Core and Cavity - I

In this exercise we will cover a fundamental design task, to create a core and cavity mould from a reference part. The task involves working with solids and surfaces and using Boolean functionality to produce the Core and Cavity parts from them. We will also learn about surface normals and how they influence the outcome of the model.

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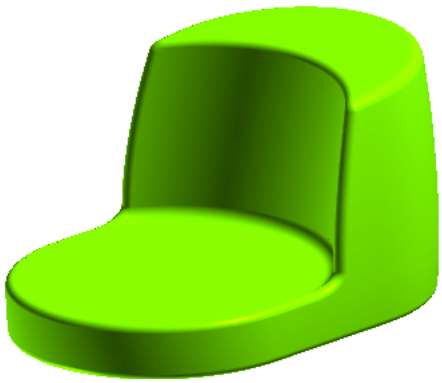
1. Step 1: Analyse Solid Part

The purpose of this step is to illustrate how to analyse the initial model in thinkdesign environment. Inside there will be a manifold static solid.



NOTE:

With a double click on the exe file you can run the webtraining session. ThinkDesign will be open with the right model to start.

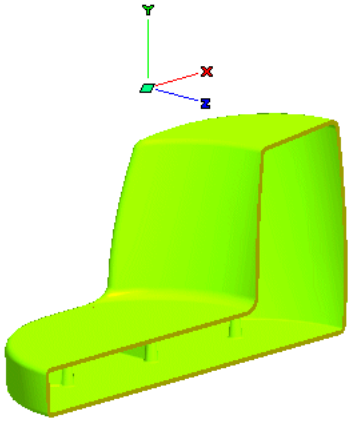
If request to open a file, you can find it in the C:\MyTraining path.

Before creating the core and cavity part, let's analyze the loaded part.

- First, activate the work plane with W key.
- Pick **View**→**Modify**→**Define View Section** and start the command.
- Let's split graphically the solid with a plane choosing it Perpendiculr to Z and passing through **Work Plane**

Origin.

- Check ☑ Show Capping.



- Drag or change Distance minidialog to move the plane.
- Click Reset button to deactivate the clipping view.

Similarly you can try to analyze the solid, using the Define Veiw Section command using other planes.

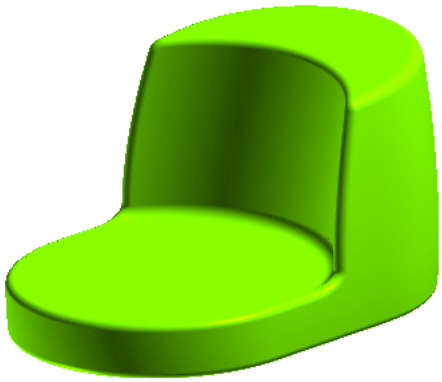
Let's define the position of the parting plane, that helps you to split the solid in two parts and subsequently define the core and cavity. But prior to this, it is necessary to choose the direction for opening the mould. Take Y axis as the reference axis from the world system.

However we have two methods to obtain the same result:

1. Surface modeling.
2. Hybrid modeling.

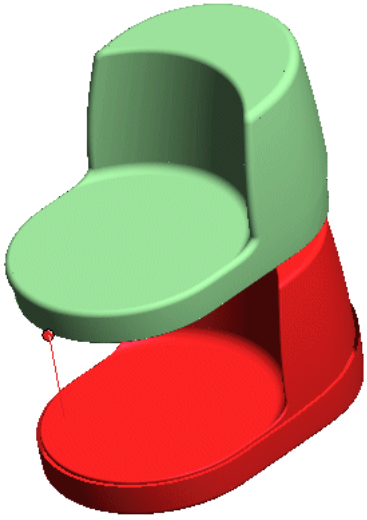
2. Step 2: Parting Curves

To create the parting curve we will use the **Silhouette** command but we introduce some commands to check the 3d model.



Follow the steps.

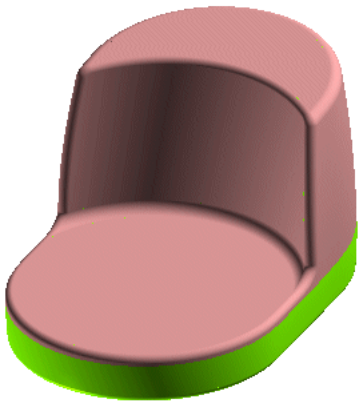
- Change the view mode to **Shaded View** before operation.
- Select **Split with Silhouette**.
- Pick Sight Mode and select Y as Parallel direction.
- Open More Options and then Quick preview.
- Press the Surfaces selection item and select all faces.



- Now check Enable and Shading
- Giving the Separation value as Separation50 mm. thinkdesign splits the model into two bodies, automatically.
- Then press Cancel to close the command.

Now, after preview, make the split.

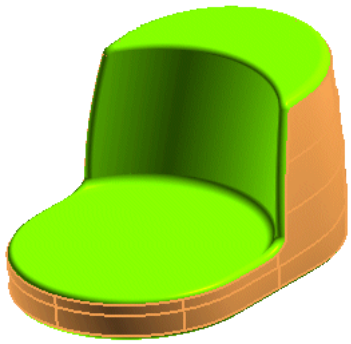
- Select again **Split with Silhouette**.
- Pick Sight Mode and select Y as Parallel direction.
- Pick Surfaces and select now all faces using the **Select Window**.
- Press Preview and then OK.
- Using the **Hide Entities** command, hide the solid. Now in the graphic area there are surfaces that are split as shown in the previous preview.



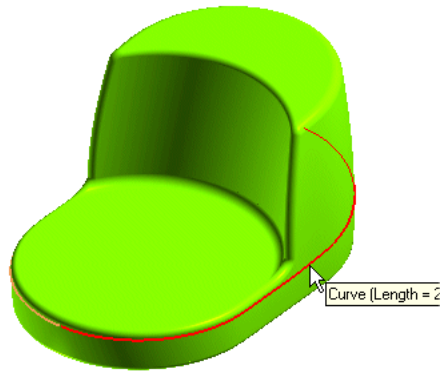
Now the entities can be separated using filters on colors and layers.

But to make the whole process easy, let's use Hybrid modeling.

- Press the **Undo** button to return to the basic static solid.
- Change the current Color to violet.
- Activate the **Silhouette** opation inside Light Contour Curve command. This command works as **Split with Silhouette** command and instead of creating surfaces, we want to obtain the curves.
- Pick Sight Mode and select Y as Parallel direction.
- Select two Surfaces as shown below.



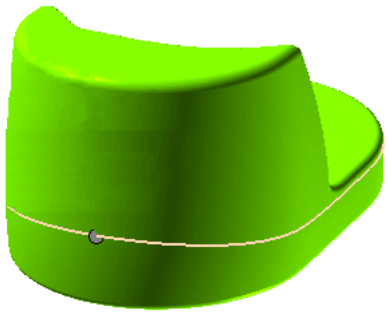
- Check Associative.
- Press Preview and then OK. You will get two curves as shown in the image below.



NOTE:

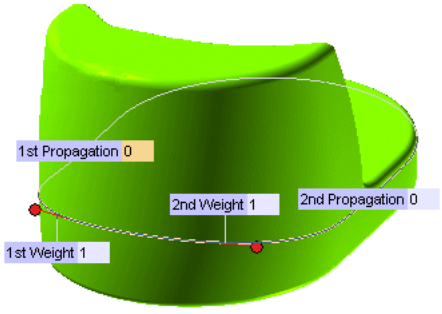
Always check and correct the continuity, tangency when required, between the two curves of silhouette.

Go to **Tools**→**Info**→**Check Continuity**command, choose Type Curve - Curve and select the two curves near in the same end points.



The previous command shows a cuspid point between two curves and now using **Curve Continuity** we want to reduce this.

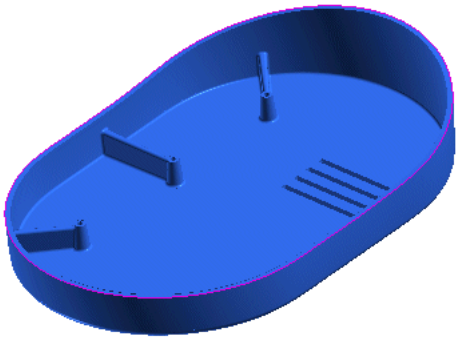
- Click the **Curve Continuity** command.
- Check Associative.
- Select the two silhouette curves, near the intersection point, as the Curves to be modified.
- Set Degree: as Tangency and MethodAdjust both.
- Click OK.



Go to **Tools**→**Info**→**Check Continuity**again, selected same curves, to analyse the continuity for better values.

3. Step 3: Split Solid

We're now going to add the Parting Plane Surfaces to our solid. Our goal is to use them to split the solid into two bodies and then create Core and Cavity in the lower part.



We need to identify all the edges that are required to define the parting plane.

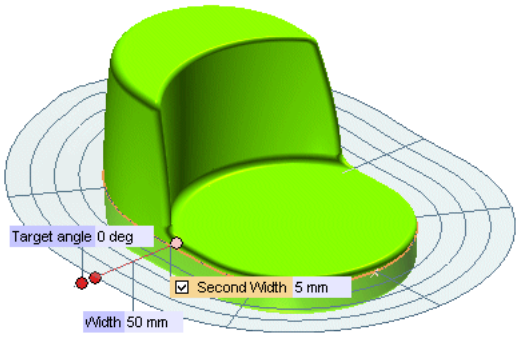
- Change your current Color to blue and activate the work plane.
- Start the **Parting Plane** command.
- Check ☑ Associative.
- Set the Direction to Y.
- Expand More Options.
- Set the Generation method to Standard.
- Set the Min fill. width as 10.
- Set the Min. radius as 10.
- Select the two silhouette curves.
- Leave all others minidialogs with default values.

Tip:

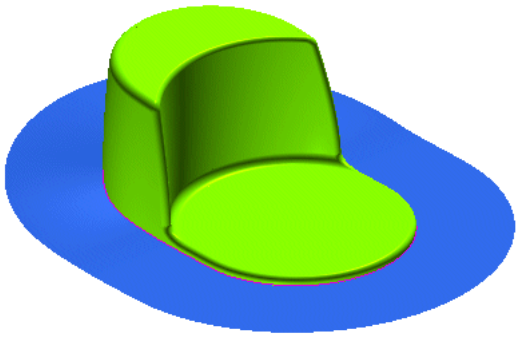
If you have accidentally picked an edge, that was not required, you can unselect it by holding the Ctrl key down and selecting that particular edge again.

Now let's have a preview of this outcome.

- Click the Preview icon in the selection list to see the results. (Ignore the possible warning message "Curvature radius smaller than requested").
- Should the surfaces be generated on the inside of the edges, right click on the Width10 mini dialog box and select Invert if necessary.
- If you had to invert the direction in which the surfaces are being created, click the Preview icon again to see the updated results.
- Change the mini dialog with a new value as Width50
- Check Second Width with 5mm.
- Click again the Preview icon.



- Click OK to accept this method.

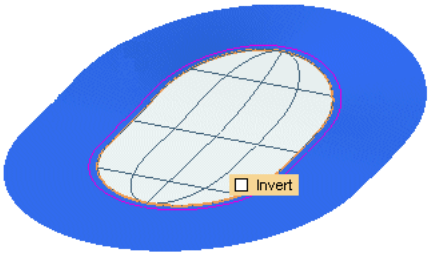


TIP:

To learn more about the selection list options for the **Parting Plane** command, please see: Creating a Parting Plane using thinkdesign Help section.

- Using the **Hide Entities** command, hide the solid.
- Start the **Lofted Surface** command.

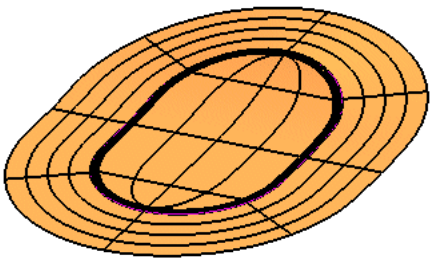
- Check Associative.
- For selecting Boundary Set A select the two edges of the inside hole.



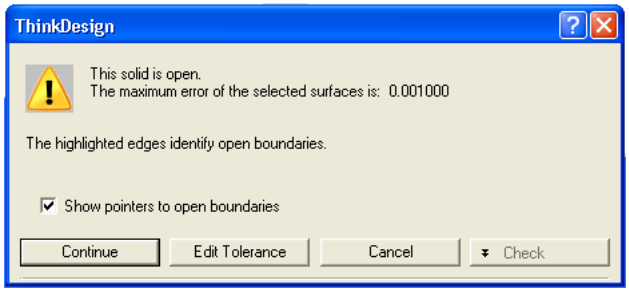
- Use the Invert mini dialog, if necessary, to invert the orientation of edges.
- Click OK.

In this step, we're now going to make a unique reference entity to split the main solid into two parts.

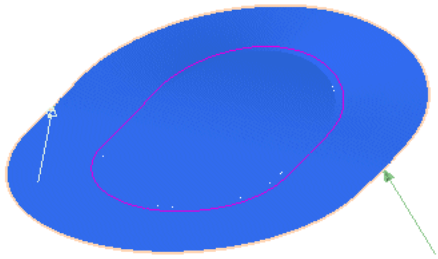
- Start the **Make Solid** command.
- Select the surfaces.
- Check Associative.
- Click OK to end the command.



- When the Solid is Open an alert dialog box appears, check Show pointers to open boundaries to look for open edge.



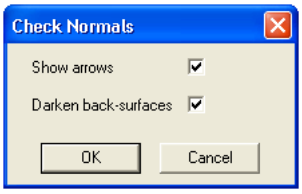
In this case they are two external edges, as indicated from the arrows.



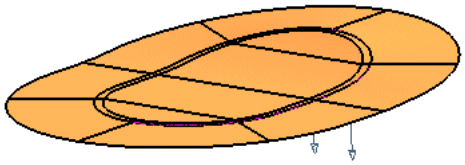
- Click Continue.
- Now look at the History Tree to make sure that there are two solids (one Static solids and one Skin.

Especially with open solid and using the Boolean features, it is always necessary to know where the hypothetical material is. We can find it using the Check Normals option.

- Pick on **Tools**⇨**Modeling**⇨Check Normals.
- Select the parting plane - open solid.
- Check the ☑ Darken the back-surfaces option if it isn't already checked.



- Check the ☑ Show the arrows.
- Click OK, two times to exit.



Rotating the model you'll find two different colors, bright and dark. One represents the material side while the other the external side. The external side can be shown also by normal arrows. See the above image.

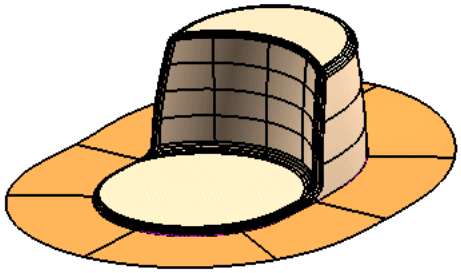
TIP:

If you have the Parting plane Normals oriented in the positive Y-direction, there are two possibilities: use the Parting plane with **Insert**⇨**Solid**⇨**Boolean**⇨**Intersect**command without inverting the normals or invert the normals and use it with **Insert**⇨**Solid**⇨**Boolean**⇨**Difference**command as explained below.

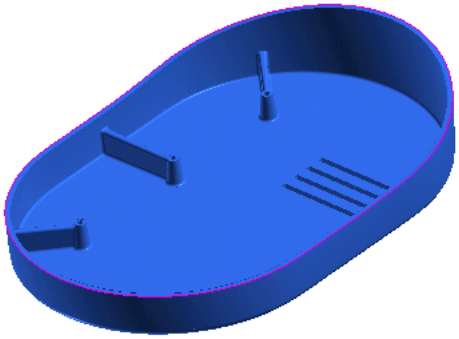
Use **Modify**⇨**Solid**⇨Invert Normals to fix the normals.

Now we can split the main solid.

- Click the **Insert**➪**Solid**➪**Boolean**➪**Difference**command.
- Pick the main solid as Solid A, then the parting solid as Solid B.
- Click the Preview icon to check how the model looks like.



- Click OK.



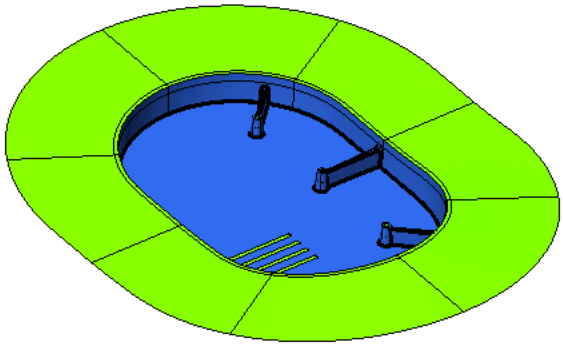
TIP:

If you want to have the upper body you have to use **Insert**➪**Solid**➪**Boolean**➪**Intersect**with normal to below or **Insert**➪**Solid**➪**Boolean**➪**Difference**with opposite direction and same sequence of selection.

Eventually check the Keep Original Solids option in the previous boolean feature if you want try to obtain also the upper side.

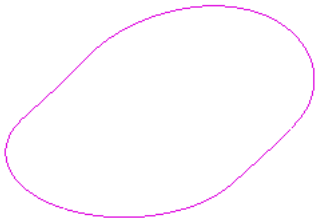
4. Step 4: Parting Planes and Holes

We're now going to add the Parting Plane Surfaces to our solid. Our goal is to use them to split the lower solid into core and cavity.

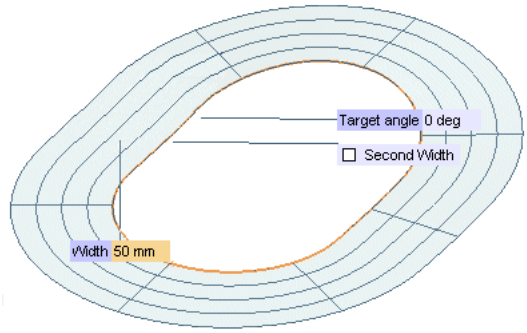


We need to identify all the edges that are required to define the parting plane.

- Using the **Hide Entities** command, hide the solid.

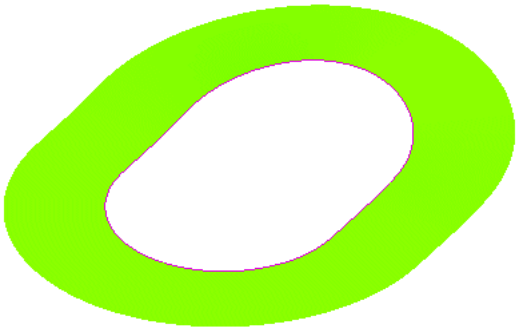


- Change your current color to green and activate the world axis.
- Start the **Parting Plane** command.
- Check ☑ Associative.
- Set the Direction to Y.
- Expand More Options.
- Set the Generation method to Standard.
- Set the Min fill. width as 10.
- Set the Min. radius as 10.
- Select the two violet curves, that were created before with **Silhouette** command and check for continuity.



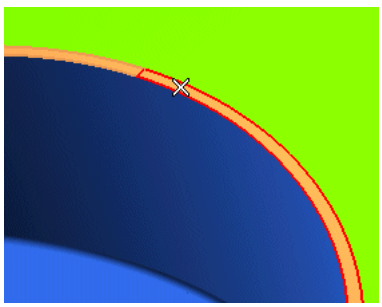
- Click the Preview icon in the selection list to see the results. (Ignore the warning message "Curvature radius smaller than requested").

- Change mini dialog with a new value of Width50
- Uncheck Second Width.
- Click again the Preview icon.



- Click OK to accept.

- Using the **Unhide Entities** command, unhide the main solid.
- Start the **Solid from Faces** command.
- Sele Mode as Local.
- As Faces select two border faces.

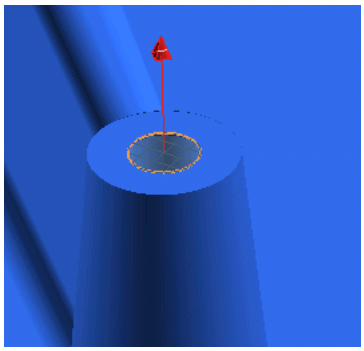


- Check the Linked copies.
- Click OK.

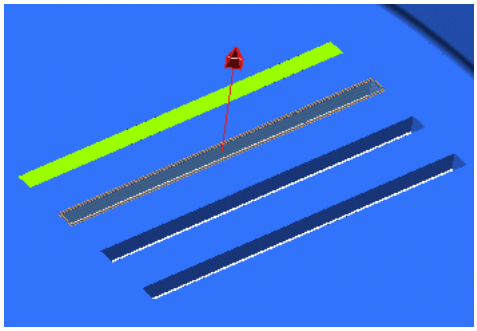
Now in the history tree you could see a new Skinz solid that joins the selected faces.

In this step, we're going to seal the holes in the part creating planar surface.

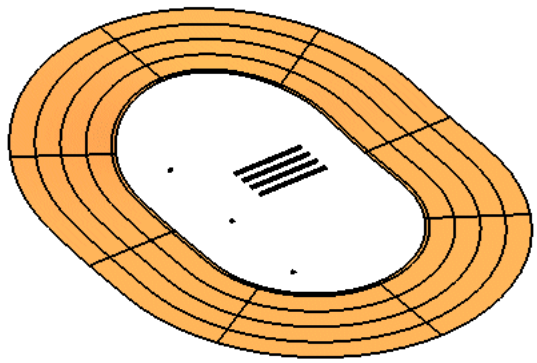
- Start the **Plane Surface** command.
- Check Associative.
- Select one hole edge as Boundaries.
- Keep the arrow in the upward direction or invert it if necessary.
- Click OK.



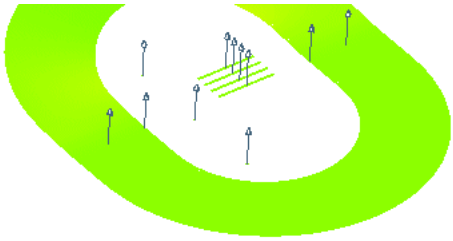
Repeat the same command for others two holes and four slots, as shown below.



- Using the **Hide Entities** command, hide again main solid and also all the curves.
- Start the **Make Solid** command.
- Select all surfaces.
- Check Associative.
- Click OK.
- When the Solid is Open alert dialog appears, click Continue.



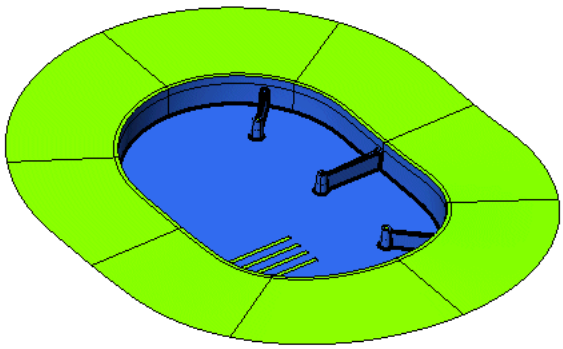
- Pick on **Tools**➪**Modeling**➪Check Normals.
- Select the parting plane - open solid.



TIP:

Use **Modify**➪**Solid**➪Invert Normals to fix the normals.

- Using the **Unhide Entities** command, unhide the main solid.



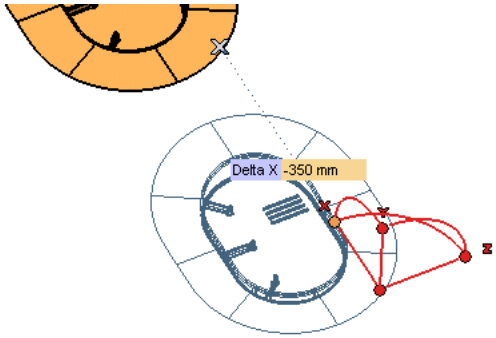
5. Step 5: Preparatory Operation

We're now going to create a solid from the surfaces made in the last sections and then create the Core and Cavity for the mold.

These two solids are going to be used to create our Core and Cavity. Let's create one more copy of the same solids so that we can create a Core and Cavity separately in this same model.

We'll use the Move/Copy command to make a copy of the solids and place them at a distance of 350 mm in the opposite X direction.

- Start the **Move Copy Entities** command.
- Select the main plastic body and the parting solid using the **Select Window**.



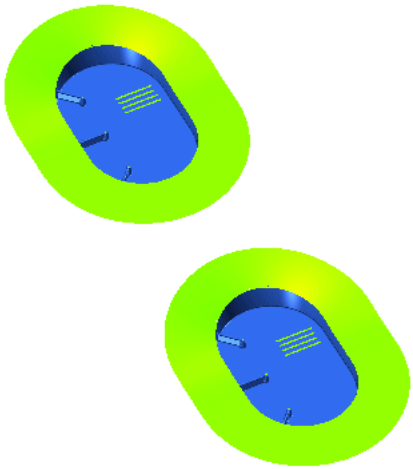
- Click on the X handle as shown. This will expose the Move X minidialog.
- Enter Move X-350

NOTE:

The location of the Origin for the Move/Copy handle is not important for this particular operation.

- Check the Copies option and enter 1.
- Check the Associative Transformation option.
- Pick More Options and check the Linked copies option.
- Click OK.

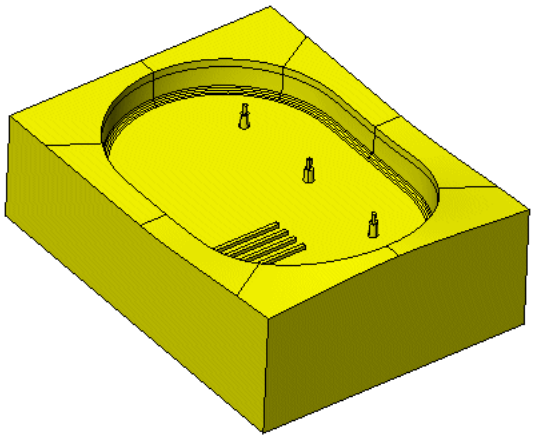
You should be left with two solids as shown.



Now from the history tree, select with right mouse button, a Transformation feature inside a new solid and choose Unlink to split parametric reference with basic solids.

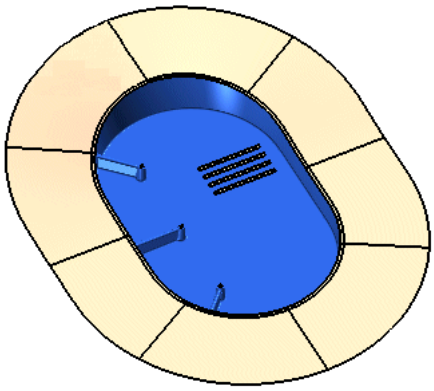
6. STEP 6 : Creating the CAVITY.

Now, let's create a Core from top part and Cavity from bottom part.



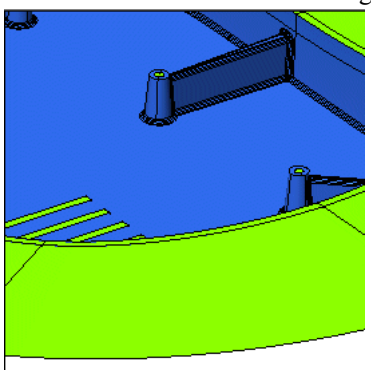
Now let's do a Boolean operation between the parting plane surface and the main part.

- Click the **Insert**➪**Solid**➪**Boolean**➪**Difference**command.
- Pick the parting faces as Solid A, then the main part as Solid B.
- Click the Preview icon to see what the model will look like.

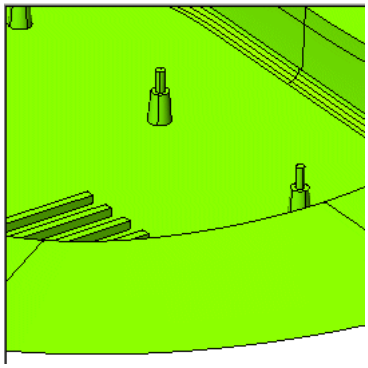


- thinkdesign will display the message "This solid is open".
- Since we know the solid is open click OK to finish the command.

Look at the result. This is the image of the solid before the Boolean operation

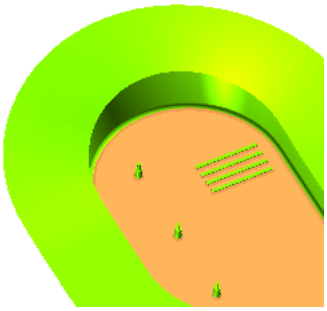


Same view but with entities after difference feature.



Now, let's create a base for a Cavity. We need to create some 2D geometry for the Cavity base.

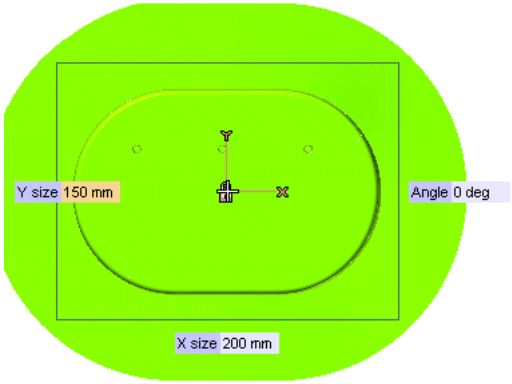
- Press the W key to display the Work Plane.
- Select with right button of the mouse the inner bottom face and choose Work Plane Here



- Press F8 to set the view to the top view of model

The Work Plane appears to be in a good position for creating geometry for the base. Change current Color to blue for better contrast.

- Start the **Rectangle** command.
- Set the Mode to Cen+Sizes.

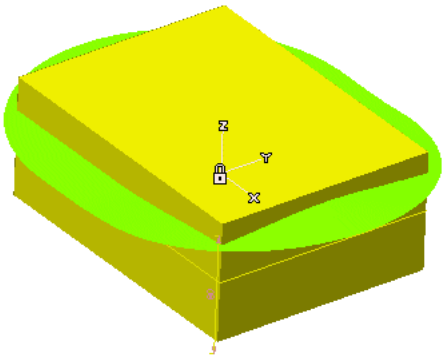


- Insert X size200 and Y size150
- And choose **Work Plane Origin**

- Press Cancel to exit from command.

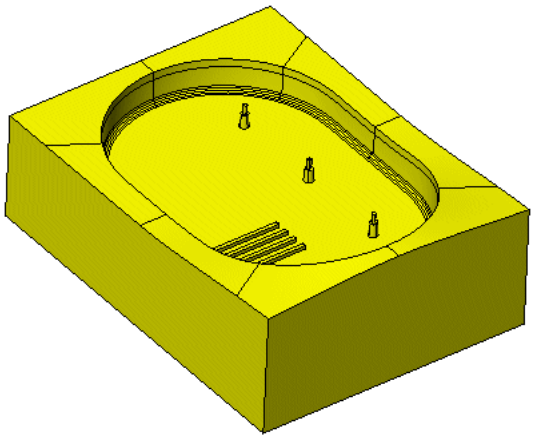
Rectangle has to be inside the parting area and outside the main solid.

- Change current color to yellow.
- Start the **Linear Solid** command.
- Select the rectangle to extrude. If you have problem selecting it switch to **Wireframe View** mode.
- Set the Depth80
- Right click on Depth80 mini dialog box and select the Symmetric option.
- Click OK.



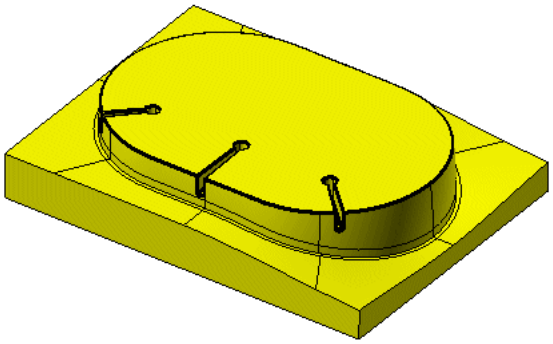
- This solid will work as a base for Cavity.
- Click on **Insert**→**Solid**→**Boolean**→**Intersect**.
- Select both the solids one by one.
- Click OK.

The final result of this operation will look like this.



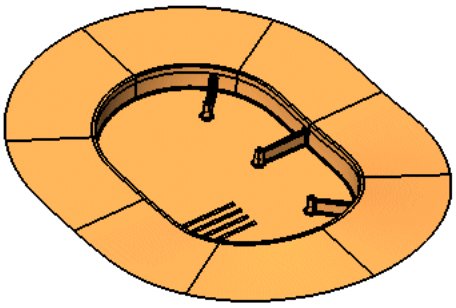
7. STEP 7: Creating the CORE.

Let's create the Core for the model now. We will have to repeat the same steps as we did before.



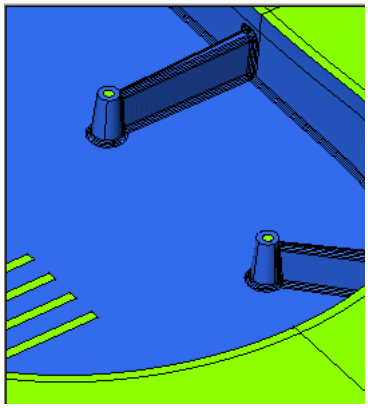
- Click **Insert**➪**Solid**➪**Boolean**➪**Union**command.
- Select both the solids.
- Click the Preview icon to see how it looks.

52_1.gif

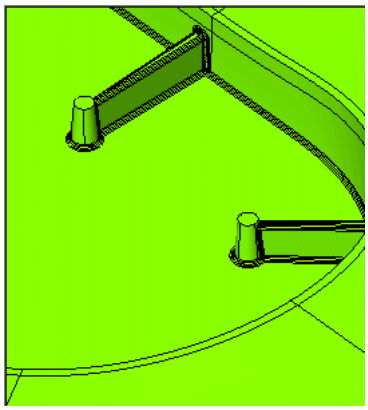


- thinkdesign will display the message "This solid is open".
- Click OK.

Look at the result. This is the image for solid before the Boolean operation



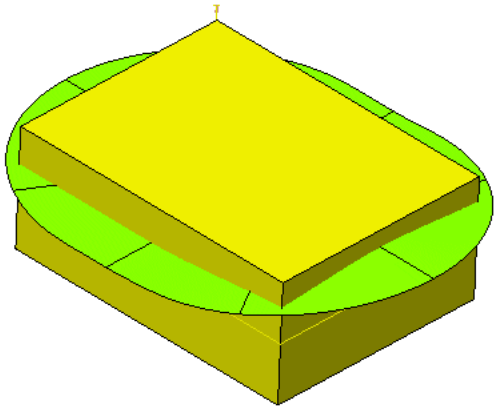
Same view but with entities after union feature.



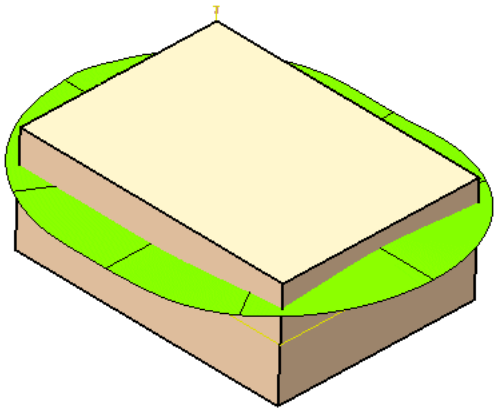
Let's create a base part for the Core now.

Now we'll have to repeat the same steps to insert a rectangle,as we did for the core.

- Press the W key to display the Work Plane.
- Select with right click, the inner bottom face and choose Work Plane Here
- Start the **Rectangle** command and then **Linear Solid** command with a symmetry depth of 80 mm.

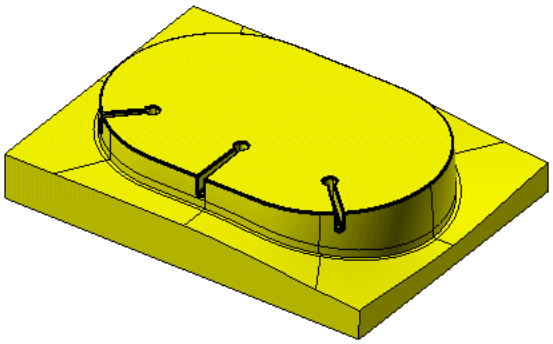


- Click **Insert**➪**Solid**➪**Boolean**➪**Difference**command.
- Select Core base as Solid A and parting plane surface as Solid B.

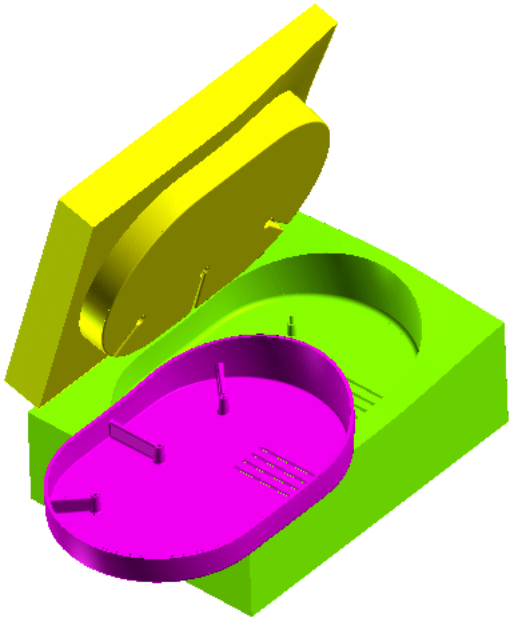


- Click OK.

Rotate the model and notice how the bottom looks.



Unhide the Cavity to have a final look at the parts created.



Congratulations!!! You have finished a daunting task on Core and Cavity. Now try to repeat the same steps to create the upper core-cavity starting from Step 2.