2D/3D libraries: Creation and Management

In this task, you will learn how to insert a model into the thinkparts catalog. You will create a parametric model of a simple bearing, add variable expressions and symbolic mating constraints and save it as a part family.Then, you will create several sizes in the part family and test it out by importing it back into a new model. This task builds on the Introduction to thinkparts task. If you are not familiar with thinkparts, you should work through that task first.

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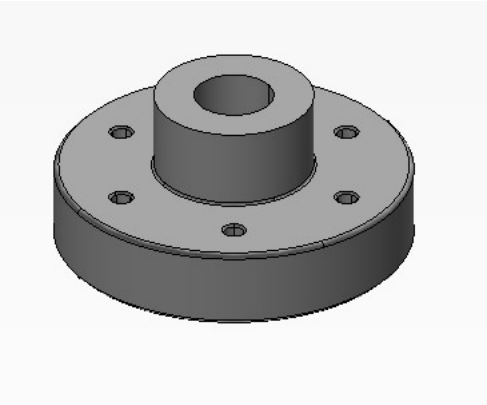
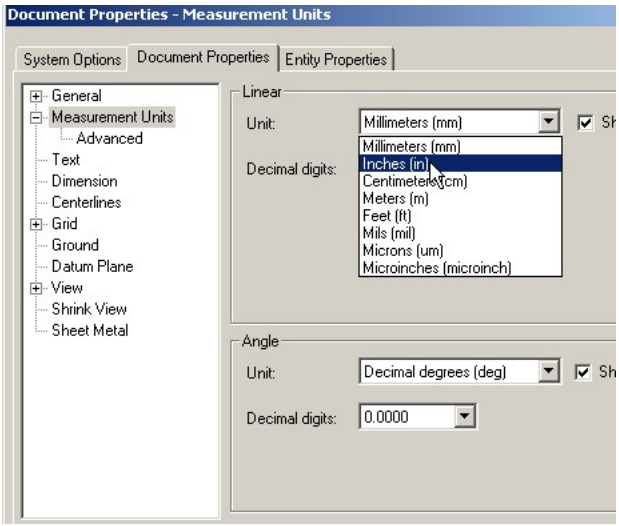
1. Step 1: Create a Parametric Model

The first step is to create the model.

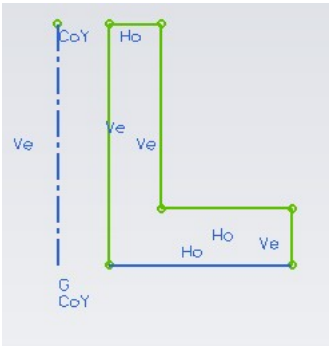


- Start a **New Model**
- Right click in the graphics background and from the choices that appear, select **Options/Properties**. Then, on the Document Properties tab, Measure Units item, set the Unit of measure to Inches and Click OK on the dialog.
- Click the Profile tab.



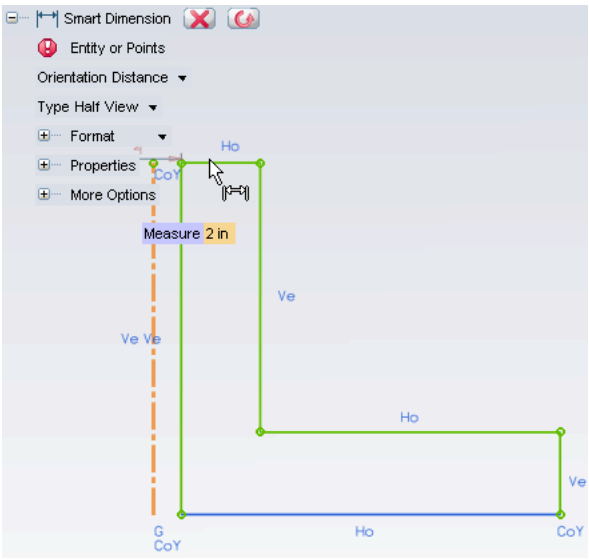
Sketch the profile for our bearing. The World Work Plane Origin will be the insertion point of the new part, so we shall use it in the sketch.

- Start by creating a vertical reference line Vertical90, with the lower endpoint at the **Work Plane Origin**.
- Right click on the line just created and select the option to Make Reference.
- Add a **Ground Constraint** to keep the reference line at the origin.
- Use **Polyline** to create the 'L' shaped profile.
- Add Y Point **Coincident Constraint** between each endpoint of the vertical reference line and the corresponding horizontal edges of the profile.

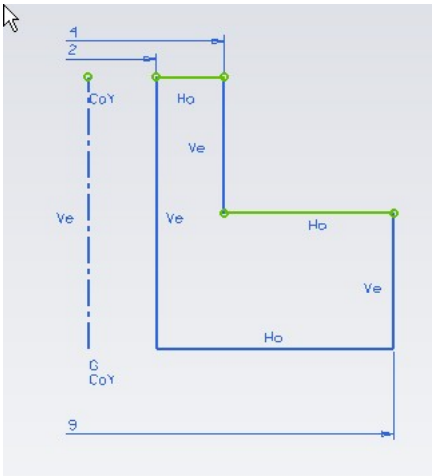


We are going to revolve this profile about the vertical reference line, so there are three places where we want diameter dimensions. Let's add those **Smart Dimension** constraints using the Half View option.

- Start the **Smart Dimension** constraint command and select the vertical reference line.
- Now select the upper left corner point on the profile, but do not place the dimension yet.
- In the Selection list, set the Type to Half View, then place the dimension above the profile.

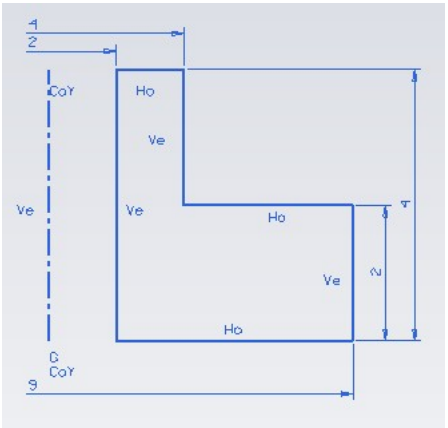


- Repeat the process for the other two diameter dimensions.



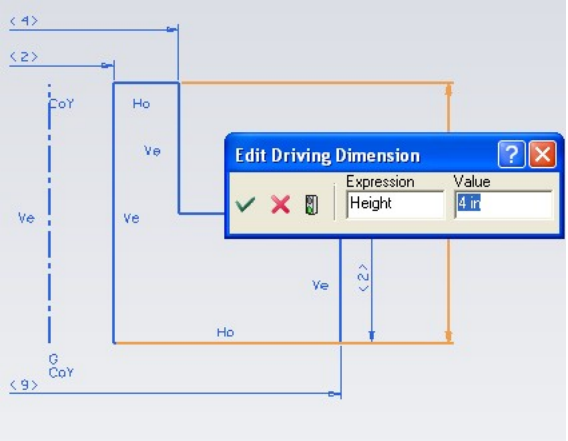
Finish constraining the profile by adding the two height dimensions shown below.

- With the **Smart Dimension** constraint command still active, add a dimension for the overall height and the flange thickness.
- Make sure the profile is fully constrained with a **Check Profile**.



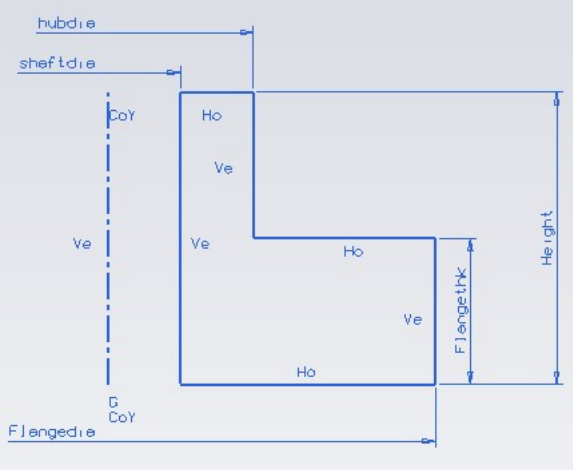
Now finish the Profile by adding the variable Expression to the appropriate dimension.

- Double click on the overall Height dimension value.
- In the Expression field, type in Height, and set the value to 7.5 and then hit OK.



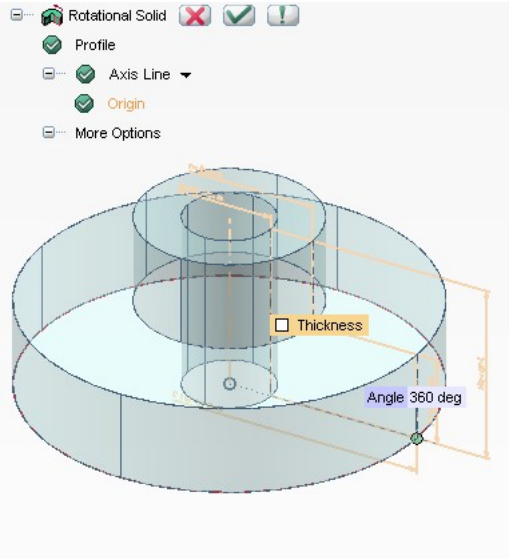
Repeat for the other dimensions, setting the values shown.

- HubDia 6.
- ShaftDia 2.
- FlangeDia 14.
- FlangeThk 2.



Now create a **Rotational Solid** with the profile.

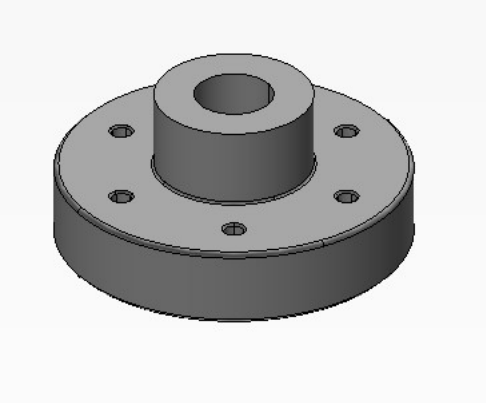
- Select the vertical reference line as the Axis with an Angle of 360. Angle360.
- Hit OK



In the next step, we shall finish the model and size it correctly.

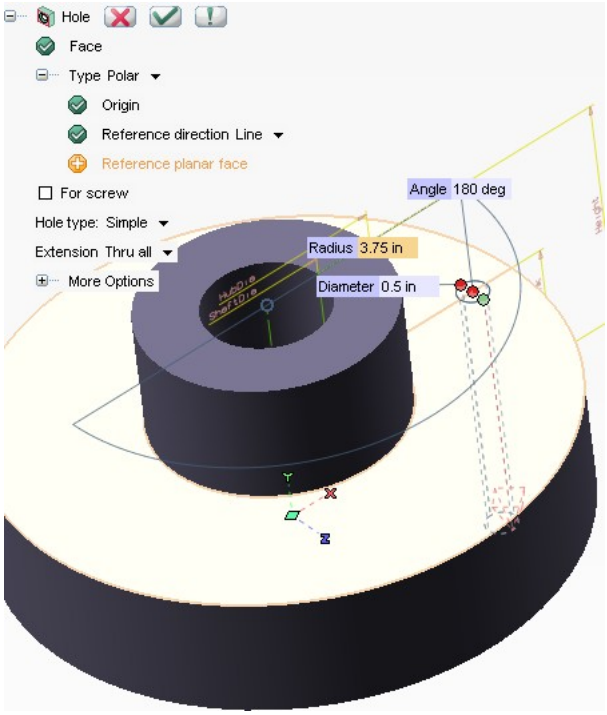
2. Step 2: Finish the Model

In this step, we shall finish up the bearing by adding the hole pattern and some fillets. Then we shall apply the correct sizes with the **Spreadsheet**.

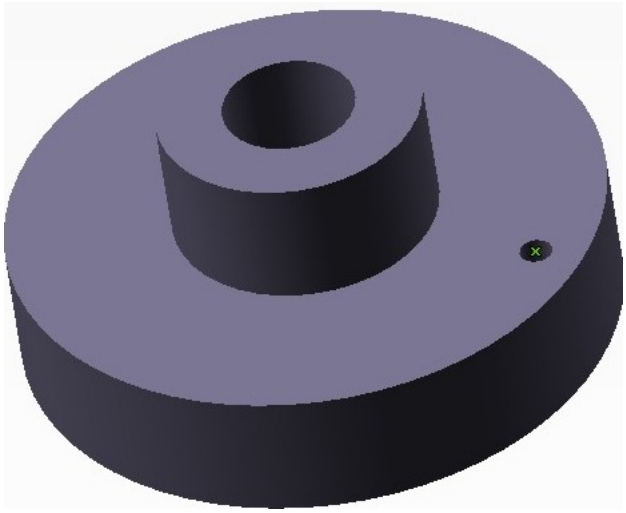


We need a way to fasten our bearing, so let's add a hole pattern to the flange.

- Select the **Hole** command. For the Face click on the top face of the flange, on the Selection List, set the Extension option to Thru All and set the Diameter of the hole to 0.5. Diameter0.5.
- On the Selection List, set the Type to Polar.
- For the Origin, use **Snap to Arc Center** to select the flange center as the reference start point for the Polar location.
- For the Reference Direction Line, click the original top flange geometry line.
- Finally, for the location Radius for the hole, enter 3.75 Radius3.75 and an Angle of 180. Angle180 then click OK.

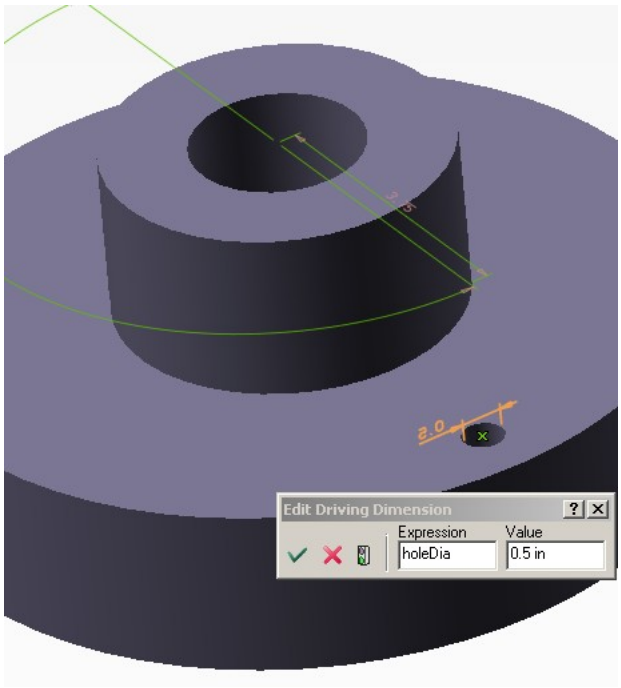


The final result should look like this.



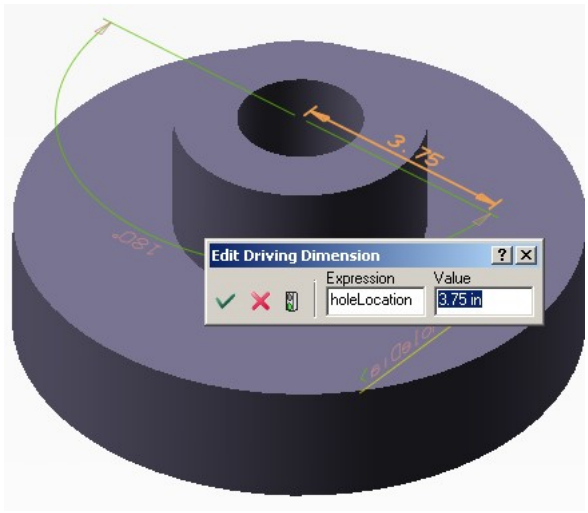
Add a variable expression for the hole diameter.

- Double click on the hole's diameter dimension.
- Type in HoleDia in the expression field.



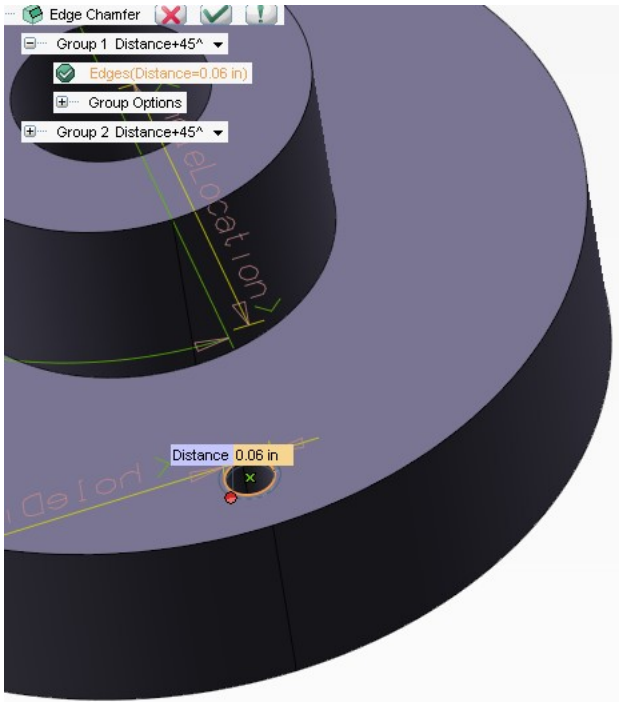
Make a variable expression for the dimension that locates the hole created in the last step.

- Change to **Wireframe View** so that you can see the dimension to edit.
- Double click the location dimension and enter Hole Location as the Expression for the radial distance from the centerline of the part to the centerline of the hole.
- Click either OK or **Rebuild Model** to complete the command.



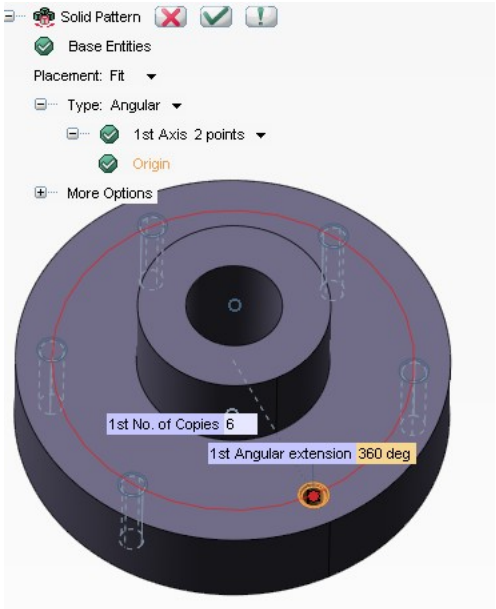
Finish the hole with a **Chamfer Edge**.

- Change your view to **Shaded View and Boundaries**.
- Use **Chamfer Edge** to add a 0.06 chamfer to the top edge of the hole. Distance0.06.
- Click OK.

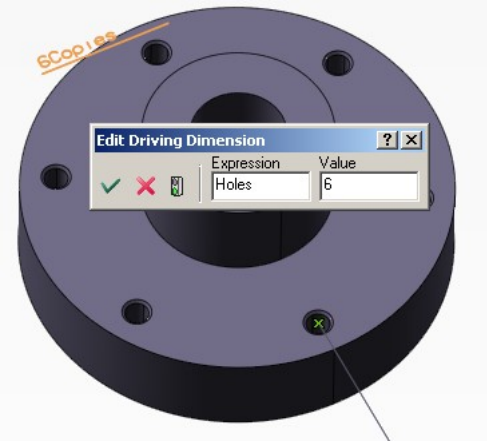


Create a **Pattern Solid**, making the number of holes variable.

- Pick the hole and chamfer for the Base Entites.
- Create an angular **Pattern Solid** of both the hole and the chamfer using the bearing profile reference line as the 1st Axis Line. Change Placement: to Fit with 6 total copies 1st No. copies6 spaced around 360 degrees. 1st Angular Extension360.
- Hit OK to complete the command.



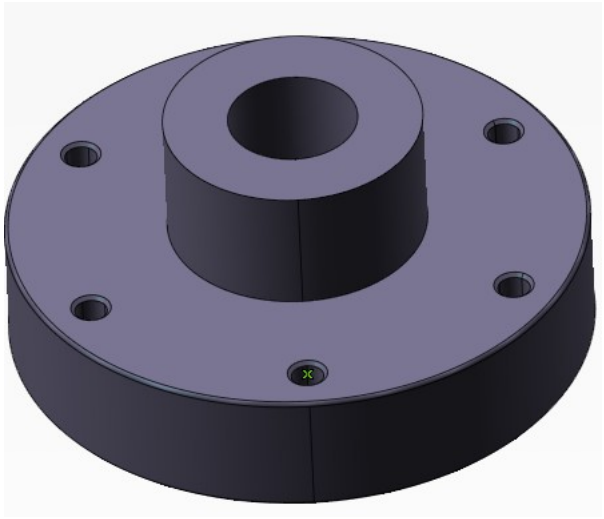
Double click the value for the number of Copies of the hole in the pattern, entering Holes as the Expression.



Finally, break the two outside edges with a 0.06 radius by using the **Fillet Edges** command, then add a 0.12 fillet using the **Fillet Edges** command on the inner corner.

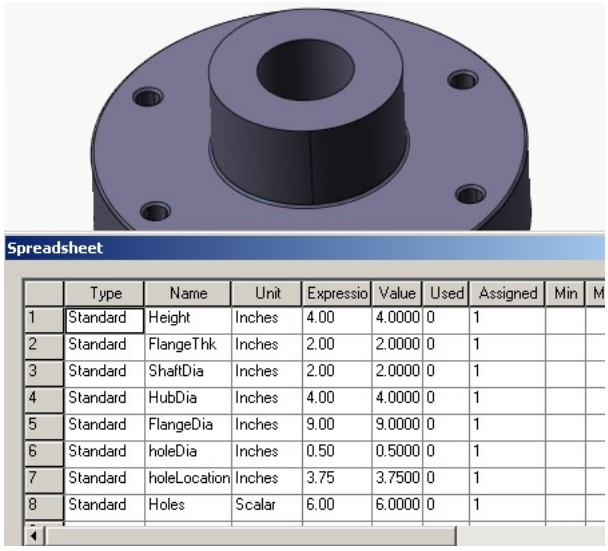
- Add the 0.06 radius to the two outside edges with the **Fillet Edges** command. Radius0.06.
- With the selection list still active, click New Group beneath Group 2, then select the Edges and add the 0.12 radius. Radius0.12.

- Hit OK to complete the command.
- When you are done, use **Hide Entities** to hide all the dimensions and the profile.



To finish it up, let's use the **Spreadsheet** to size the bearing correctly.

- Start the **Spreadsheet**.
- Hit on Name for Sorting the order to Alphabetic.
- Now change the Expression for the Hole Location to be (FlangeDia+HubDia)/4.
- Finally, make the changes to each of the Expression values for the variables listed as shown in the figure, below.

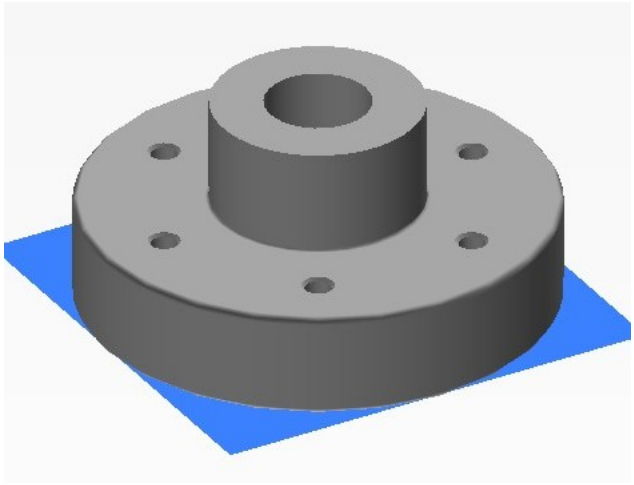


- Hit Rebuild to update the model with the new values
- When you are done, close the Spreadsheet with OK.

With the bearing cleaned up and sized correctly, we can move on to the next step.

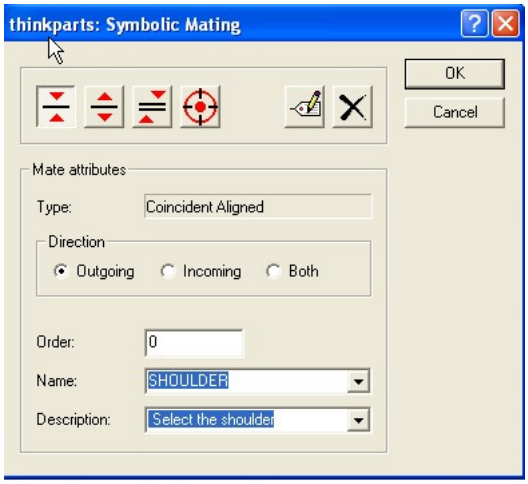
3. Step 3: Symbolic Mating

In this step, we shall add **Symbolic Mating** constraints to the bearing. **Symbolic Mating** allows you to define mating constraints on a thinkparts template file by specifying the type of constraint, the first of the two faces involved in the constraint, and the prompt that the user sees when the part is imported.

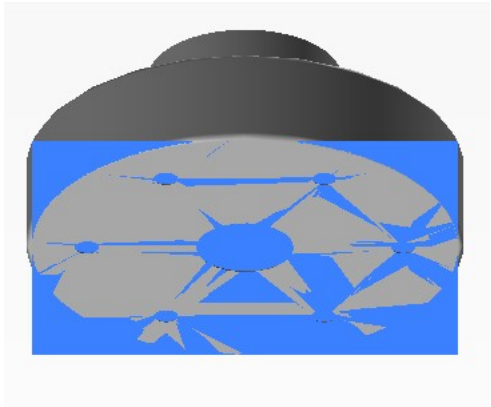


For starters, we shall define **Symbolic Mating** for the back face of the bearing. This will be an aligned coincident constraint for the mounting face of the part. Before you start, you may want to change the color so the **Symbolic Mating** objects come in as a different color than the part.

- Start **Symbolic Mating** from the thinkparts toolbar to display the Symbolic Mating dialog box.
- Select the
  
  Coincident Aligned button and set the Order to 0. For the Name, enter Mounting and for the Description, enter "Select the mounting surface."

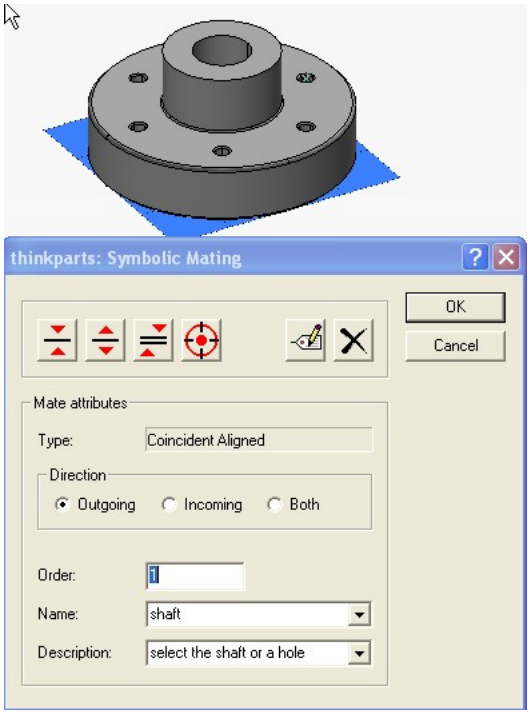


Now select the bottom face of the bearing. A temporary datum plane is inserted, representing the Symbolic Mating. If we need to edit this constraint later, we shall select this plane.



Now add a concentric mating for the shaft/hole.

- Select the
  
  Concentric button
- Enter an Order value of 1.
- Enter SHAFT as the Name.
- Enter "Select the shaft or hole" as the Description.
- Select the inner cylindrical face of the bearing as the first face of the Concentric mating sequence.
- Click OK to end the command set.



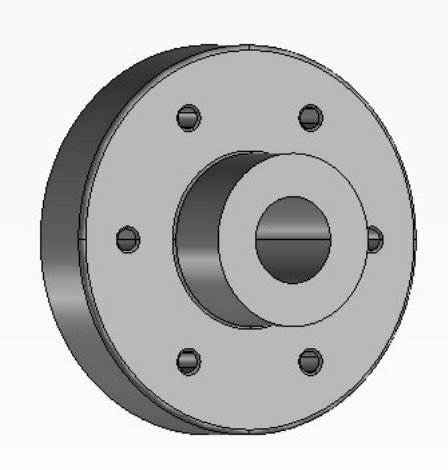
In the next step, we shall save our bearing into the catalog.

4. Step 4: Add it to the catalog

In this step, we shall save the bearing into the catalog as a Part Family, then we shall edit the parts to create multiple standard sizes. With just the standard ThinkDesign license you get the thinkparts run-time version, which gives you the basic functionality needed to create and edit your own part families. For the increased creation, editing and organizational functionality, including exporting your catalogs for others to use, you need the thinkparts **Catalog Manager**.

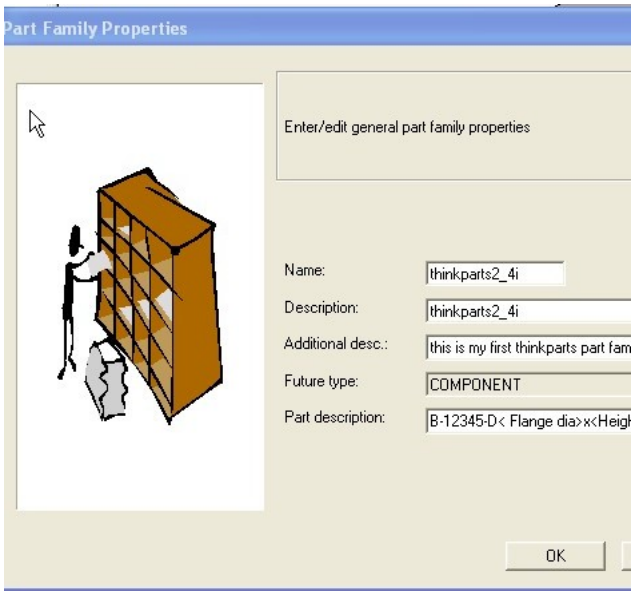
When you save the part into the database, the drawing area is captured to serve as the preview image.

- Clean up the display and orient the model for the image capture.



With the model complete, it's time to save it into the thinkparts database with **Create Part Family**. The Part Family Properties dialog box has multiple fields where you define the general properties of the part family, including the name and description of the part. How you name and describe your parts is up to you, but we shall give you some tips you can use to organize your parts.

- Start the **Create Part Family** command to display the Part Family Properties dialog box. For the part family Name, enter something resembling a part number (See figure, below). This value appears in the List View window of the Catalog Browser when the part family is selected in the Tree View. It is also the Component name when the part is inserted in an assembly.
- For the Description field, enter a brief description of the part. This field is displayed in the Tree View with the Catalog icon.
- The Part Description field is transferred to the description field of the Component when you import the part. You can use the variable names in the template to build a parametric description of the part. For our part, enter B12345 - D<FlangeDia> x <Height> x d<ShaftDia> to include the flange diameter, height and shaft diameter values in the component description.



With the data entered in the proper fields, hit OK to save the part family.

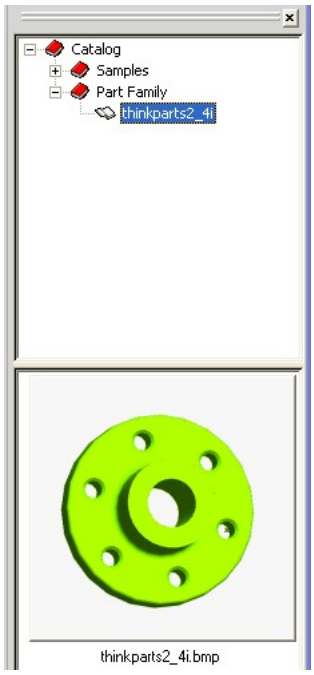
- The display is adjusted and a screen image is captured to serve as a preview image in the catalog.

- The file is saved to the Name you entered in the properties dialog box.
- Once the process is complete, you will get the following message:

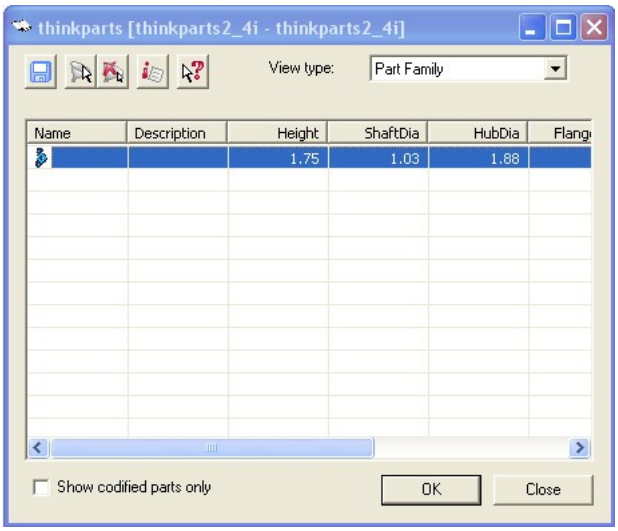
- Click Close.

Now open the **Catalog Browser** to see what we have done.

- Expand the new Part Family Index to see the thinkparts2_4i catalog.
- Select Mythinkparts2_4i to see the preview image.

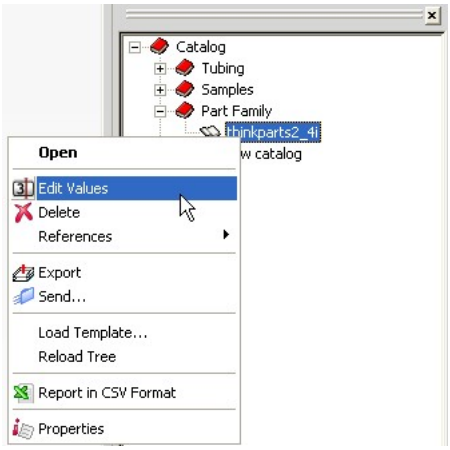


- Double click on the thinkparts2_4i in the list to open the parts window.
- Notice the values in the list.
- Also notice that there is one part in the list, with the variable values equal to the variable values when we created the part.



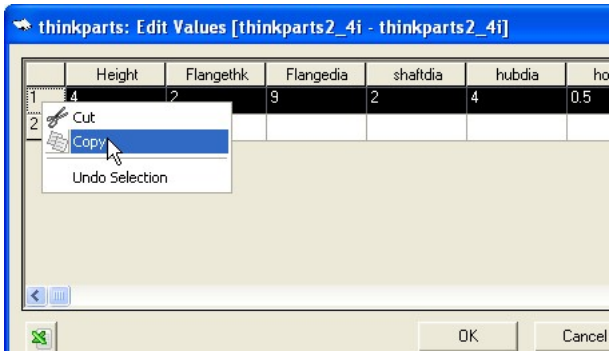
- Close the Parts window.

That's cool and all, but we wanted at least one, and perhaps several, different standard sizes for the bearing. To do this, we need to use Edit Values.

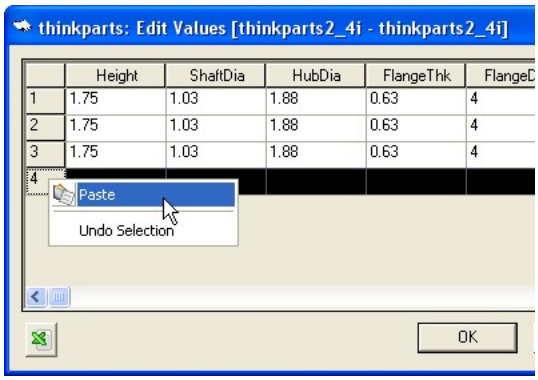


- Right click on thinkparts2_4i in the Tree View and select Edit Values... from the context menu.

- In the Edit components dialog box, select the 1st row in the table, right click on it and hit Copy.

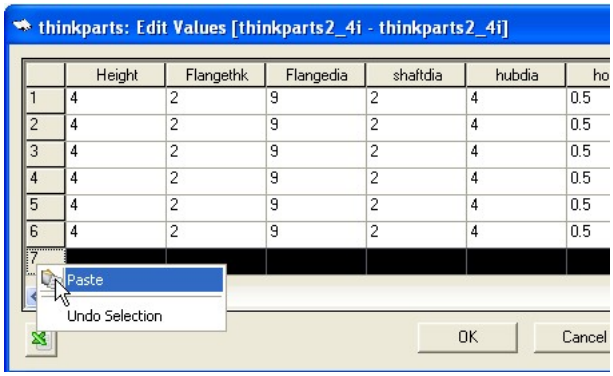


Right click again and hit Paste to paste in a new copy of the first part definition. Paste in at least one, or perhaps several more copies; we shall edit the copies in the next step.



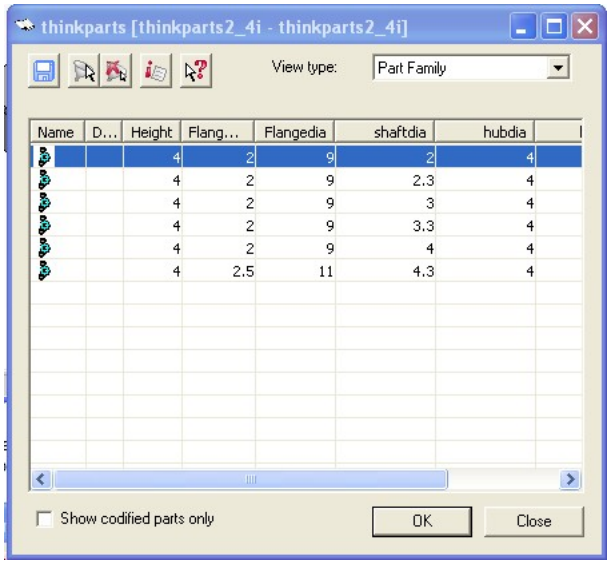
Now edit the value(s) to create a part, or series of parts, with new values for the parameters. If you are just entering one additional part, select the data that has the same ShaftDia as the original part so that you can fit it on our test part correctly in an upcoming step.

- Create as many parts as you like using the values suggested below.



Close the Edit components dialog box with OK to create the new part sizes in the part family.

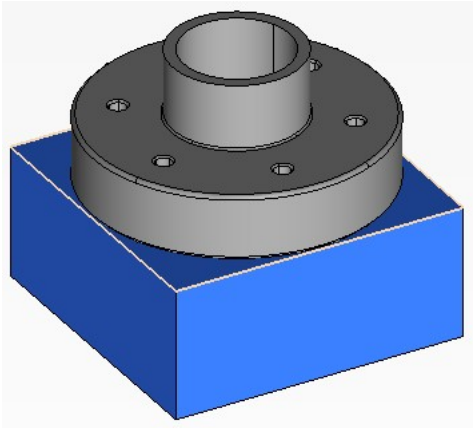
- The new parts are added to the table in the Parts View.
- Note that the number of parts is updated in the Parts View header.



We are done with the template, so close the Model file window. In the next step, we shall test out our new part family.

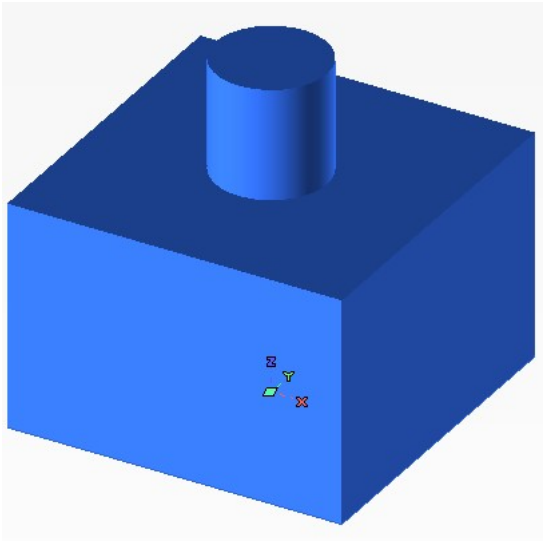
5. Step 5: Insert the new part

In this step, we shall see how we did by inserting one of the bearings into a simple test model.



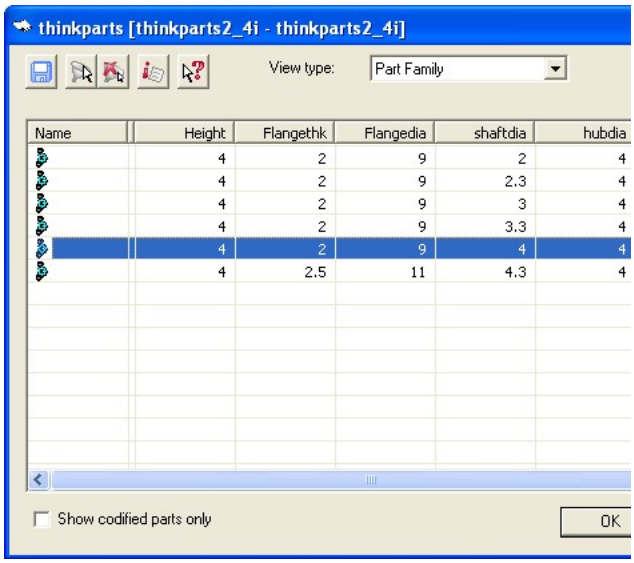
Before we can test our new part family, we need a new model on which to test it.

- Open a **New Model** file and create a simple part.
- A 6 inch by 6 inch block, 4 inches thick with a 2 inch diameter by 2 inch long shaft on top of the block, will do. (See figure) .

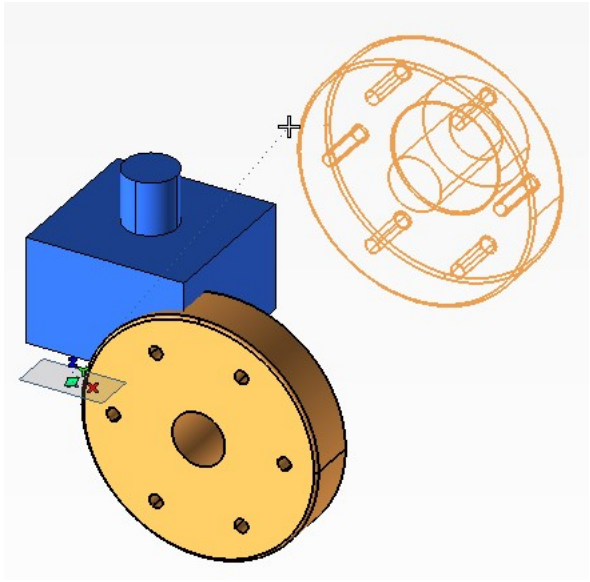


Switch back to the **Catalog Browser** and Import one of our new part sizes.

- Open the Parts window for the Bearing, thinkparts2_4i.
- Select a size that works with the shaft on the test model.

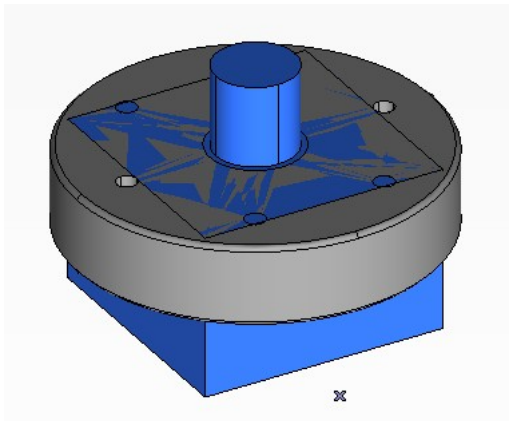


Depending on the part you selected to Import, you should see something like this.



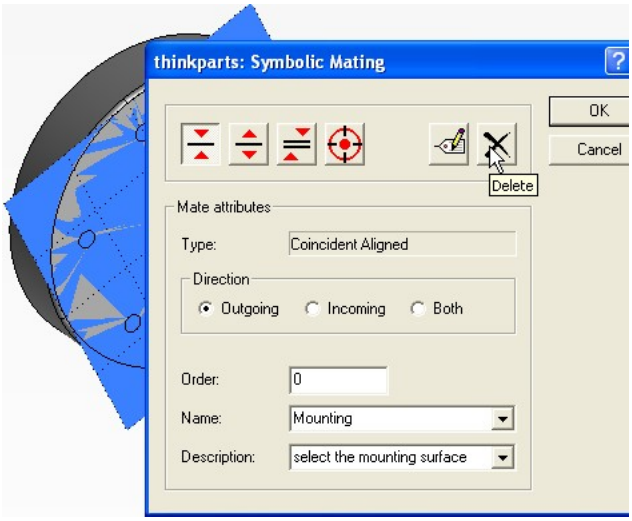
- Select the insertion point, mounting surface and shaft or hole to mate the part to.

D'oh! We added the wrong Symbolic Mating constraint. The mounting surface constraint should have been a Coincident Antialigned constraint.

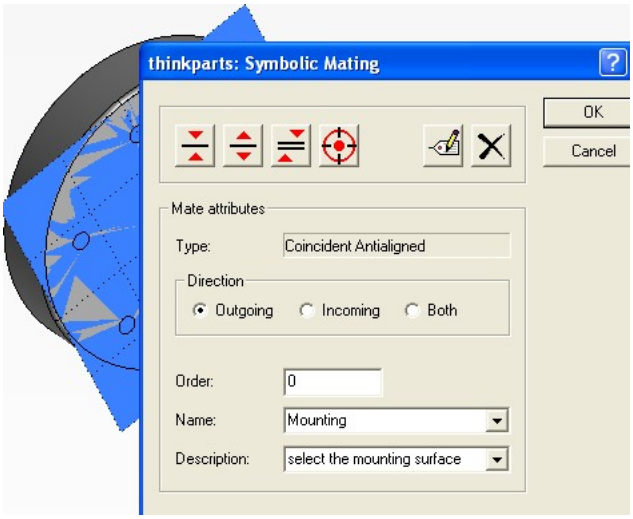


It looks like we need to do some work on the template, but we already closed it. We shall need to load the template from the Catalog Browser and make the edits.

- Activate the Catalog Browser window and right click on thinkparts2_4i in the Tree View. Right click within the Catalog window and select the Load Template option from the context menu to reload the thinkparts2_4i.e3 template.
- Start the **Symbolic Mating** command and hit the **Delete** button. Select the plane symbolizing the mounting constraint.

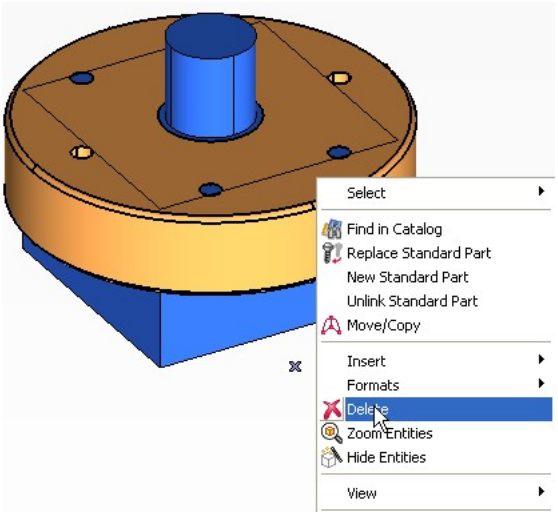


- Now hit the
  
  Coincident Antialigned button and define the correct MOUNTING constraint, selecting the bottom surface to complete the definition.

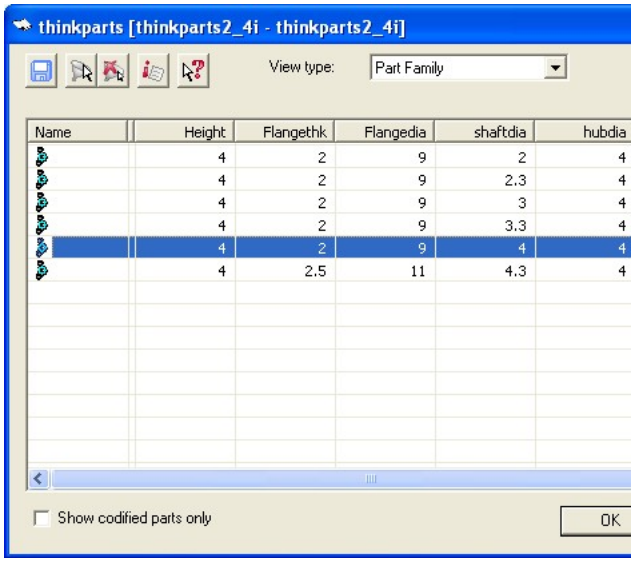


That should do it, and we are ready to try it again.

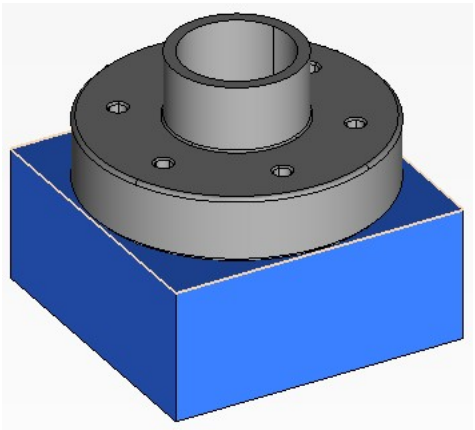
- Close the Symbolic Mating dialog box with OK.
- **Save** and close the template file.
- Activate the test model window and **Delete** the first insertion attempt.



Now go back to the **Catalog Browser** and Import the Bearing.



- Select the insertion point, mounting surface and shaft to mate the parts.
- The new part should now have the correct orientation to the test model. (Change the color of the original part if desired for better visualization.

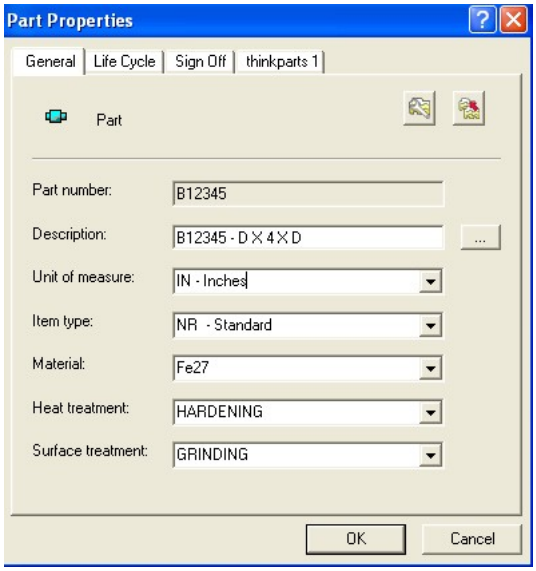


Nice work! This part will behave like the parts in the Samples index you imported in the Introduction to thinkparts task. You can use **New Instance of This**, **Replace Part** and other tools to copy, replace and edit the

bearing. In the last step, we shall look at **Edit Part Data** and part families.

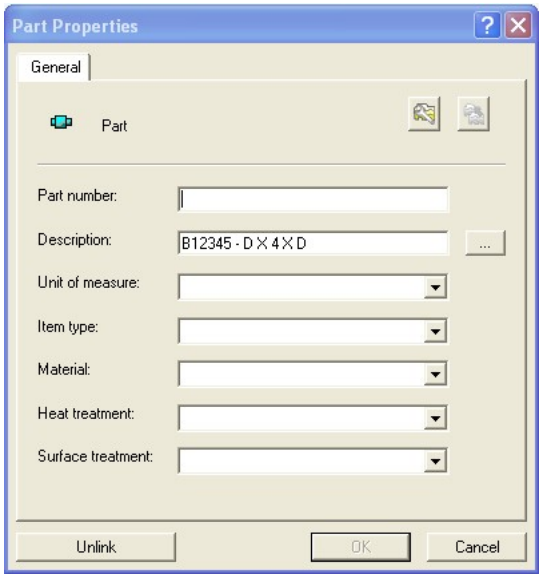
6. Step 6: Partdata and thinkparts

In this last step, we shall learn how to add numbered parts to the part family and enter via use of **Edit Part Data** for the bill of materials. As you know, ThinkDesign stores parts list/bill of materials information in components as Part Data.



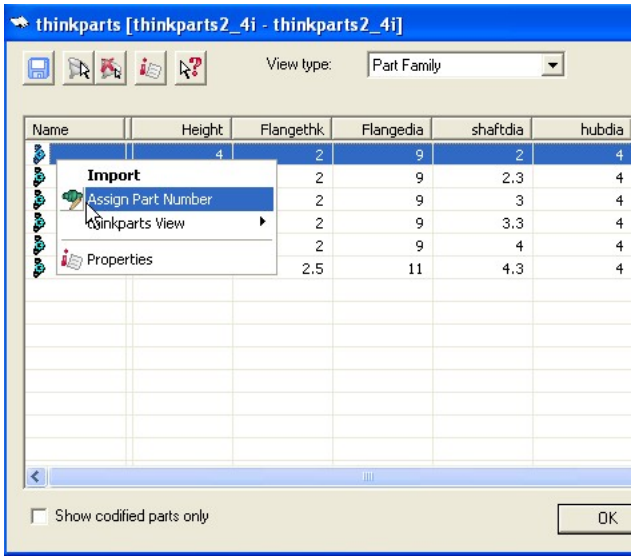
We shall start by looking at the Part Data associated with the bearing we inserted in the test model.

- Right click on the component either in the Graphics area or the History Tree.
- Select **Edit Part Data** from the context menu to launch the Part Properties dialog box.

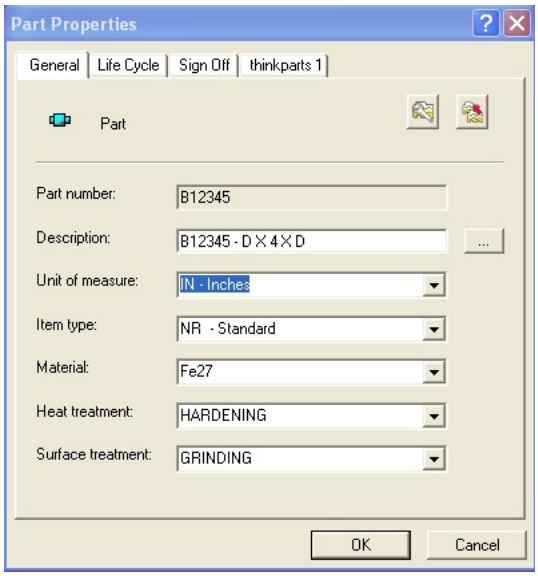


Notice that the only data associated with the component is the Description, which we defined with variable expressions when we created the part family. We could enter the part data here for this part, and depending on the part family, this may be the best approach. But we can also define part numbers and part data within a part family, so the part data will be complete when the part is imported into an assembly. Let's go back to the Catalog Browser and use **Assign Part Number** to add part data to some of the part family members.

- Activate the Catalog Browser window again. Open the Parts window for the thinkparts2_4i catalog. Right click on the first part in the List View and select Assign Part Number from the context menu.

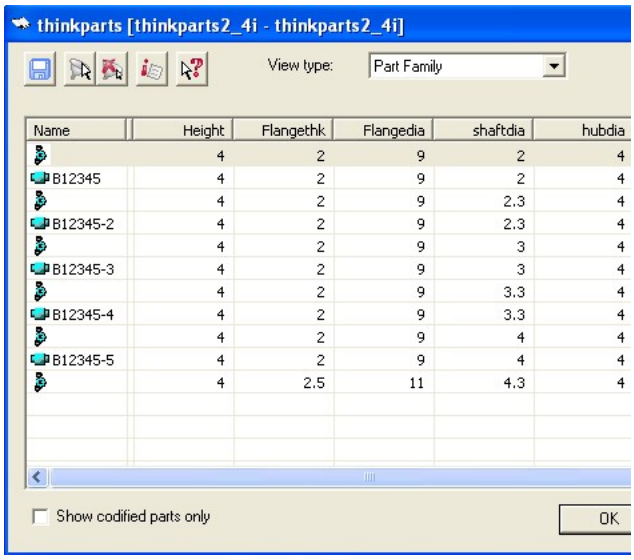


On the General tab of the Standard Part Properties dialog box, enter the appropriate data in the part data fields. You don't have to enter data for all fields.



- Close the dialog box with OK to apply the data. Notice that the part data is assigned to a new part with the same size values, but the new part has a different icon and the part number and description fields are filled out.

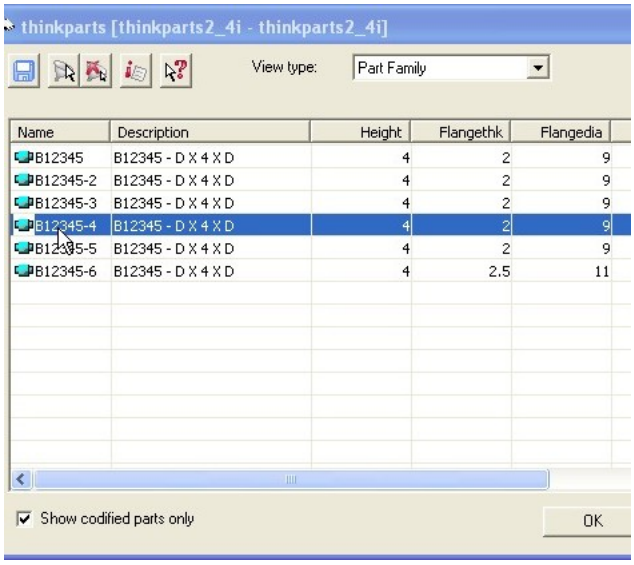
- Repeat the process to assign part numbers to several part sizes.



- Close the Parts window.

Now go back to the test model and replace the existing part with one that is numbered.

- Activate the test model window, right click on the existing bearing component and select **Replace Part** from the context menu.
- A new Parts window is opened. Click Show codified parts only. Now, only numbered parts are visible.
- Click on the ShaftDia column heading to sort the list by the shaft diameter. Finally, select a numbered part as a replacement. Once it's replaced, verify the component part data in the test model.



So there you have it. Go forth and populate your thinkparts database. If you need more tools to manage the database structure, or you would like to share your catalogs with others, look into the thinkparts Manager. There are also catalogs available from think3 and third party vendors.

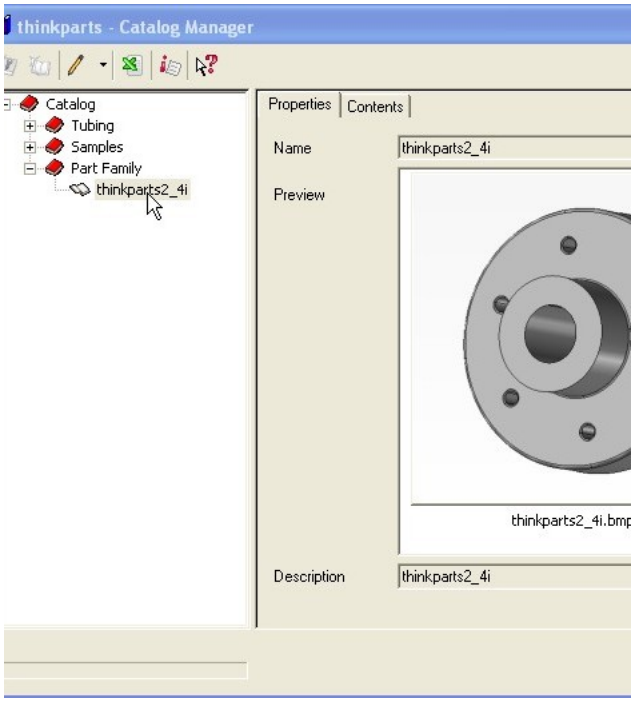
7. Step7 : Export/Import Part numbers

Suppose there is a need to re-use Part numbers of components from an old project in the current one, What do you do? It is now possible to Export/Import Part numbers. There is also the possibility to Copy catalogs from

one Part catalog to another. Lets explore these functionalities.

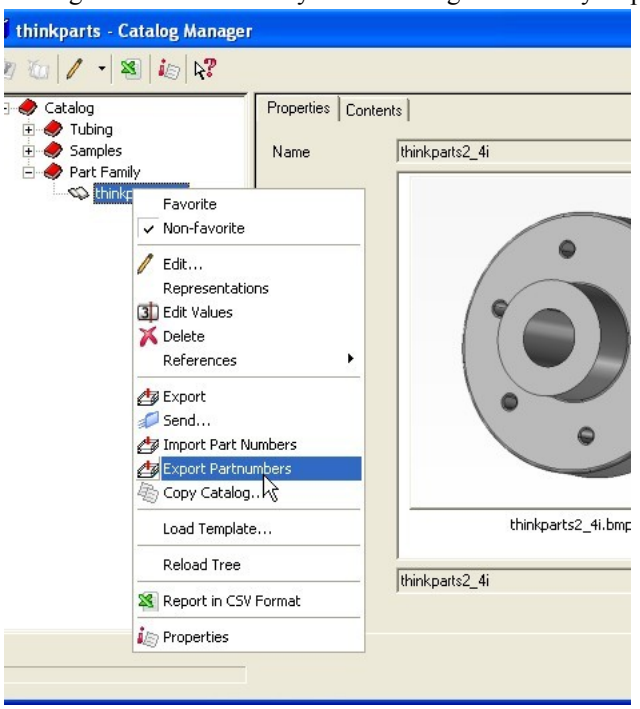
You have the thinkparts2_4i family that you created. We will try to export the part numbers of components in this part family and import them.

- Open the thinkparts Catalog Manager and check the presence of thinkparts2_4i part family.



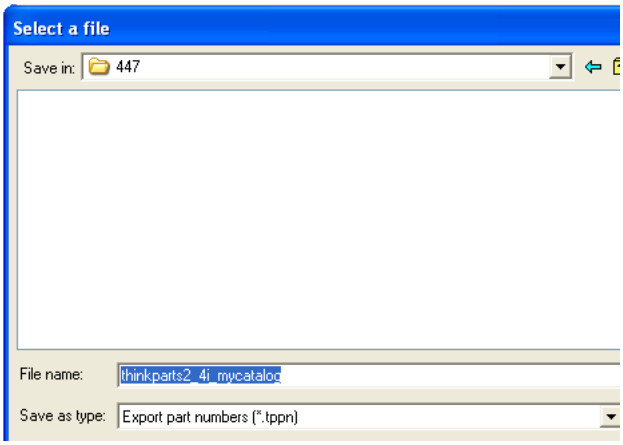
Let's now Export the part numbers from this family.

- Right click on the family in the catalog tree and say Export Partnumbers .

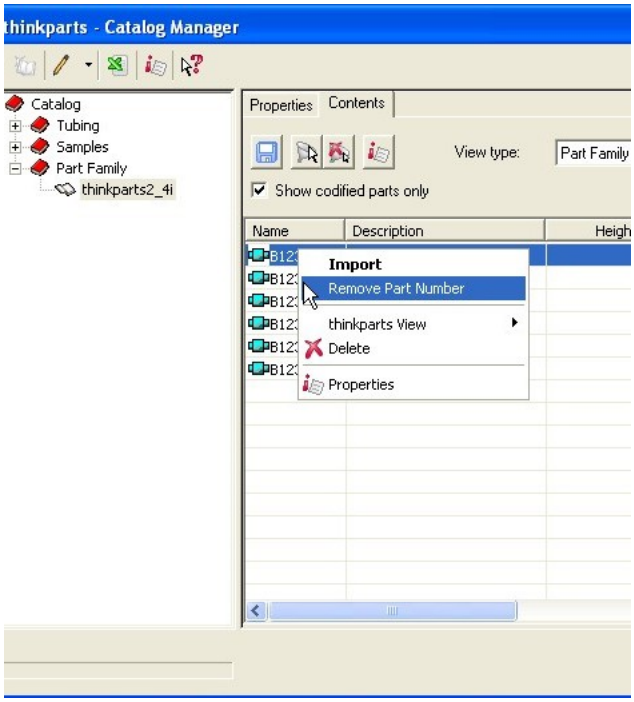


The Save file dialog comes up.

- Give an appropriate name as shown and save the file with .tppn extension.



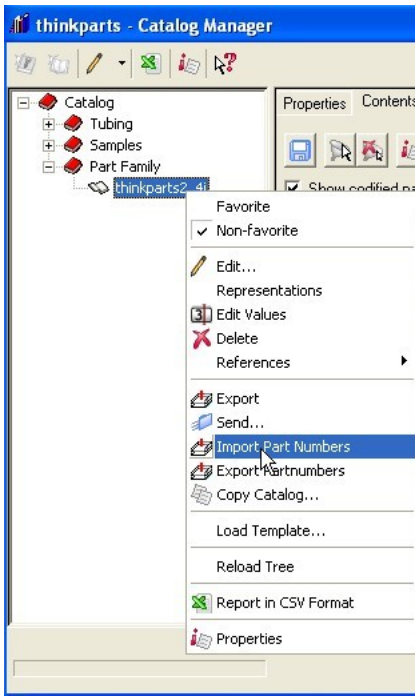
- In the Catalog Manager, Click on Contents tab.
- Check Show codified parts only to view only the components that have part numbers.
- Right Click the first part and say Remove Part Number.



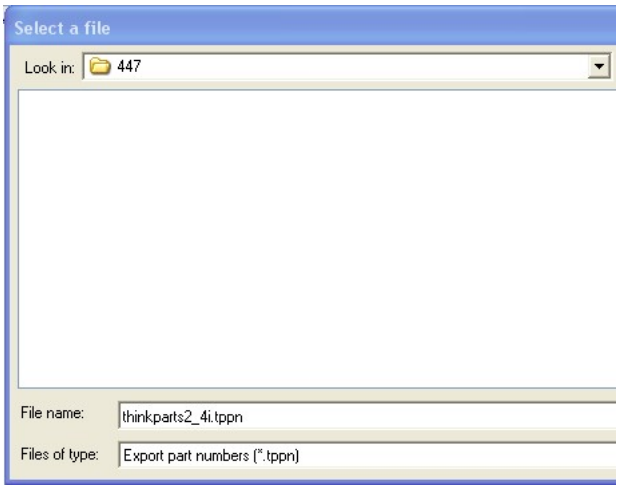
Repeat this action for the other two parts also.

Now that the Part numbers have been removed, lets Import a set of Part numbers using Import Partnumbers command.

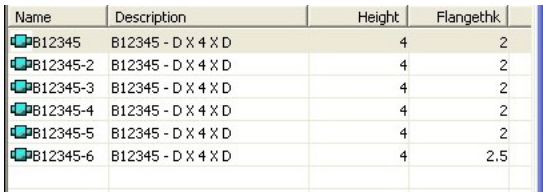
- Right click on the family in the catalog tree and say Import Partnumbers .



- In the file selection box, browse to select the Exported part numbers file and say Open.



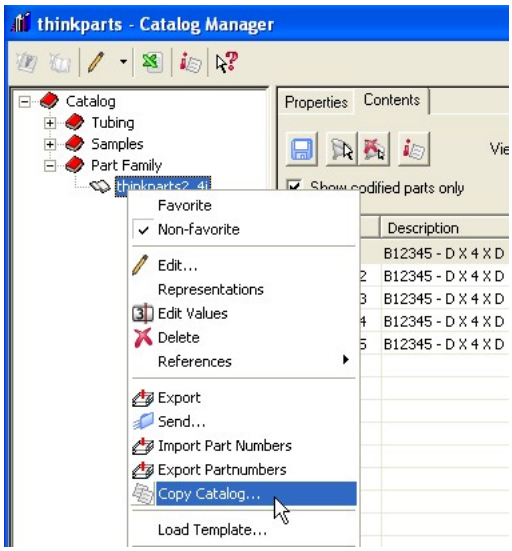
You can see that the Part numbers are again seen in the catalog manager.



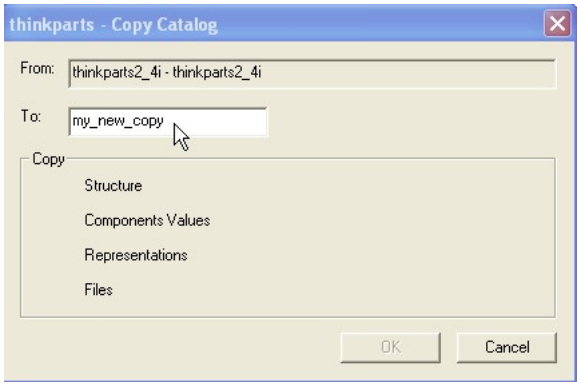
Name	Description	Height	Flangethk
B12345	B12345 - D X 4 X D	4	2
B12345-2	B12345 - D X 4 X D	4	2
B12345-3	B12345 - D X 4 X D	4	2
B12345-4	B12345 - D X 4 X D	4	2
B12345-5	B12345 - D X 4 X D	4	2
B12345-6	B12345 - D X 4 X D	4	2.5

We will now copy the existing Part family.

- Right the family in the catalog tree and say Copy catalog.



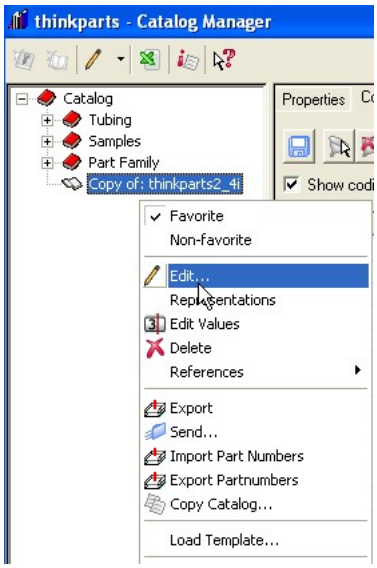
- In the Copy catalog dialog that comes up, key in the new name of the catalog.



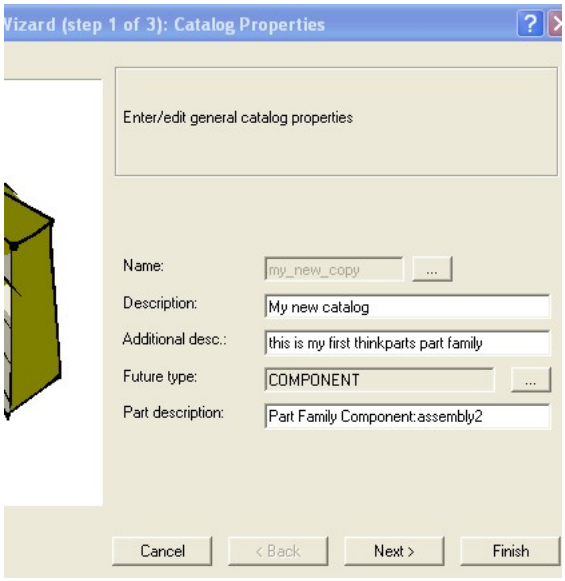
- Then hit the Tab key to enable the OK button.
- Click OK button to make a copy of the catalog.

You will see the new catalog in the catalog tree.

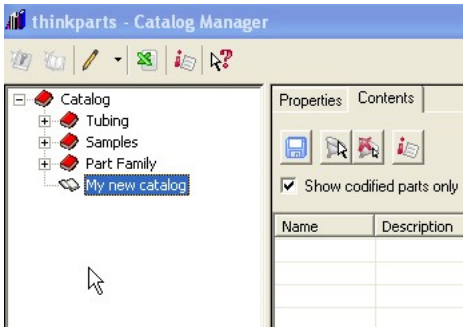
- Right click on the new copied family in the tree and say Edit to open the thinkparts Catalog wizard.



- Change the Description as shown and say Finish.



You can see the name being changed in the catalog tree also.



You can now Import any of the parts in this new family into ThinkDesign environment. This step ends this task. Now you are familiar with the utility of being able to Export/Import part numbers as well as copy catalog which help you share ..../../Common information across multiple projects and also avoid repetitive work of creating similar part families.

A.

- Introduction to Solid Modeling.
- Project Browser 2.