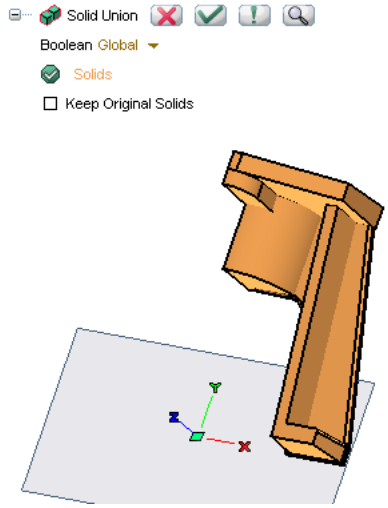
Join all the solids using the **Solid Union** command.

Note :

Up till now, we have had the freedom and flexibility for creating fillets and draft angles, by treating each as individual solids. On the downside, the file size becomes larger. So let's make it as a single entity using the **Solid**

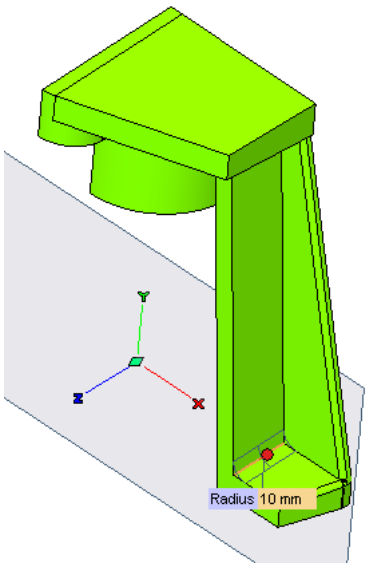
Union command.

- Start the **Solid Union** command.
- Pick all of the solids.
- Click OK.



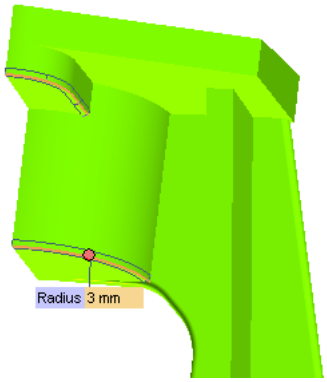
Now let's start creating **Fillet Edges** on the joined solid.

- Start the **Fillet Edges** command.
- Add a Radius10 fillet to the edge shown.
- Click OK.



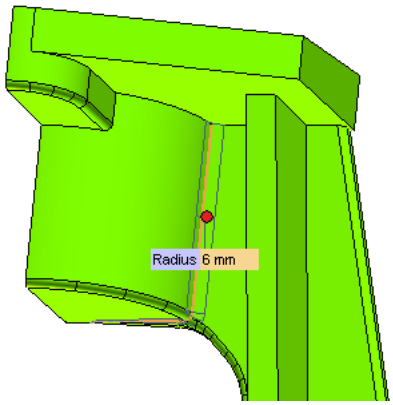
- Again start the **Fillet Edges** command.
- Add a Radius3 fillet to the edges shown.

- Click OK.

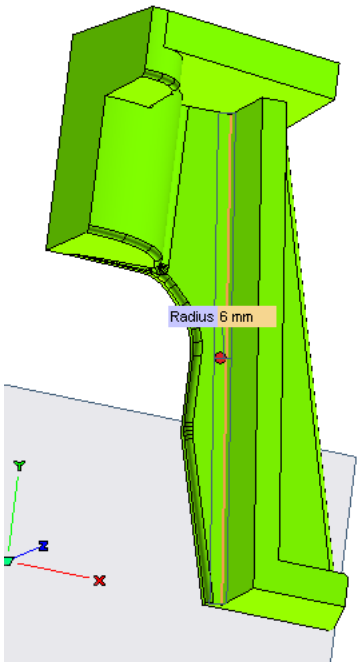


5. Step 5 : More fillets to the Casting part.

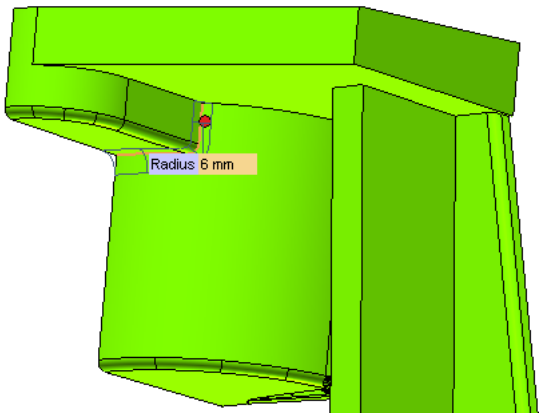
- Start the **Fillet Edges** command.
- Add a Radius6 fillet over both previous fillets



- Rotate the model and add the Radius6 fillet as shown.
- Click Apply.

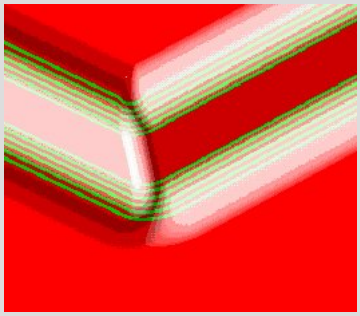


- Add the Radius6 fillet, rolling over the previous one.
- Click OK.



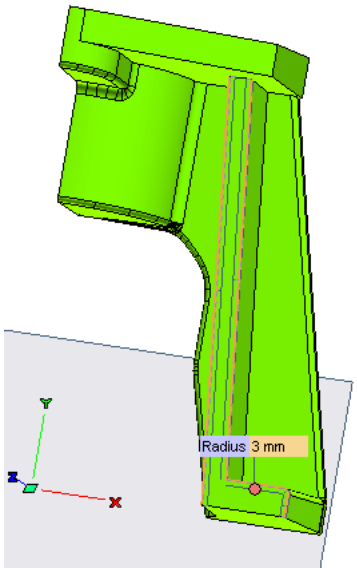
Note :

Larger fillets blend better on smaller radii on external edges, like in this case, referring to the following snap-shot. There's a 10mm radius rolling over a 5mm. In the upper case the edge is internal, that is the 10mm radius overlaps itself; in the lower case it runs without overlapping.

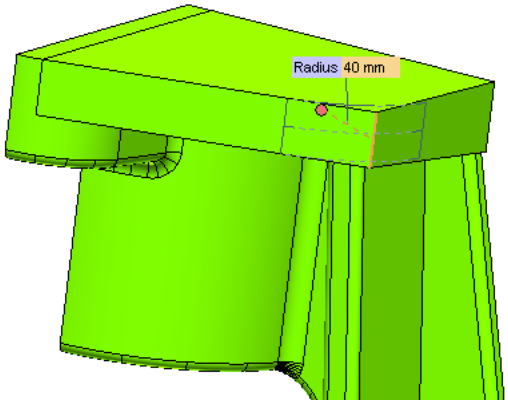


Note that, in lower case we have quality surfaces while in the upper case we do not.

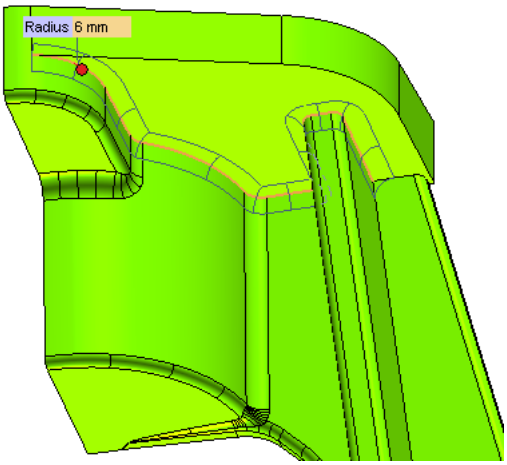
- Start the **Fillet Edges** command.
- Add another Radius3 fillet to the 2 edges as shown.
- Click Apply.



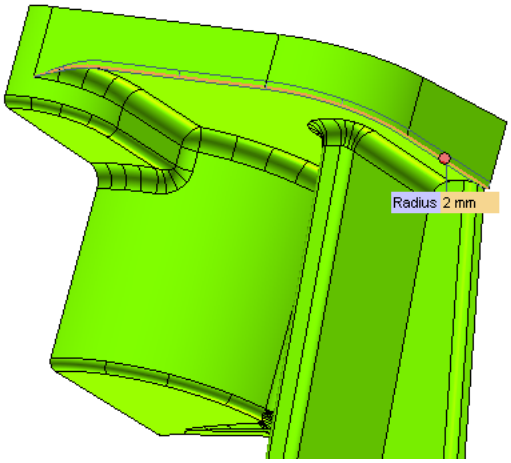
- Add the last Radius40 fillet as shown.
- Click OK.



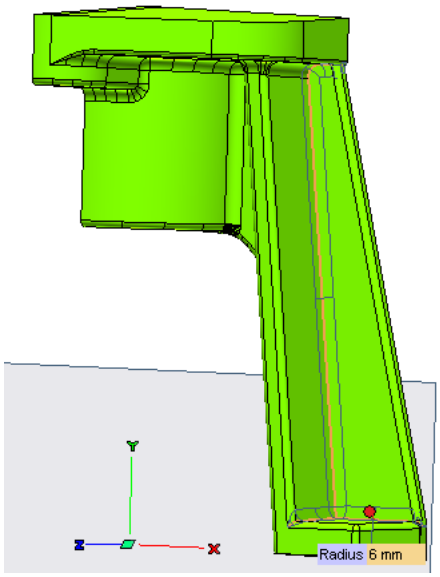
- Start **Fillet Edges** command.
- Create a Radius6 fillet as shown.



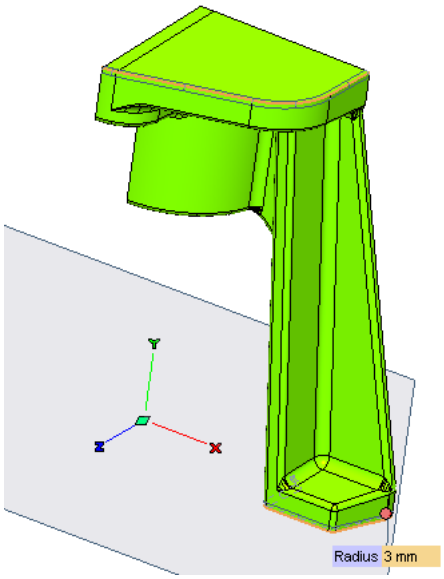
- Create a Radius2 mm fillet as shown.
- Click OK.



- Start the **Fillet Edges** command.
- Add a Radius6 fillet as shown.
- Click Apply.



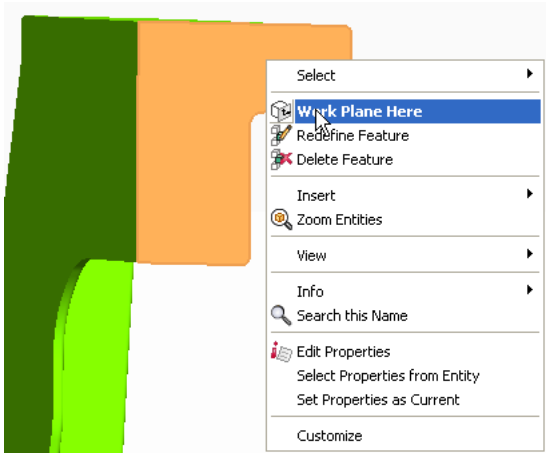
- Add a Radius3 fillet to the edges as shown.
- Click OK.



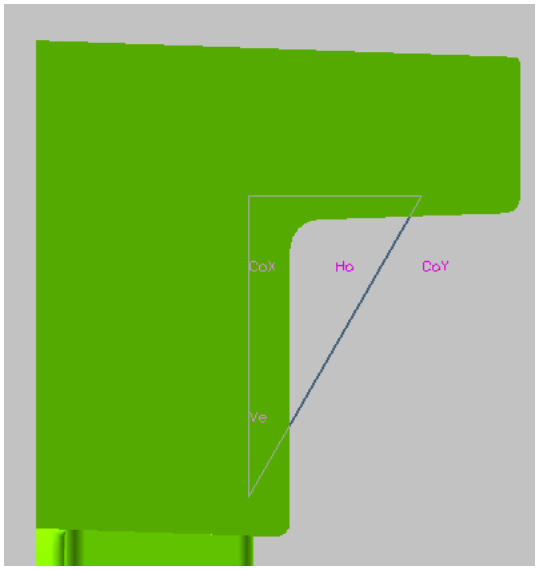
6. Step 6 : Mirroring features and Union mirrored features.

Now let's create a gusset.

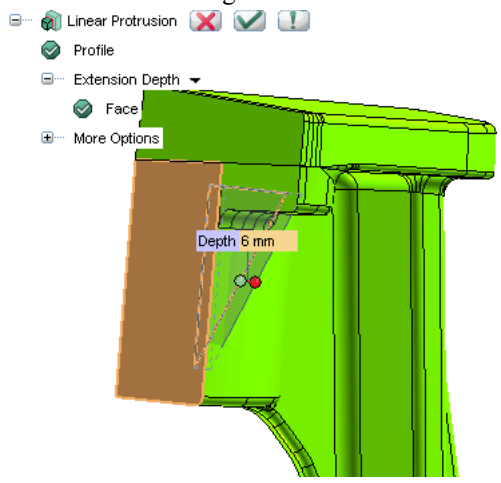
- Unhide the work plane if it is not visible by pressing W in keyboard.
- Right click on the workplane and select on face or datum plane in the context menu as shown
- Select the face highlighted in the image.
- Or right click on the face and select Work Plane Here.



- Hit F8 and F for proper orientation.
- Go to Profile mode.Click on insert menu and select profile 2D
- Using **Polyline** or **Two-point Line**, create a profile as shown.
- Hit Esc.

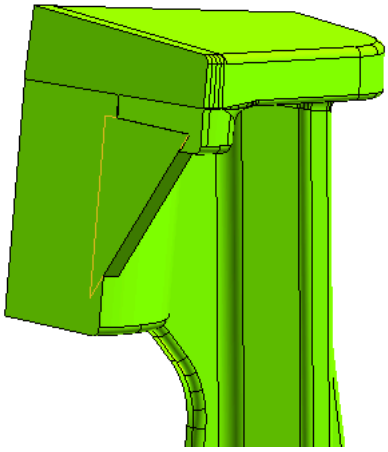


Extrude it 6mm using **Linear Protrusion**.

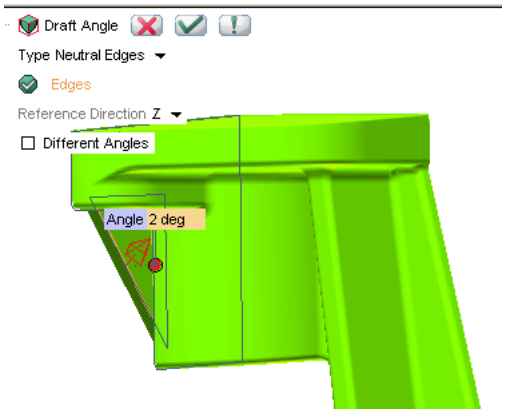


Let's create draft on this face using Neutral Edges option.

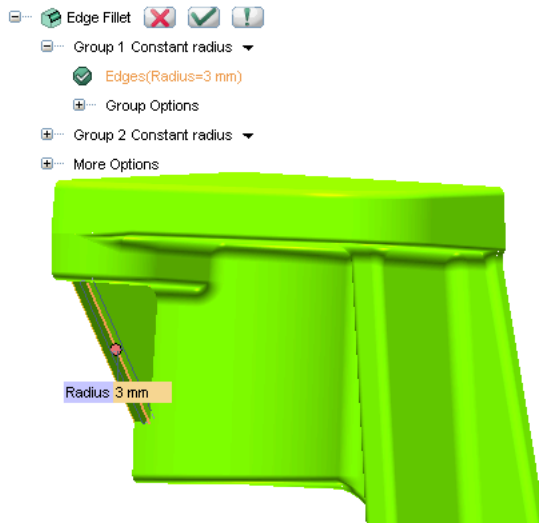
- Right click on work plane and say Absolute reference system(World)



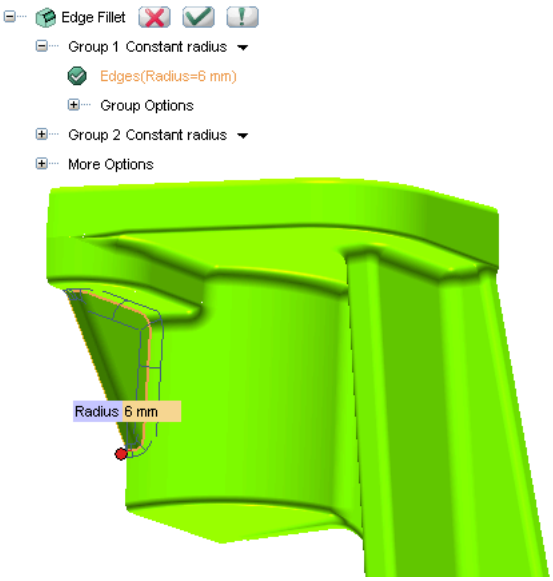
- Activate **Draft Angle**.
- Change the type to Neutral Edges.
- Select the inner edge of the gusset.
- Set the Reference Direction to Z axis in selection list.
- Note the Arrow direction, should be away from the user.
- Let the draft angle be 2 deg.
- Hit OK.



- Activate **Fillet Edges**.
- Select the same edge.
- Set the radius 3 mm.
- Hit Apply.

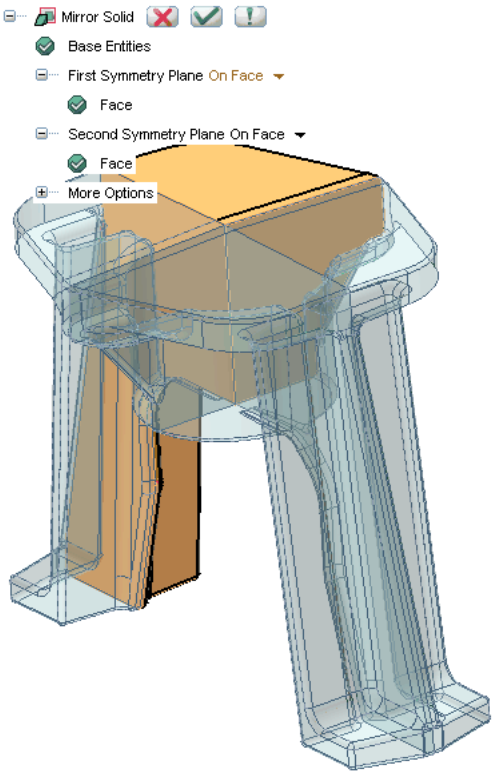


- Now select the inner edge of the gusset.
- Change the radius to Radius6.
- Say OK.



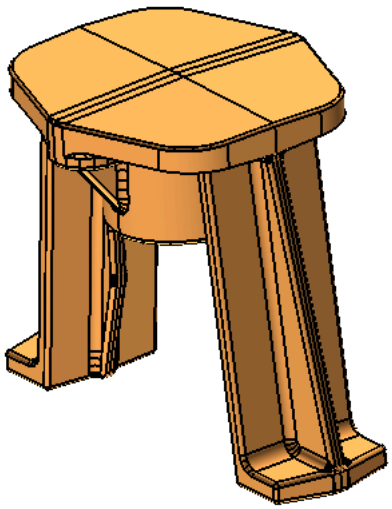
Now let's use the **Mirror Solid** feature on 2 faces and then join all the solids.

- Start the **Mirror Solid** command.
- Select the solid as the Base Entities.
- Set the First Symmetry Plane to On Face
- Select one of the planer faces on the solid.
- Set the Second Symmetry Plane to On Face.
- Select the other planer face on the solid
- Click OK.



Join all 4 parts of this model together using the **Solid Union** command.

- Start the **Solid Union** command.
- Select all the 4 solids.
- Click OK.



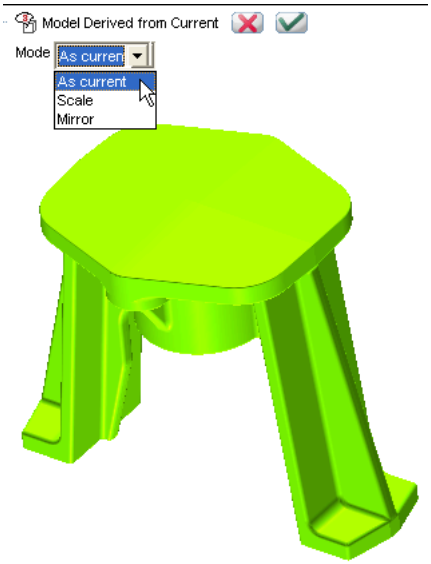
Using the **Unhide Entities** command, unhide all the profiles and dimension lines.

Save the model.

7. Step 7 : Model Derived from Current.

In this step, we will work on a Derived Model. Note that solid features can be created on a Derived model as surely as you do on the orginal solid model.

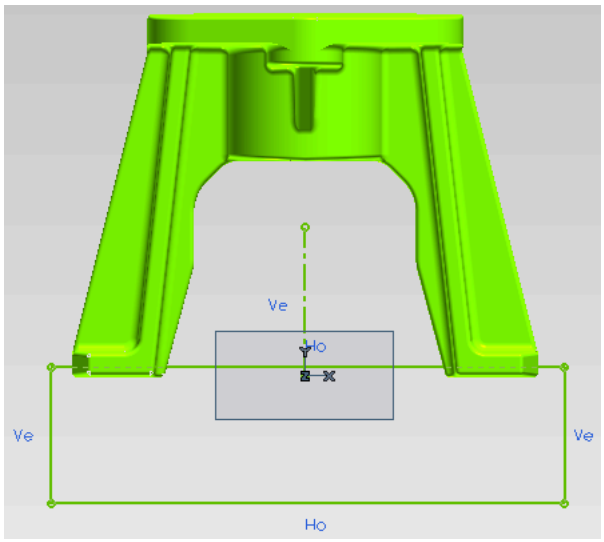
- Start command **Model Derived from Current**.
- Set Mode to As current.
- Select OK.



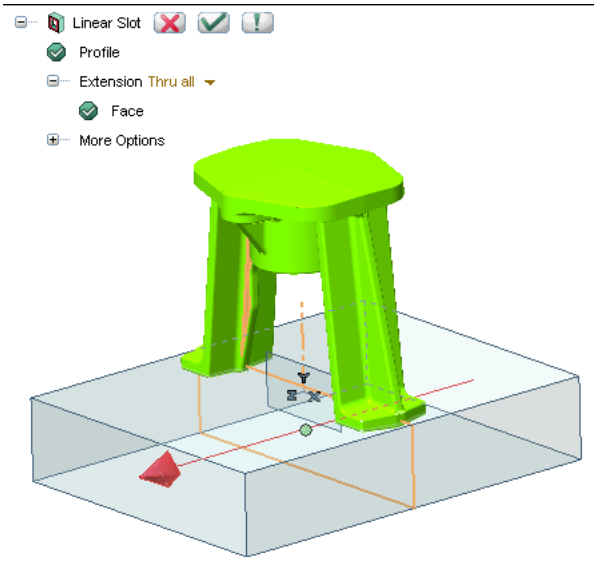
Close the original model. We will continue our work on the Derived model..

Since we have added draft to the legs of the model, the bottoms will not be flat. We will use a rectangle and the **Linear Slot** command to cut the legs slightly so that the bottoms are planer.

- Enter the **Edit Profile** mode.
- Start the **Two-point Line** command.
- Draw a vertical line starting from the **Work Plane Origin**.
- Right click on the line select Make Reference.
- Add the vertical **Orientation Constraint**.
- Constraint the lower point of the line with **Ground Constraint**
- Start the **Rectangle** command.
- Create a rectangle profile as shown.
- Place a dimension between the upper edge of the rectangle and the ground point. Set it to 0.5mm.
- Start the **Linear Slot** command.
- Set the Extension to Thru all.
- Select the solid.

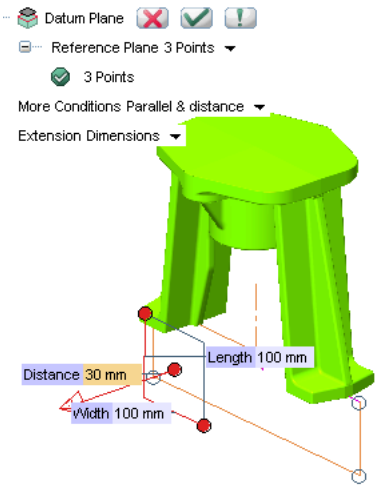


- Right click on the arrow and select Both Sides.
- Click OK.



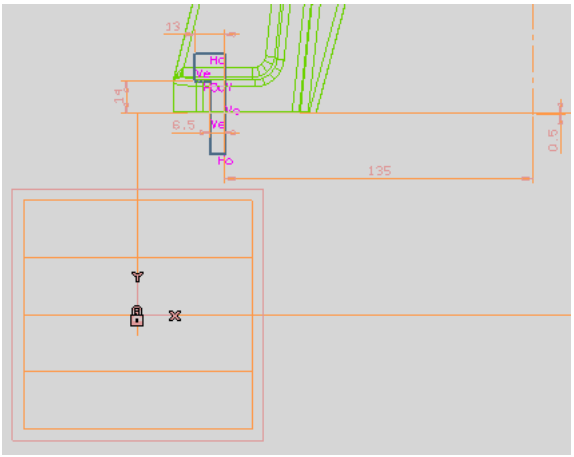
Create a **Datum Plane** parallel to the profile. Offset it by 30mm, from the rectangle. Place the Work Plane on the datum plane and create a profile for a counter bored hole.

- Start the **Datum Plane** command.
- Set the Reference Plane to 3 Points.
- Select 3 corner vertices on the rectangle profile.
- Set the More Conditions to Parallel and Distance.
- Set the Distance as Distance30
- Click OK.

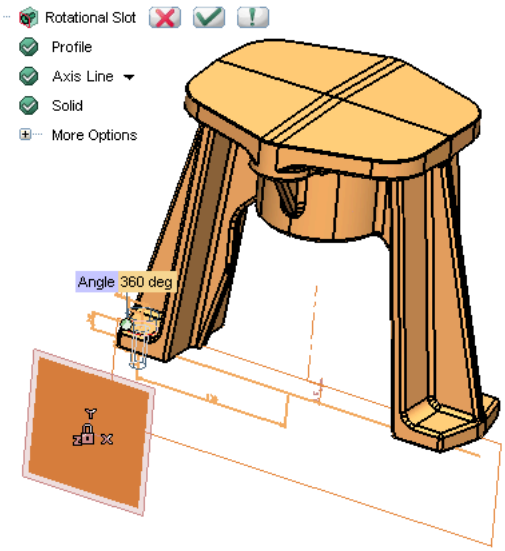


Snap the Work Plane to the datum plane, enter Profile mode and draw a profile for the **Rotational Slot**.

- Right click on the Work Plane and select **Set Work Plane on Face**.
- Pick the datum plane to snap the Work Plane to it
- Enter the **Edit Profile** mode.
- Construct a profile as shown.
- Please refer to the image shown below for inserting the dimensions.

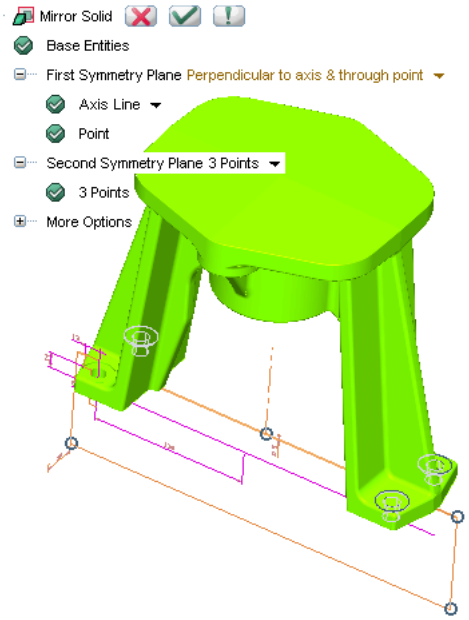


- Start the **Rotational Slot** command.
- Set the Axis to Line.
- Select the vertical line as the axis.
- Set the Angle as Angle360
- Pick the solid.
- Click OK.



Mirror this counter bored hole to both sides using **Mirror Solid** command.

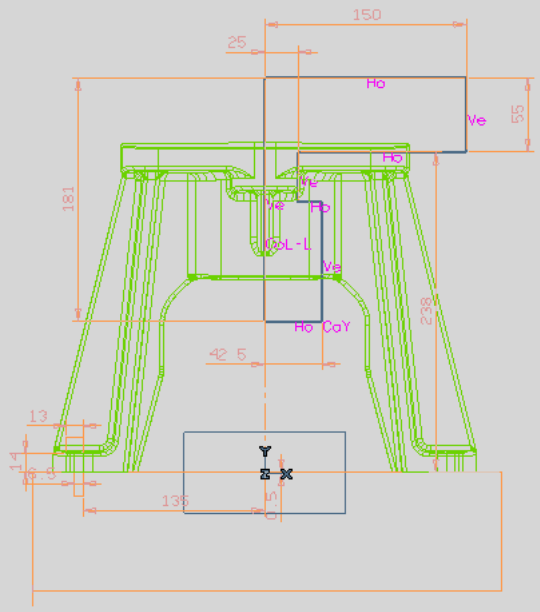
- Set the First Symmetry Plane to Perpendicular to axis and through point.
- Select the horizontal line of the Rectangle profile.
- Select the end point of the vertical reference line for the point, using **End Point Snap**.
- Set the Second Symmetry Plane to 3 Points.
- Select the corner 3 vertices of the rectangle.
- Click OK.



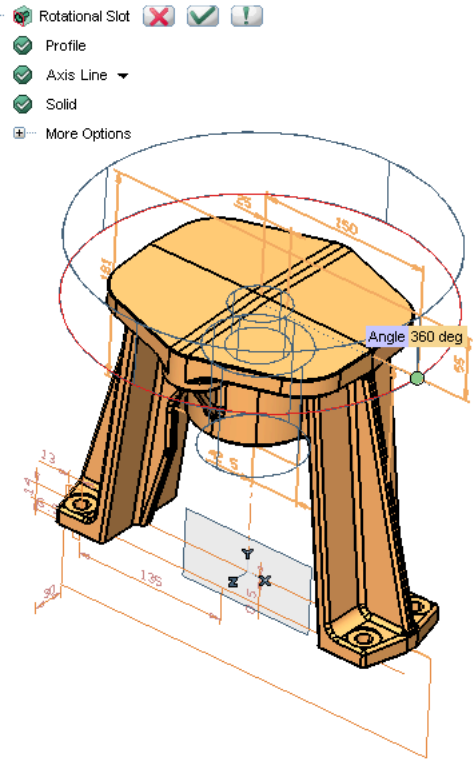
Create another Profile and use the **Rotational Slot** command to cut the hole and trim off the top portion of the model.

- **Set Work Plane to World** .

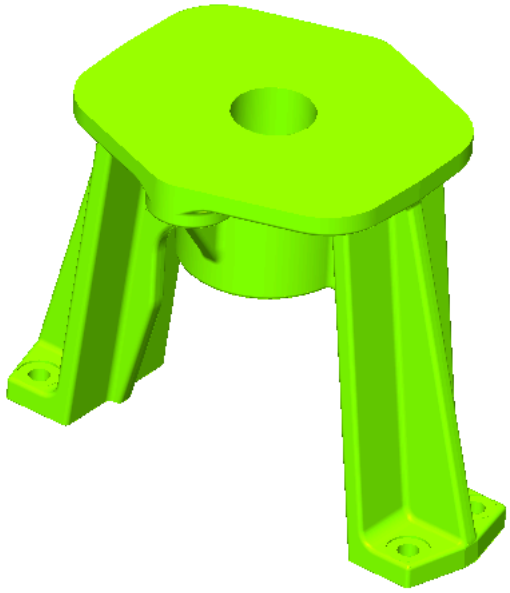
- Enter the **Edit Profile** mode.
- Draw a profile as shown in the image below.



- Start the **Rotational Slot** command.
- Select the Profile.
- Set the Axis to Line.
- Select the vertical line as the axis.
- Set the Angle to 360 Angle360
- Pick the solid.
- Click OK.



Hide all the dimensions and profiles and **Save** the model.



Congratulations!!. this is a fantastic piece of work.