



**Autodesk Inventor Tutorials**

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# **Introduction to iParts**

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The purpose of this document is to introduce Autodesk Inventor users to iParts. This tutorial was written with R5 in mind but many of the same principles apply to R4. The first part of this tutorial describes the method to make a simple iPart and includes concepts such as filenames, properties, feature suppression and key selection. Future lessons will describe the process for threads and material and color changes.

As with many modeling programs there are countless ways to design and model parts. This is one way. It may not be the most effective way to design the part but it was constructed this way to facilitate this tutorial. As your experience grows you will find ways to shortcut some of the steps that I present here. I encourage you to explore different ways of dimensioning, constraining and developing sketches.

In this lesson we will be focusing on a standard linear bearing that is made by many manufacturers such as NB, IKO and IGUS. They all have the same basic size and shape and the part is fairly simple. There are four basic types of linear bearing that include standard, adjustable, open and double. Below are some photos of each type. (see Figure 1) Bearing can be found at [Charles Bliss' site](#).

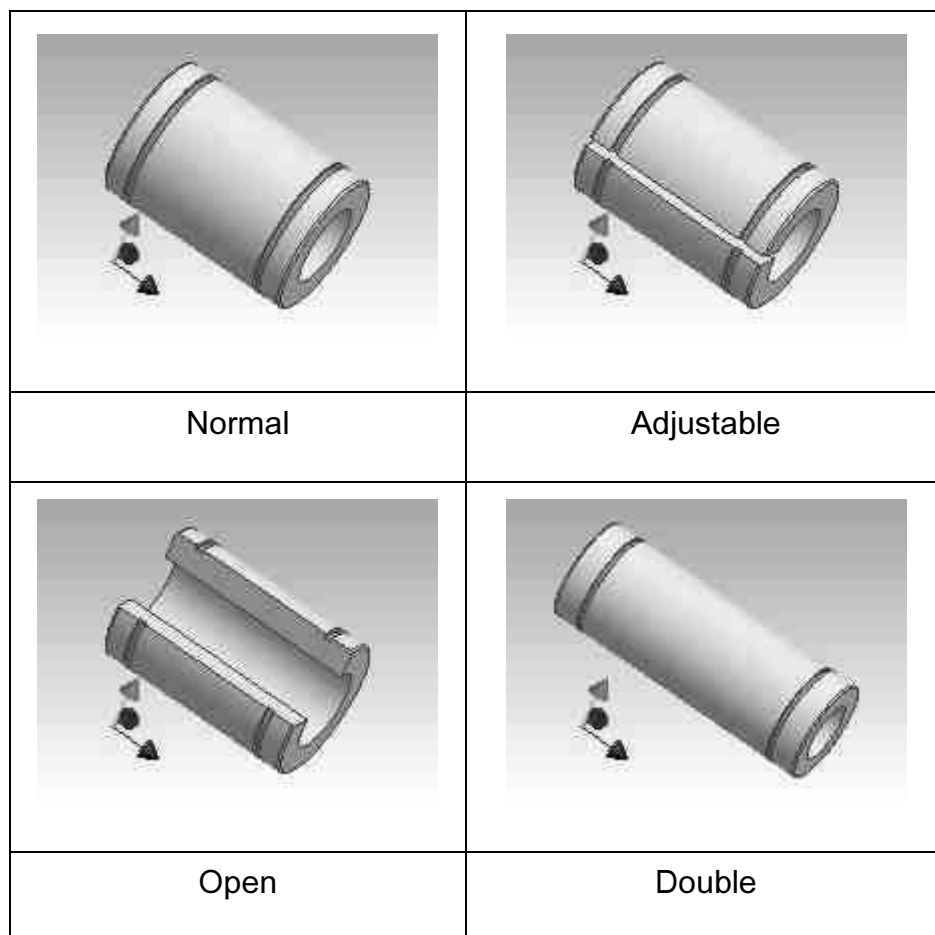


Figure 1 - Types of Bearings

To begin any iPart start a new IV drawing in an english or metric template.

We begin the part by making the base sketch. It is two concentric circles. We arbitrarily give them a diameter for now. We then extrude the hollow cylinder to a length of 1.25". Again this is arbitrary, as we will change this later. (see Figure 2)

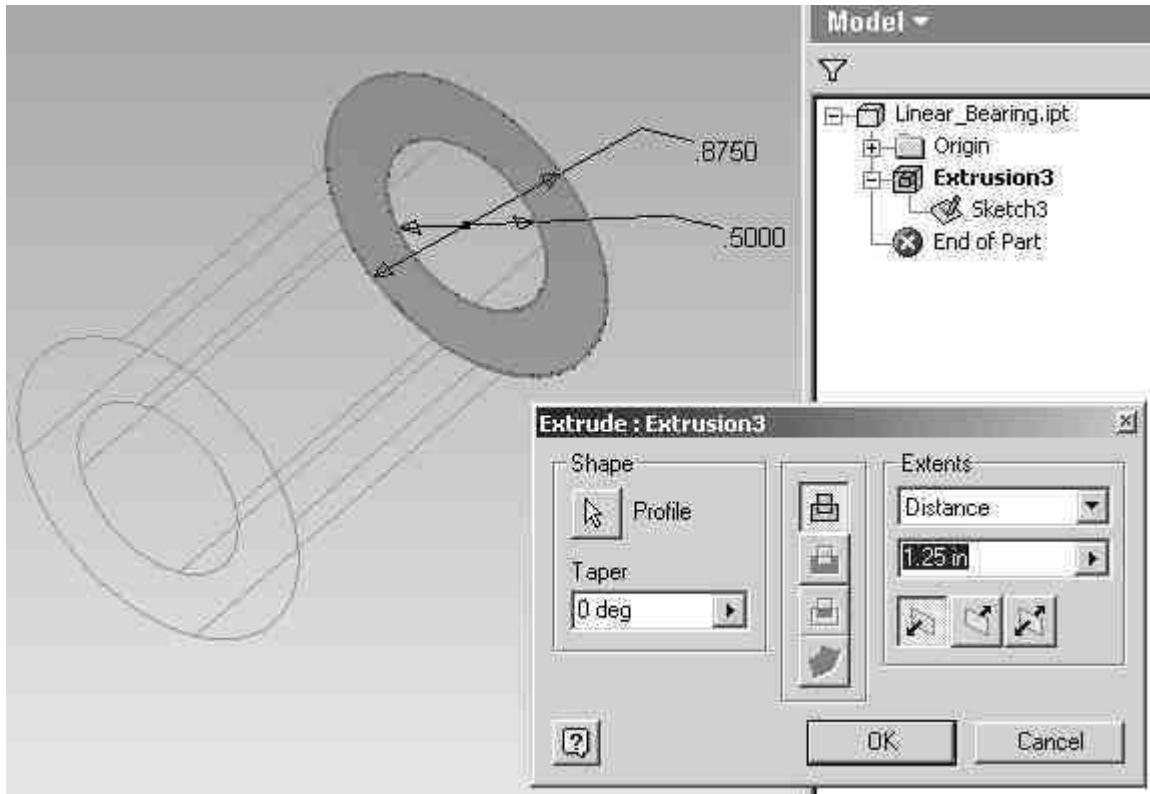


Figure 2 - First Sketch

We now can start assigning parameter names to the iPart. Click the parameters button at the top of the screen. This will open the parameters window. (see Figure 3)

You see that each dimension is given a name such as d6, d7, d8...

**IMPORTANT NOTE: Depending on how you draw the sketch your unnamed dimension designators may be different e.g. they may be d1 instead of d6 etc.. Inventor gives sequential #s to each dimension. If you place a dimension and then delete it the next dimension will have a value one higher than the last dimension. Therefore you could have different dimension names than in this example. This is not a problem.**

While you can use these values it is easier to change them to something that makes more sense. Also changing them will allow them to be automatically imported into the iPart

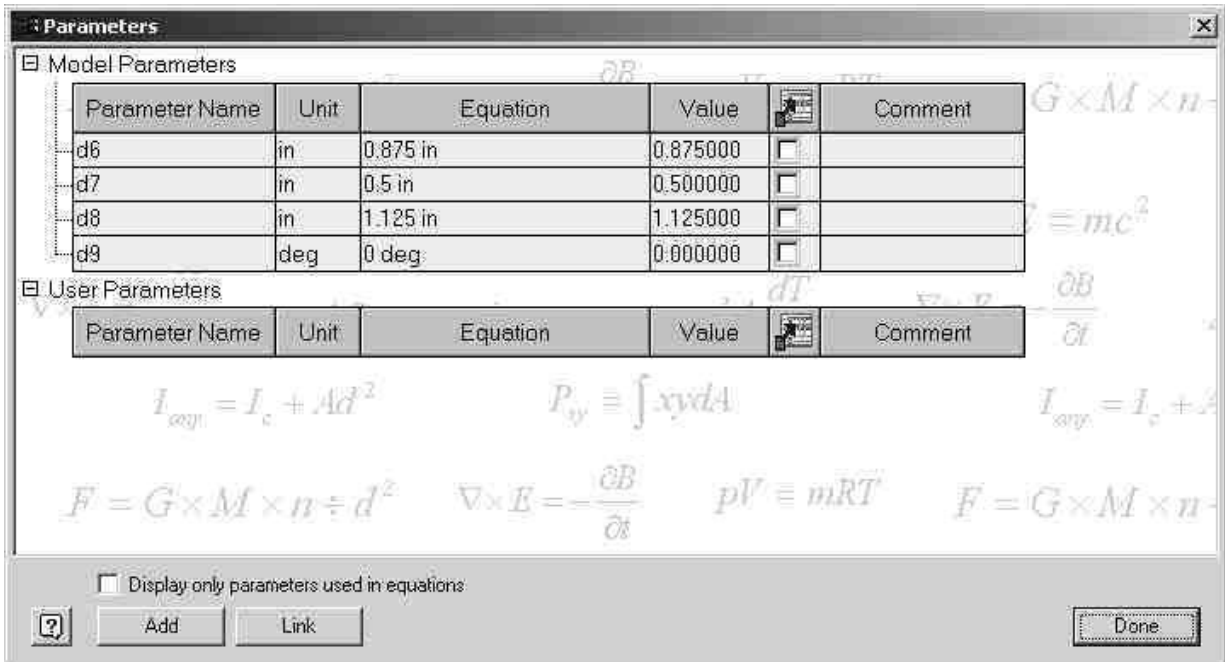


Figure 3 - Parameters Dialogue

table in later steps. We will rename d6 to OD, d7 to Bore and d8 to length. (see Figure 4)

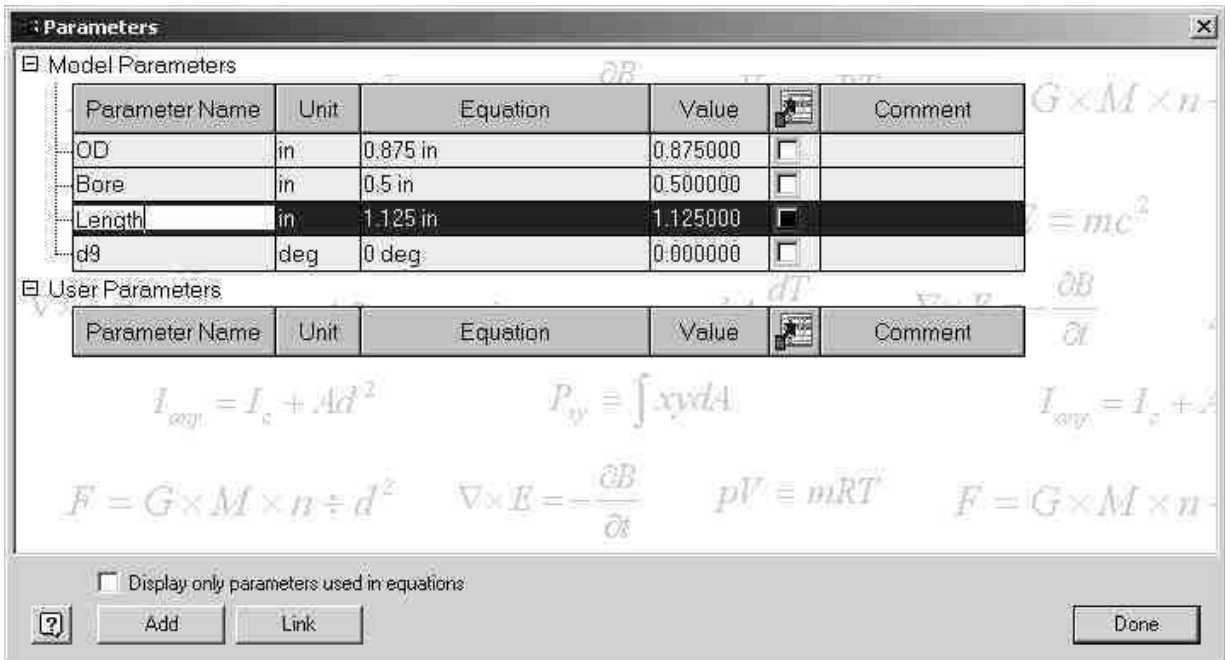
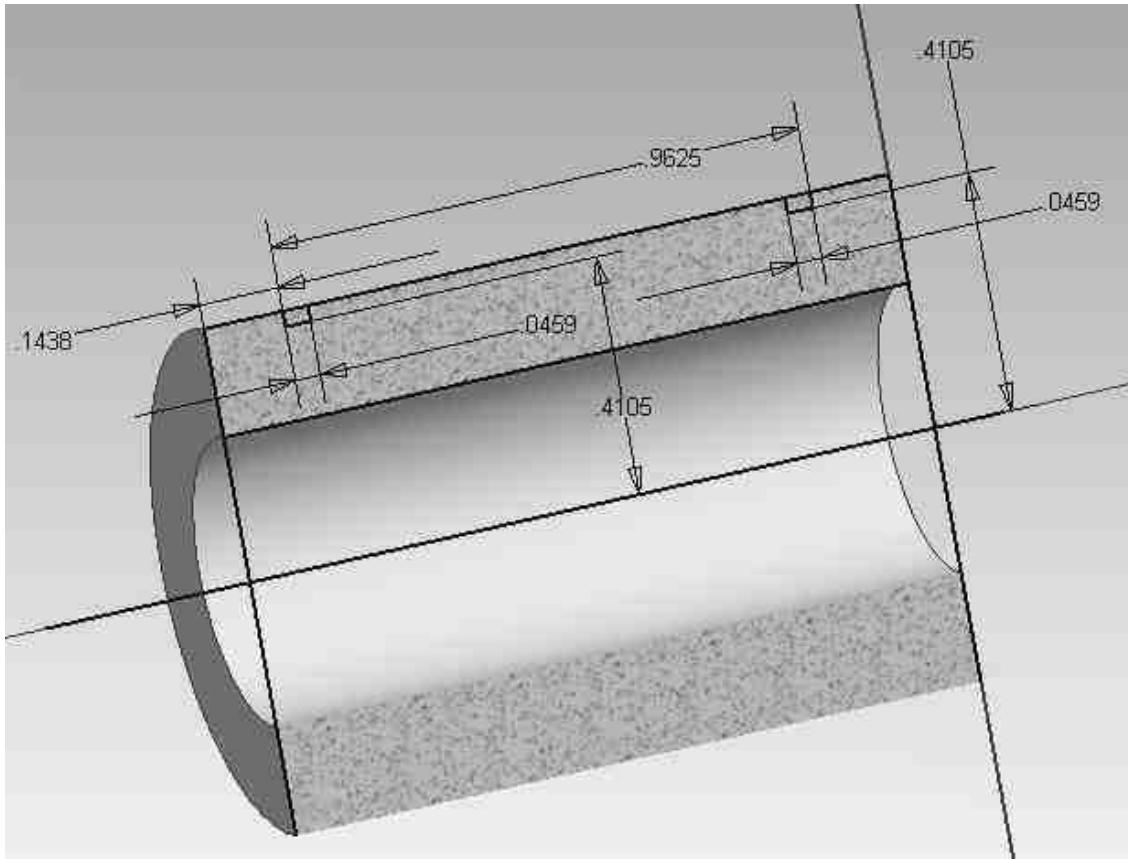


Figure 4 - Parameters Named Changed

We do not bother to rename d9 (the extrusion taper angle) as this value will not be changed. Since we have now named the dimensions you can refer to them in other dimensions. For example you can make another feature be equal to 2\*Length or

2\*Bore/Length etc...(you could always do this with d6, d7 but it's easier to remember them this way.) Exit out of the parameters window.

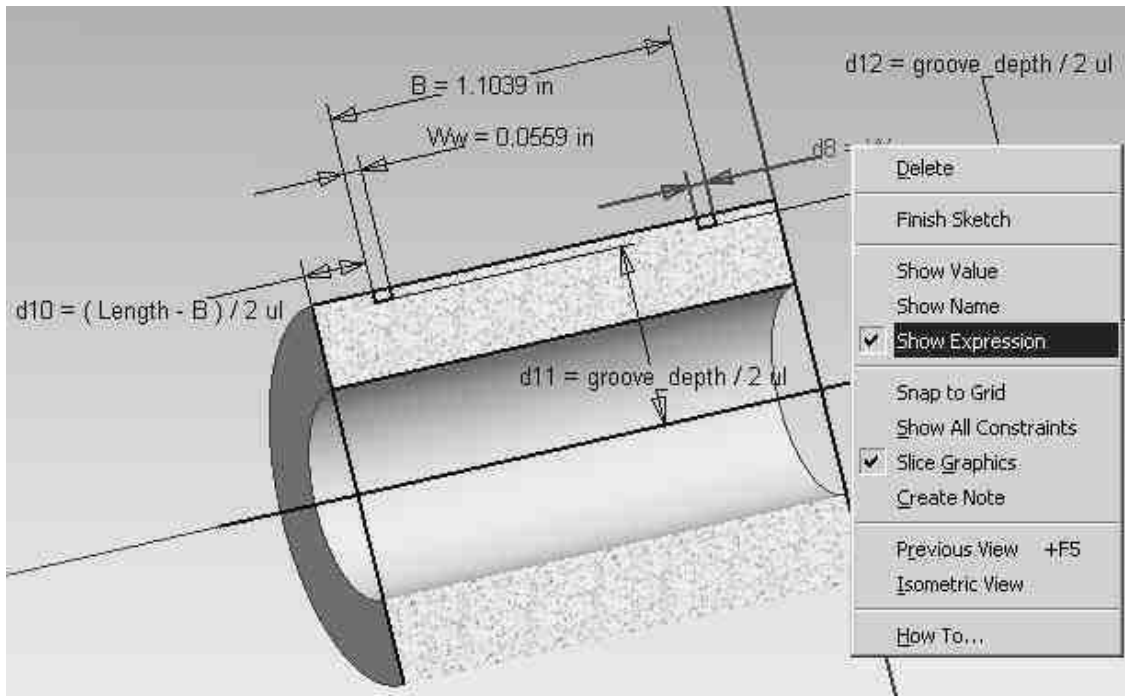
We now look at creating the snap ring groove around the outside of the part. Create a new sketch on the axis of the part. If you drew the circles centers around the origin you can use one of the origin planes. Create a sketch like the one shown below and (cut) revolve it around the part:. (see Figure 5)



**Figure 5 - Revolved Cut Sketch**

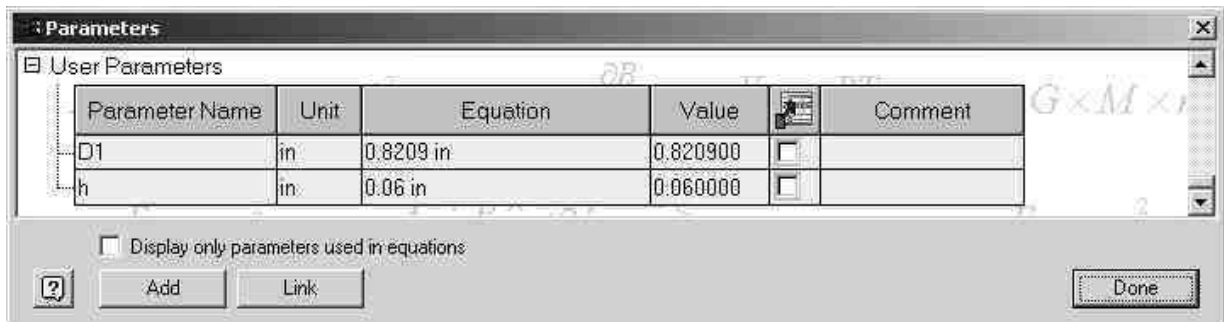
Again, the dimension values are not important at this time. You might want to try to dimension the part correctly to one particular size of bearing so that my relationship is pretty close to correct. Some thought needs to be given to how the part is dimensioned to make it easy to fill out the data tables. For example in this part the distance between the bottom of the grooves was provided by the manufacturer, therefore this was dimensioned as opposed to the depth of the groove from the surface. (the 0.4105" dim)

Remember that you can use previously defined parameters to dimension a part. In the picture below I have chosen to define the distance from the ends of the part by  $d10 = (\text{Length} - B) / 2$  to center the rings. We have not yet defined B so you cannot do this yet but we will get there soon...



**Figure 6 - Expressions Shown**

So before I can define  $d10 = \text{groove\_depth} / 2$  I need to define groove\_depth. Go back to the parameter table and scroll to the bottom. Here you can click add and add a user parameter. Name the parameter groove\_depth and set it's value to something (again I like to use the actual values for a certain size part, in our case set it to 0.8209")



**Figure 7 - Adding User Parameters**

We can now go back up to the top part of the table and rename the rest of the values. Notice that d7 is renamed to Ww and we set d8 equal to Ww (this could have also been down in the sketch editor). We rename d9 to B. We leave the rest of the parameters alone as they will be driven off other parameters. We finally define d11 and  $d12 = \text{groove\_depth} / 2$  as well as setting  $d10 = (\text{Length} - B) / 2$ . Whether you set the parameters first and then change the values or vice versa is up to you.

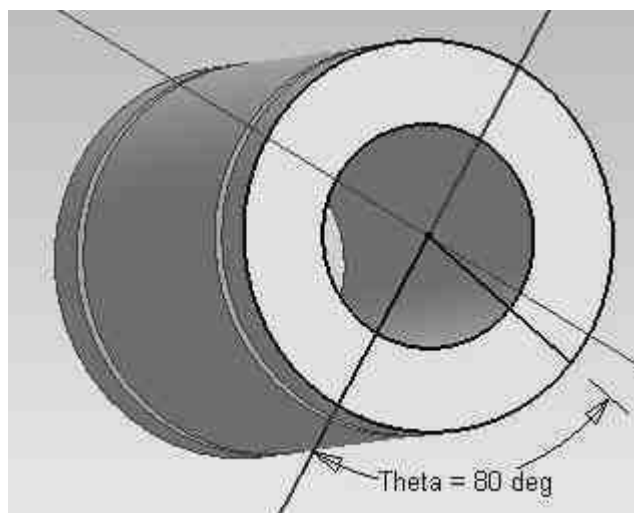
Parameter Name	Unit	Equation	Value		Comment
Bore	in	0.5 in	0.500000	<input type="checkbox"/>	
OD	in	0.875 in	0.875000	<input type="checkbox"/>	
Length	in	1.25 in	1.250000	<input type="checkbox"/>	
d6	deg	0 deg	0.000000	<input type="checkbox"/>	
Ww	in	0.0459 in	0.045900	<input type="checkbox"/>	
d8	in	Ww	0.045900	<input type="checkbox"/>	
B	in	0.9625 in	0.962500	<input type="checkbox"/>	
d10	in	(Length - B) / 2 ul	0.143750	<input type="checkbox"/>	
d11	in	groove_depth / 2 ul	0.410450	<input type="checkbox"/>	
d12	in	groove_depth / 2 ul	0.410450	<input type="checkbox"/>	

Display only parameters used in equations

Buttons: Add, Link, Done

**Figure 8 - Parameters Changed**

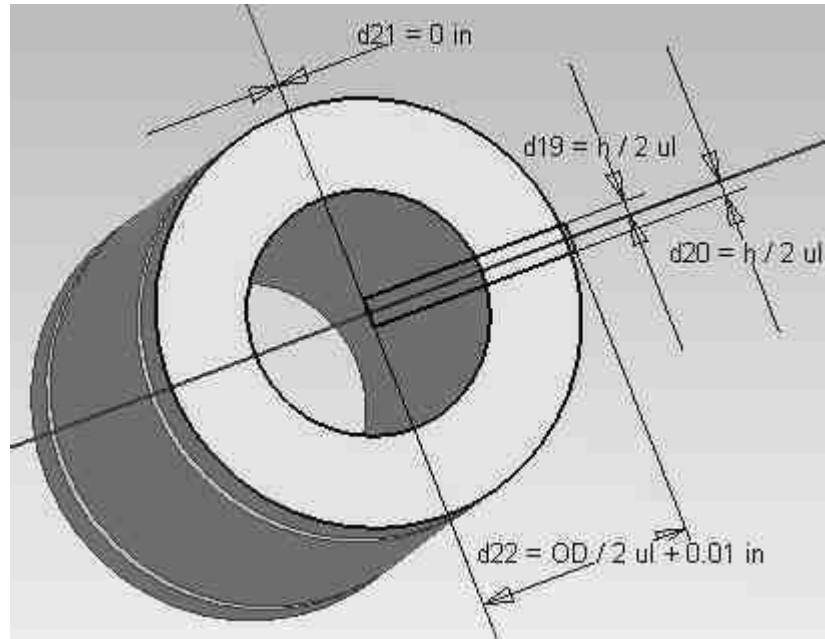
You should now have a part with two snap-ring grooves running around it. The next step is to make the cuts for both the open and the adjustable type bearings. Create a sketch on one end of the bearing as shown below and (cut) extrude it the length of the part. Use the parameter **Length** in the extrusion distance box so that the cut will change with the length of the part. Again set the dim values to a standard size. You can then enter the parameters window and rename these values to something easy to remember. In our example we are using **Theta** to define the angle and setting the diameter of the pie shaped cut to be coincident with the OD of the part (in the sketch mode). This way the diameter of the cut will change with the OD.



**Figure 9 - Definition of Theta**

Now for the rectangular cut for the adjustable type of bearing. Since it runs through the cut for the open type, suppress the extrusion for this cut. You now have a clean face to

work on. Make a sketch like the one below and perform the cut. Again use the parameter **Length** in the extrusion distance box. Note that we used a parameter named **h**. This was added in the user parameters before dimensioned the sketch. We did this due to the fact that the manufacturer gave us the width of the cut and we wanted to dimension the cut by each half. You could have just as easily defined the width of the entire cut and then changed this parameter name to be **h**. The preference is yours.



**Figure 10 - Sketch for Adjustable Cut**

You should now have a completed part (you can add the fillets at the end at any time. They do not change size with the bearing, as they are purely aesthetic). (see Figure 11).

Your parameter table should like something like this. Again remember that your dimension #s may be different than mine. (see Figure 12).

You are now ready to begin the iPart creation process. Click the iPart Author button in the toolbar. (Next to the Parameters icon.)

You will be presented with the iPart author table (see below). It is in this table that you decide what features you want to control. In the table you are presented in the left hand pane with all of the features and their associated parameters. In the right hand pane you have the parameters you want to control.

**IMPORTANT NOTE: Depending on how you draw the sketch your unnamed dimension designators may be different eg. they may be d1 instead of d6 etc.. Inventor gives sequential #s to each dimension. If you place a dimension and then delete it the the next dimension will have a value one higher than the last dimension.**



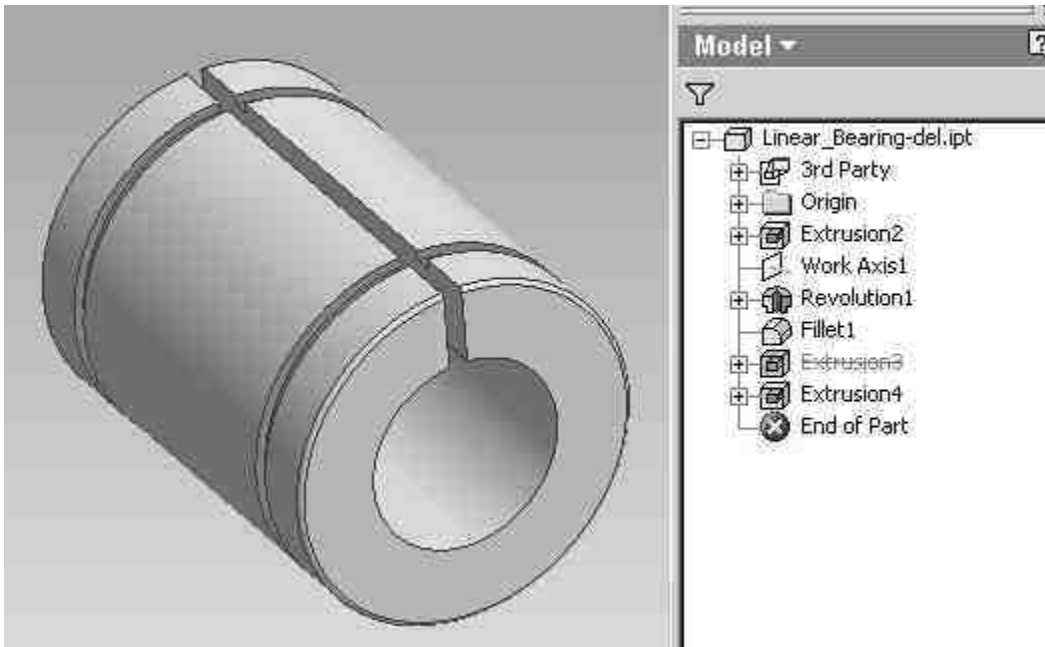


Figure 11 - Completed Part

Parameters

Model Parameters

Parameter Name	Unit	Equation	Value		Comment
Bore	in	0.5 in	0.500000	<input type="checkbox"/>	
OD	in	0.875 in	0.875000	<input type="checkbox"/>	
Length	in	1.25 in	1.250000	<input type="checkbox"/>	
d6	deg	0 deg	0.000000	<input type="checkbox"/>	
Ww	in	0.0459 in	0.045900	<input type="checkbox"/>	
d8	in	Ww	0.045900	<input type="checkbox"/>	
B	in	0.9625 in	0.962500	<input type="checkbox"/>	
d10	in	(Length - B) / 2 ul	0.143750	<input type="checkbox"/>	
d11	in	groove_depth / 2 ul	0.410450	<input type="checkbox"/>	
d12	in	groove_depth / 2 ul	0.410450	<input type="checkbox"/>	
Theta	deg	80 deg	80.000000	<input type="checkbox"/>	
d18	deg	0 deg	0.000000	<input type="checkbox"/>	
d19	in	h / 2 ul	0.030000	<input type="checkbox"/>	
d20	in	h / 2 ul	0.030000	<input type="checkbox"/>	
d21	in	0 in	0.000000	<input type="checkbox"/>	
d22	in	OD / 2 ul + 0.01 in	0.447500	<input type="checkbox"/>	
d24	deg	0 deg	0.000000	<input type="checkbox"/>	

User Parameters

Parameter Name	Unit	Equation	Value		Comment
groove_depth	in	0.8209 in	0.820900	<input type="checkbox"/>	
h	in	0.06 in	0.060000	<input type="checkbox"/>	

Display only parameters used in equations

Figure 12 - Completed Parameters Box

Therefore you could have different dimension names than in this example. This is not a problem.

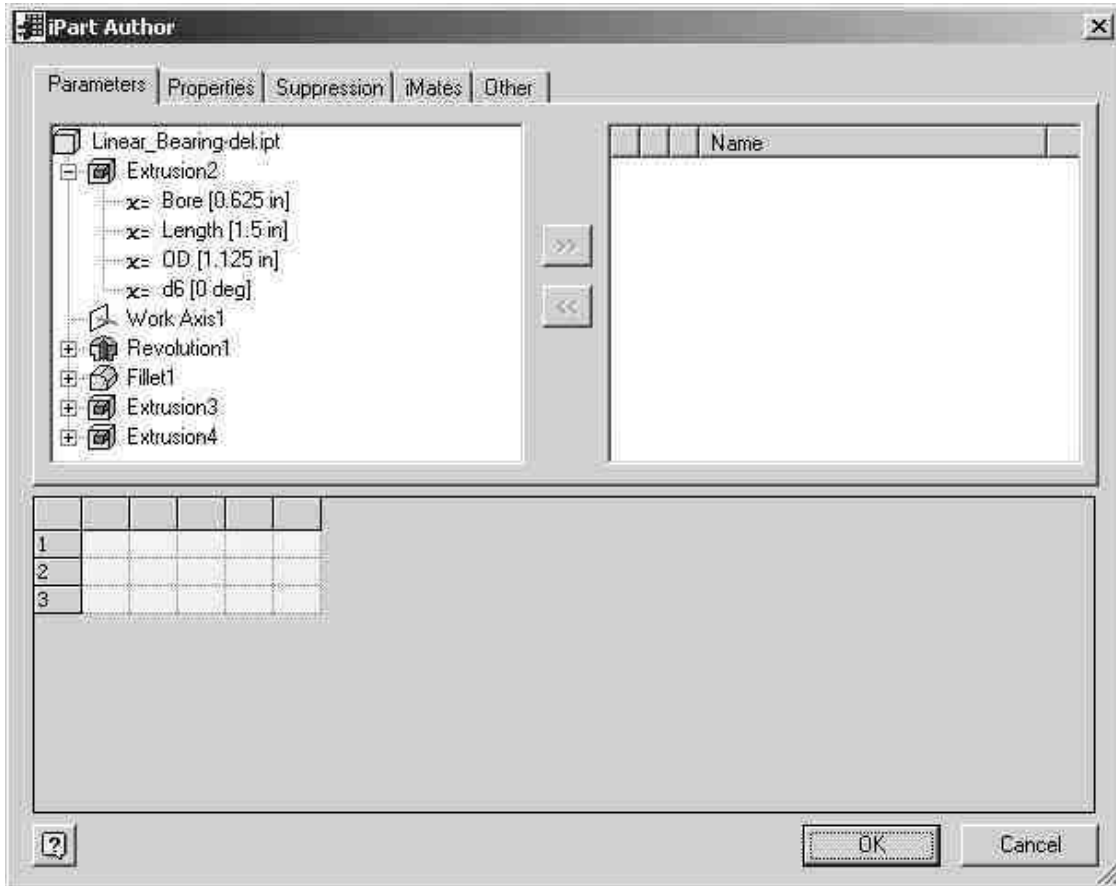


Figure 13 - iPart Author Dialogue Box

**IMPORTANT NOTE:** When you open the iParts Author all of the renamed parameters (such as *Bore*, *Length*, *OD*, *groove\_depth* etc.) should automatically transfer over to the right hand pane. If this does not occur (or if you are editing a older part) follow the process below to transfer parameters into the table.

Click on Bore and then the right facing arrow. This will transfer the parameter into the table. You can delete the parameter from the table by highlighting it in the right hand pane and then clicking on the left facing arrow. Transfer Bore, Length, OD, B, h, groove\_depth, Ww, and Theta into the table. (ignore the fact that your table does not have multiple values in the lower pane). (see Figure 14)

Next click on the properties tab. This is optional but it allows you to set the properties of each iPart child. In our example I want to define the Part Number and Title. Again highlight these choices and transfer them into the table. (see Figure 15)

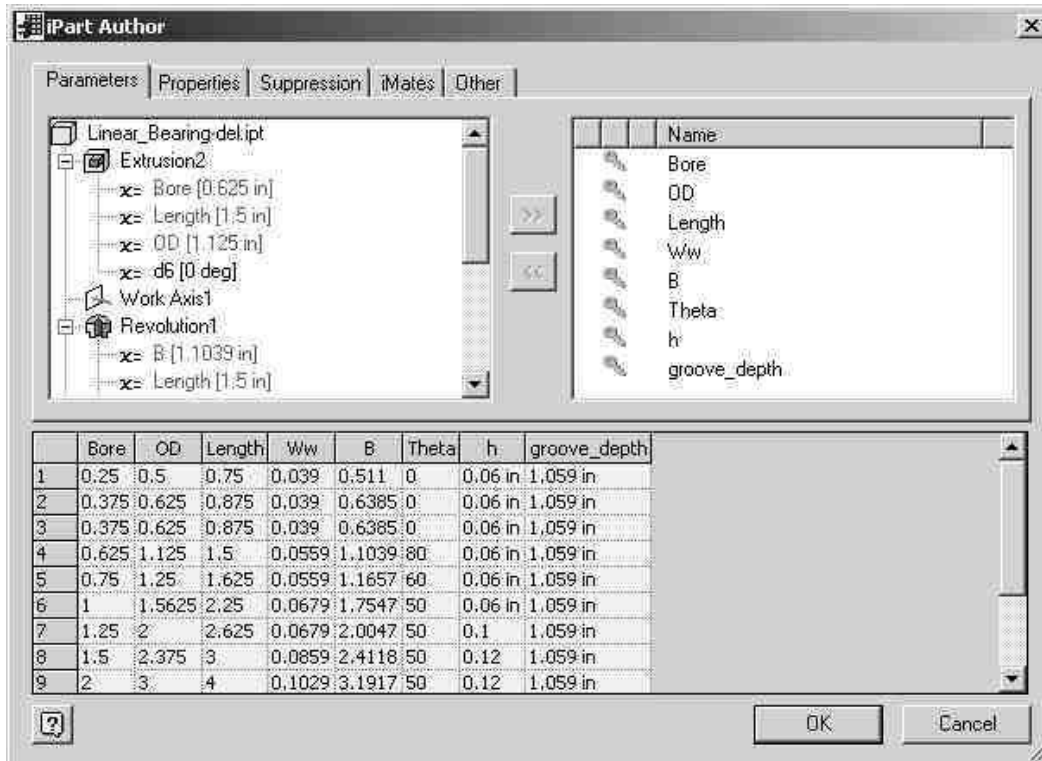


Figure 14 - Parameters Selected in iPart Author

The next tab is suppression. This is how we can have the normal, adjustable and open bearings all in one file. By suppressing the extrusion that makes the cuts for the open and adjustable bearings I can have a normal bearing. By suppressing the adjustable cut and computing the open cut I can have an open bearing. Transfer Extrusion 3 and Extrusion 4 into the table. (see figure 16)

Since this part has no iMates we will be skipping this section. Finally go to the Other tab. This is where you can set custom keys, thread sizes, thread types, filenames, materials and colors. In this lesson we will focus on filename and custom keys. Threads, materials and colors will be addressed in Part2 of this tutorial. Click on the "Click here to add value" and name the value **Style**. This word is simply the author's choice and you can change it. This will be the key that will define if the bearing is an open, adjustable, normal or double bearing, so style seems to fit. Click again and add a value named **Filename**.

To make the **Filename** column control the filename of the iPart child you must make it a *File Name column*. Do this by RMB on the **Filename** value and checking the *File Name Column* choice. (see Figure 17)

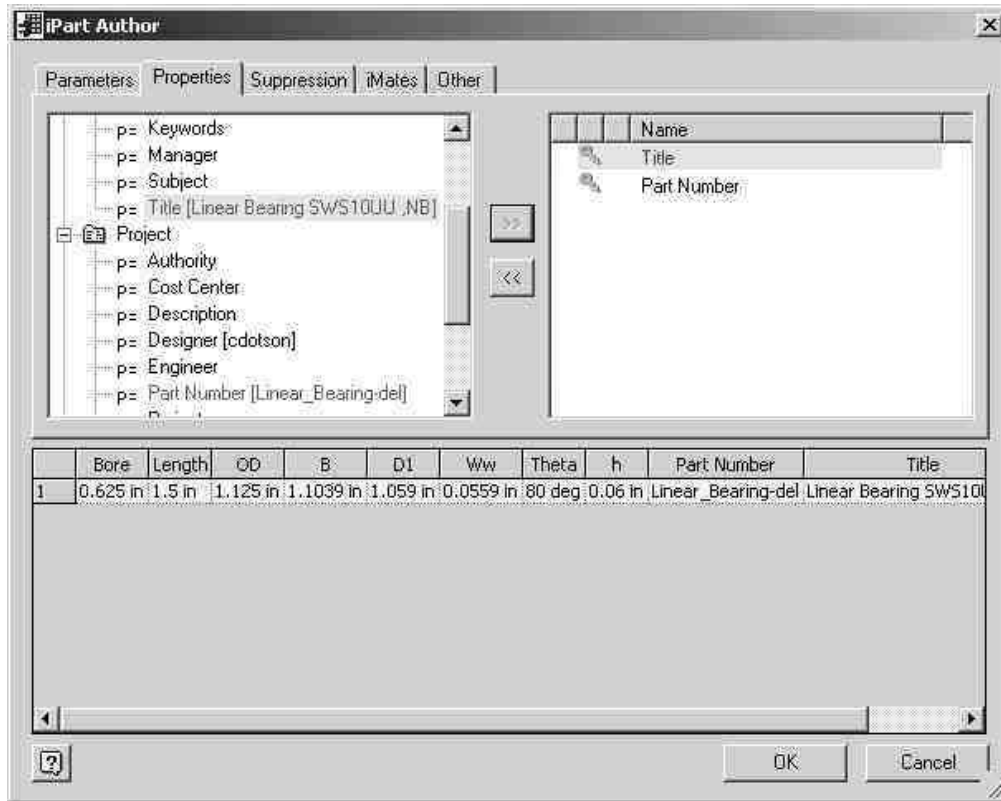


Figure 15 - iPart Author Properties Tab

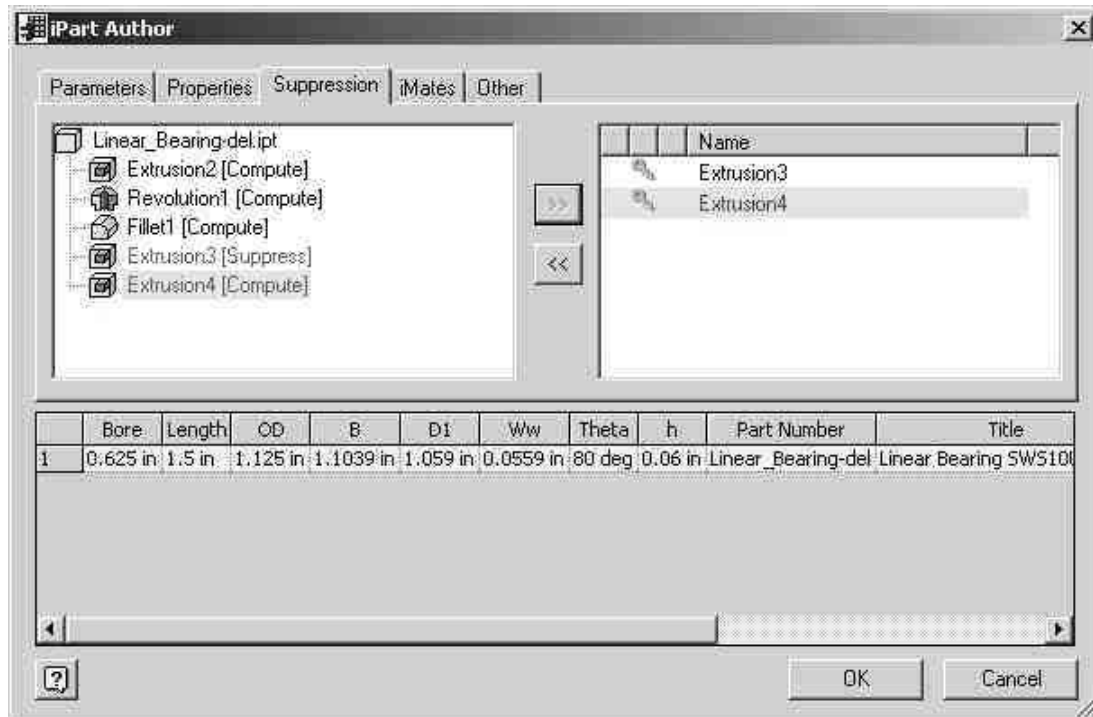


Figure 16 - iPart Author Suppression Tab

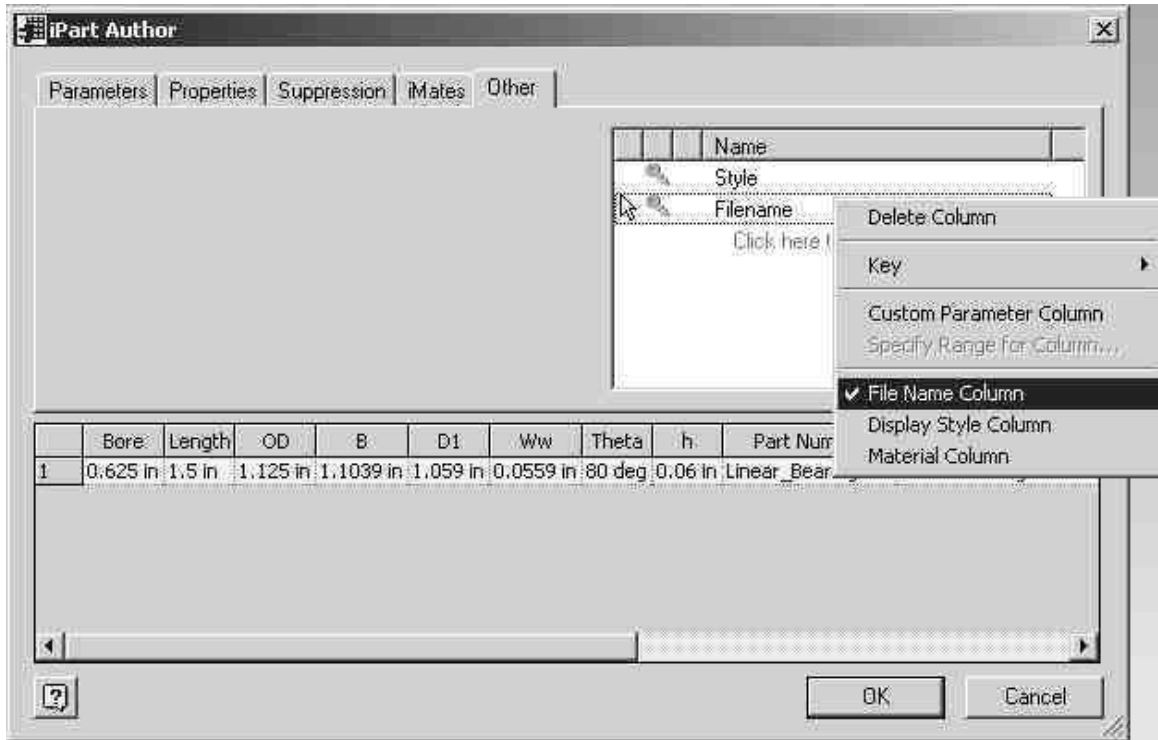


Figure 17 - iPart Author Other Tab

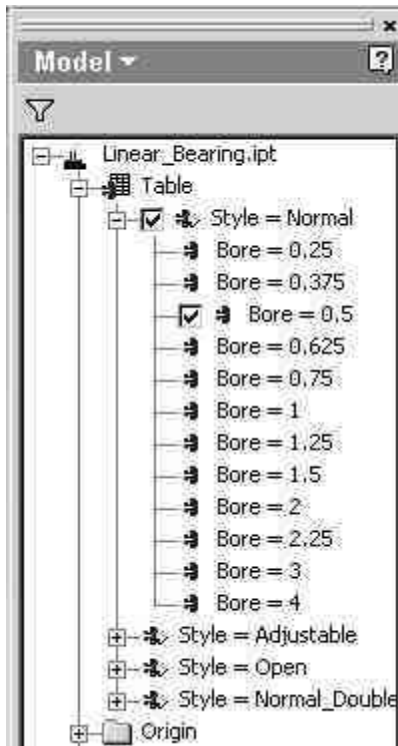


Figure 18 - Illustration of the Concept of Keys

Next you must set up keys.

Keys are what are used to define the hierarchy of an iPart. This is probably the most understood aspect of iParts. By setting keys you group your selections into logical orders. For example we could order our bearings by **Bore** or we could order our bearings by **Style** (or by **Length** etc..) This also controls the hierarchical grouping of the parts. In the picture below we see that the first tier of choices is the **Style** of bearing and under each style there is a **Bore**.

In this case the **Style** is the first key and the **Bore** is the second key. Keys may be nested up to 9 deep. Since you would likely choose a **Style** of bearing and then choose the **Bore** we shall set the **Style** to be the #1 key. We do this by RMB on the **Style** value and choosing Key>[1]. This sets **Style** to the #1 key. (see Figure 19)

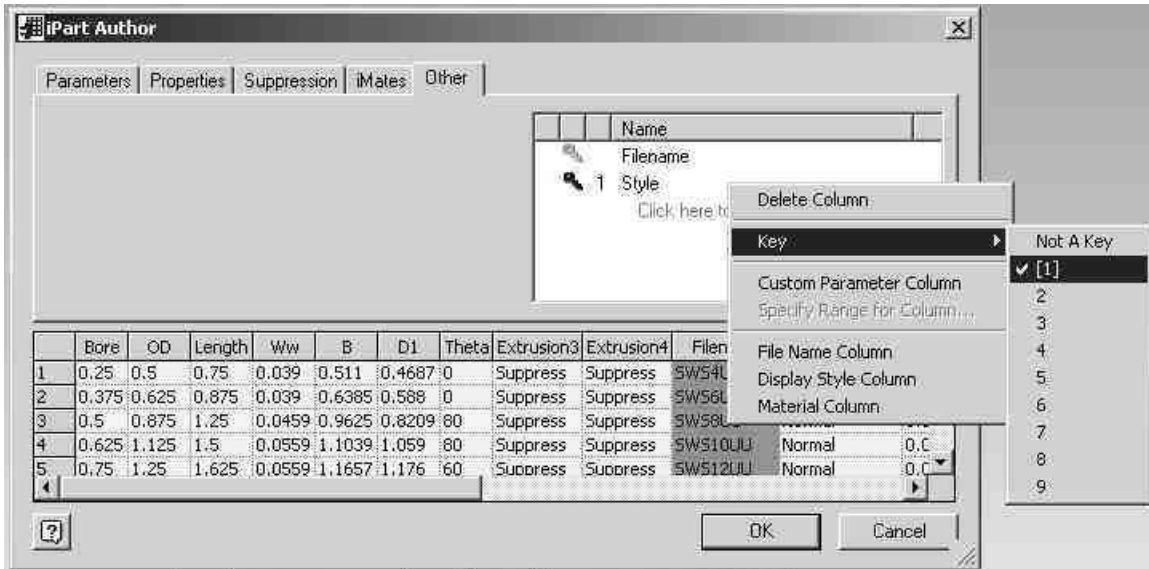


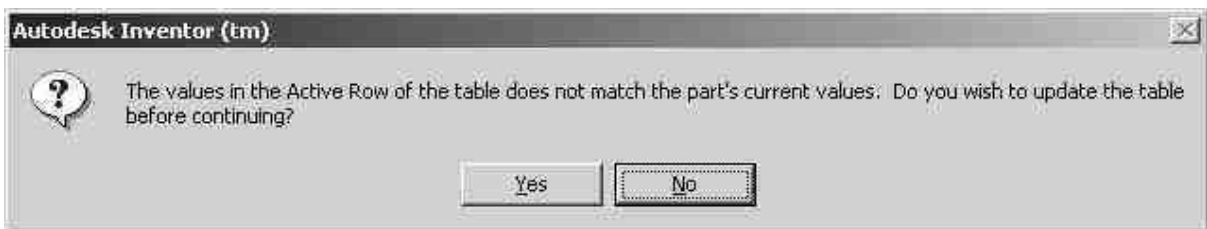
Figure 19 - Setting Up Keys

We now need a secondary key. Since we have defined **Style** as key #1 it would be logical to define the **Bore** as the #2 key. Click on the Parameters tab and RMB on **Bore** and set it to key #2 by the same procedure.

We do not need to define any other keys as the **Bore** and **Style** define all of the other parameters. This is true in this part due to the fact that there is only one length for any given bore. If there were others we would need to define a third key.

Click OK and return to the part.

Throughout the next few steps you may encounter the following message.



Simply hit OK and the part will be recomputed. This error occurs when the model does not reflect the data in the table.

Now for the fun part...entering in all of that data! RMB on the Table icon in the browser and choose *Edit Via Spreadsheet...*

You will be presented with an Excel document that looks something like this. Your columns may be in a different order depending on what order you brought in the parameters in the iPart author. The Excel table is nothing more than a copy of the iPart

table that is listed in the iPart creation dialogue. The Excel format makes it easier to enter information and allows you to use functions like copy and paste and Excel formulas to create the data. You could just as easily create the data in the iPart table (but I wouldn't recommend it)

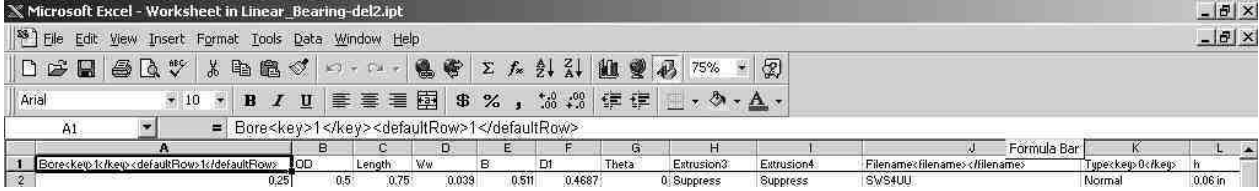


Figure 20 - Excel Table

The columns are the parameters and values that your brought in from the model. The rows beneath these headings are the values that these parameters will take for each iPart child. Currently you only have one row. We need more!

There is some strategy to entering the data. You want to vary the data of one key while holding the other constant. In our example we want to hold **Style** (key #1) constant while varying **Bore** (key #2). Since the **Bore** dictates the **Length**, **OD**, **groove\_depth** etc.. for this part we only have two keys. Begin to fill out the table by starting with the smallest **Bore** and working our way up while holding the **Style** constant as "normal". Since we do not want Extrusion 3 & 4 to show up we give them a value of *suppress*. The **Filename** and **Part #** fields can be filled out however you like, however it is a good idea to name the file in a consistent manner. (see Figure 21 or the Excel File Included in the zip file)

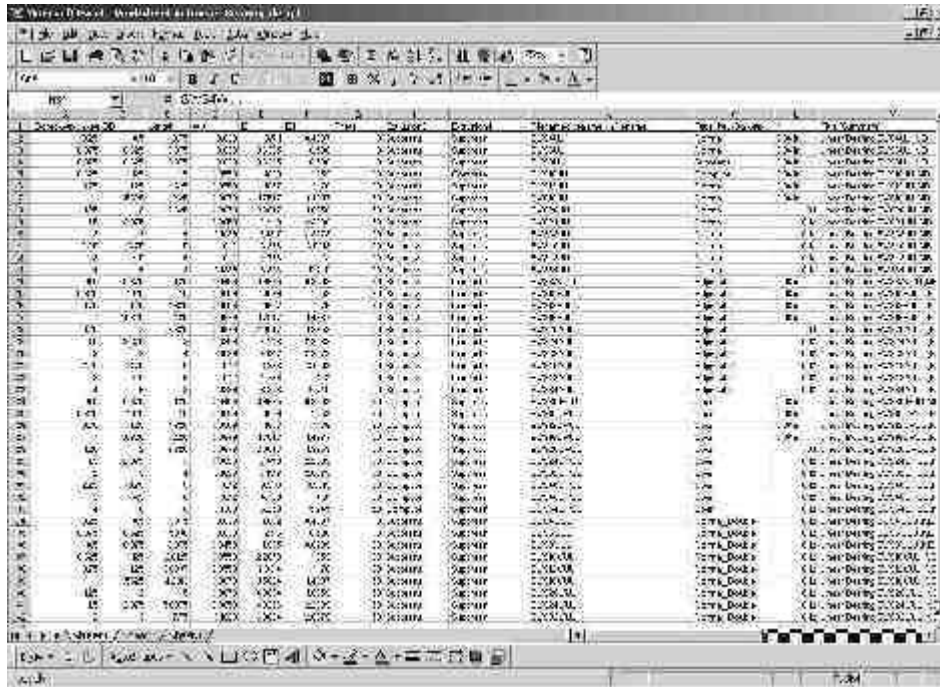
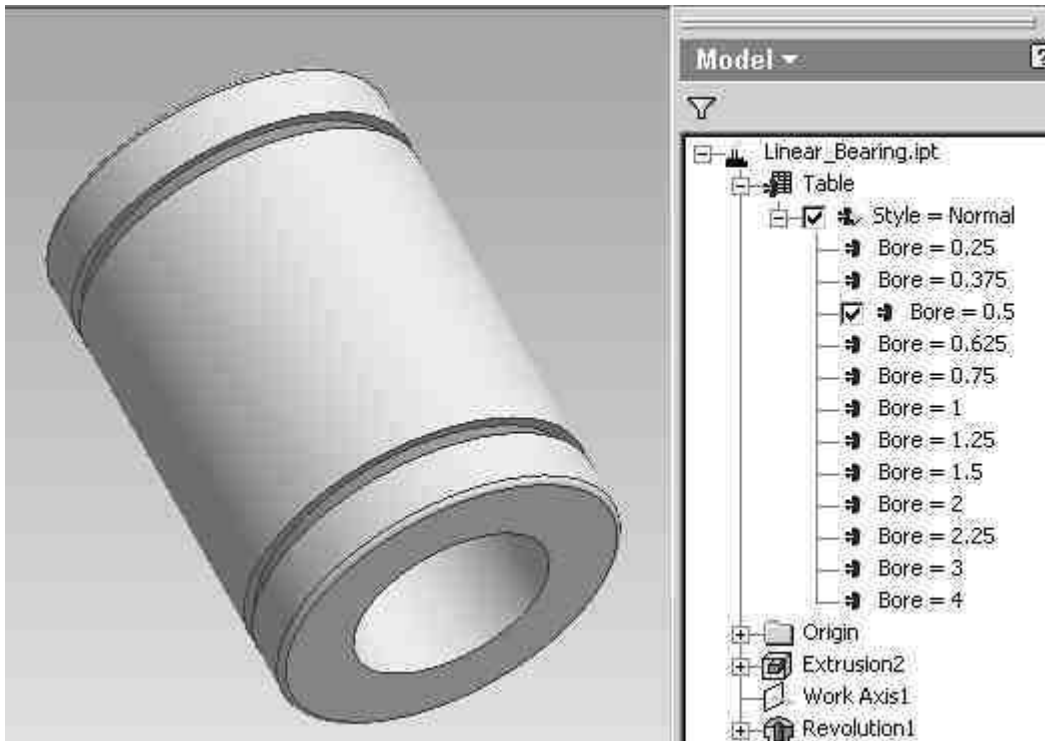


Figure 21 - Completed Excel File

After all of the "normal" **Bores** have been filled out you can save the Excel file, and select *File>Close and return to....* You will likely get the error mentioned above. If everything was done correctly you will get something that looks like Figure 22.



**Figure 22 - Completed "Normal Bores" iPart**

You can vary the size of the part by clicking on the table and choosing a bore. The file will update to show the size changes.

Go back to the spreadsheet and add the rest of the data. Again group the data by the first key (**Style**). Remember that with the open and adjustable styles you will need to compute or suppress Extrusion 3 and/or 4 depending on the style.

Once you are done save the Excel file, and select *File>Close and return to....*

Congratulations! You have made your first iPart! Be sure to test all of the configurations as it is easy to make a typo in the excel sheets. Your finished part should look something like Figure 23.

One last thing to do is to set the default row. This sets what version of the part will be presented to the user as the default size/style/type. We do this by RMB on the Table icon and selecting *Edit Table*. In the table in the bottom pane you see all of the data you entered in Excel. In reality you could have entered all of your data in this table (but Excel make it much easier to handle). To set the default row, RMB on one of the row headings (the numbered rows) and select *Set as Default Row* The row will turn light green. This indicates that this will be the default choice when the iPart is inserted into an assembly. (see Figure 24)



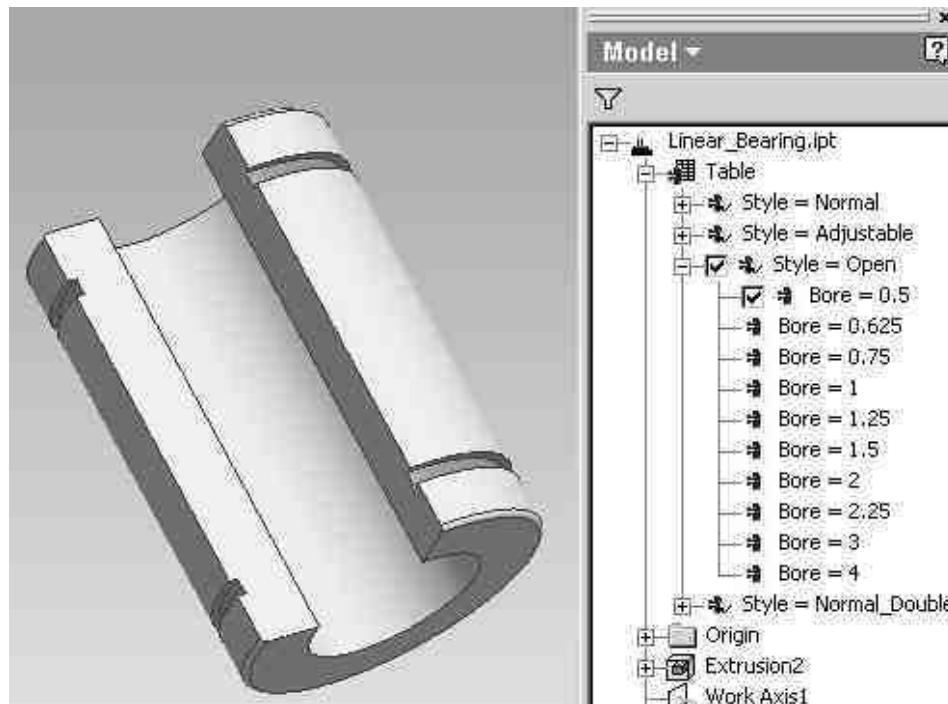


Figure 23 - Completed iPart

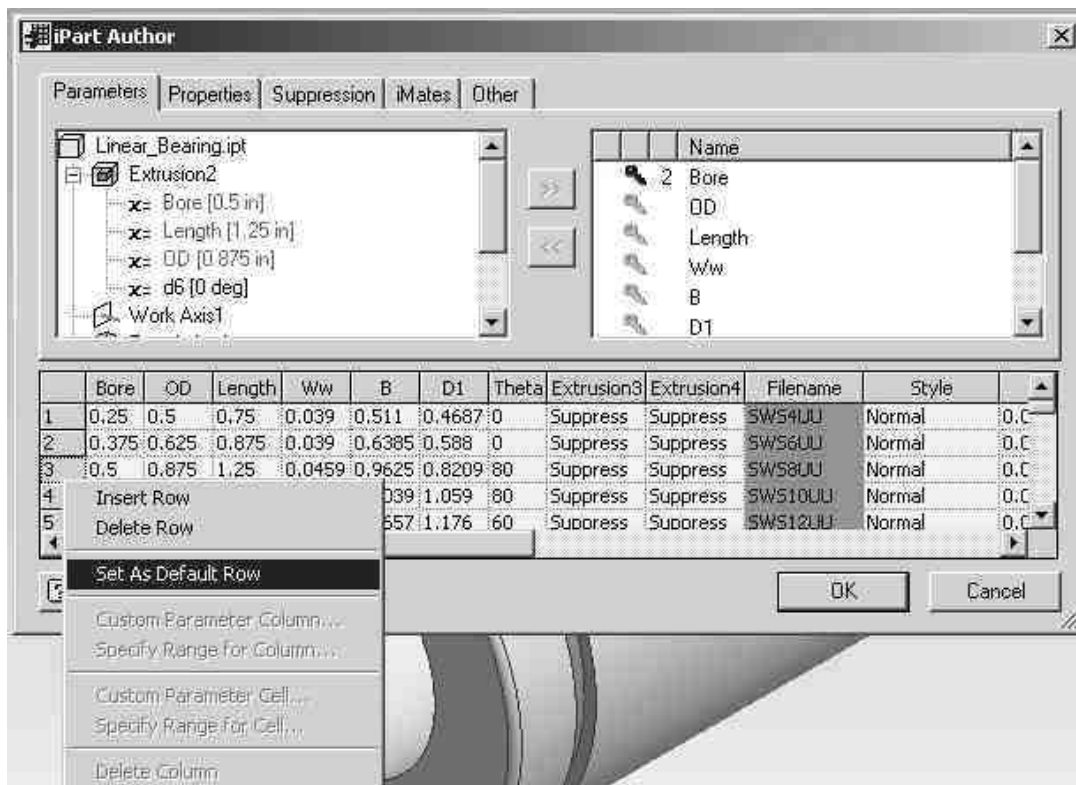


Figure 24 - Setting Default Row

Finally, in our picture you may notice that some of the cells are red. This simply indicates that these rows were computed in Excel as opposed to simply entered. In our photo we see that the filename column is red. This is due to the fact that this part used a "concatenate" function to develop the name. You can simply type in the name if you are unfamiliar with this Excel function.

In the next lesson we will talk about how to add threads, materials and colors to your iPart.