AUTODESK UNIVERSITY 2005
Walt Disney World Swan and Dolphin Resort
Orlando, Florida

Improve Quality Using Model Tolerances

Anthony B. Rodriguez - KETIV Technologies

MA11-1 Several validation tools come with Autodesk Inventor, including interference checking, visualization, properties, and motion. Using model tolerances during design, along with other validation tools, will help you increase the quality of your product. When you've completed your design, you'll also save time by having these tolerances displayed automatically on the drawing. Learn how to embed engineering calculations, based on how parts function, into your designs from the start. Use the built-in mechanical engineering handbook to reduce time looking up engineering calculations and information.

About the Speaker:

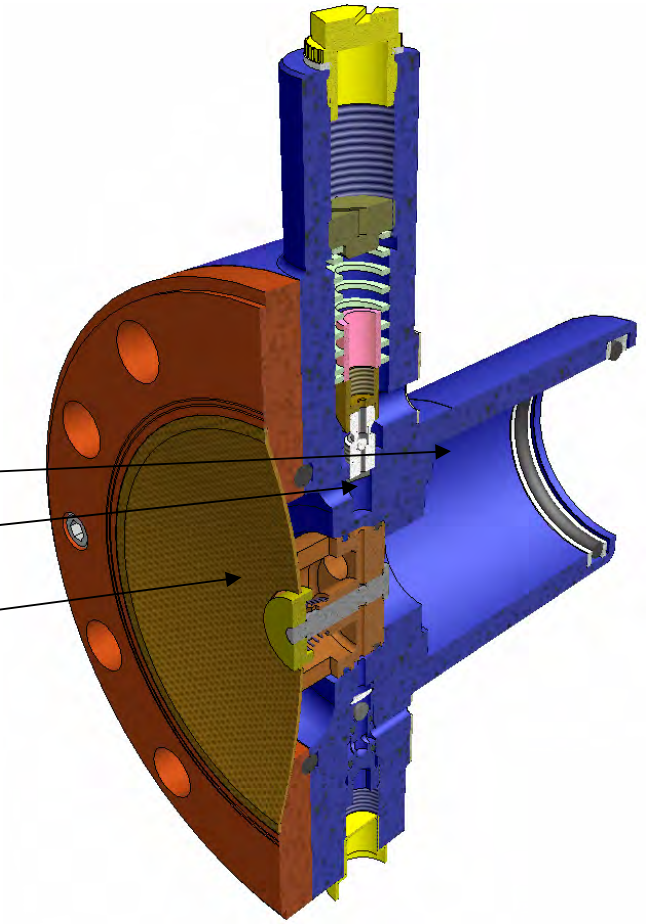
Anthony is responsible for conducting business audits and Autodesk Inventor implementations for KETIV Technologies, one of the top mechanical resellers. He was recently awarded the Inventor Channel Excellence award for his work. He is a Certified Inventor Expert and a Certified Inventor Implementation Expert. Anthony is also one of the founders of the Southern California Autodesk Inventor User Group. Prior to joining KETIV, he worked as a product R&D manager in the telecommunications industry for 5 years. Anthony is certified as an ISO9001 auditor.

anthony@ketivtech.com

Introduction

This Plunger assembly design will be used to study the validation tools Autodesk Inventor provides. Applying tolerances to your design within the design process gives you the ability to validate your design throughout its development cycle, instead of waiting until 2D drawings are produced. We will explore this technique together with Inventor's interference detection tools enabling you to improve the quality of your design.

There are several components internal to this Plunger Assembly that we will be looking at.



Parameter Tolerances

As you build up a model you have the ability to assign tolerances to each dimension/parameter that you use to describe your part. There are two ways in which you can apply the tolerance depending on the type of parameter generated. Each parameter can be set to the Upper, Nominal or Lower tolerance to then be used to evaluate your design.

The process to assign tolerances is described next:

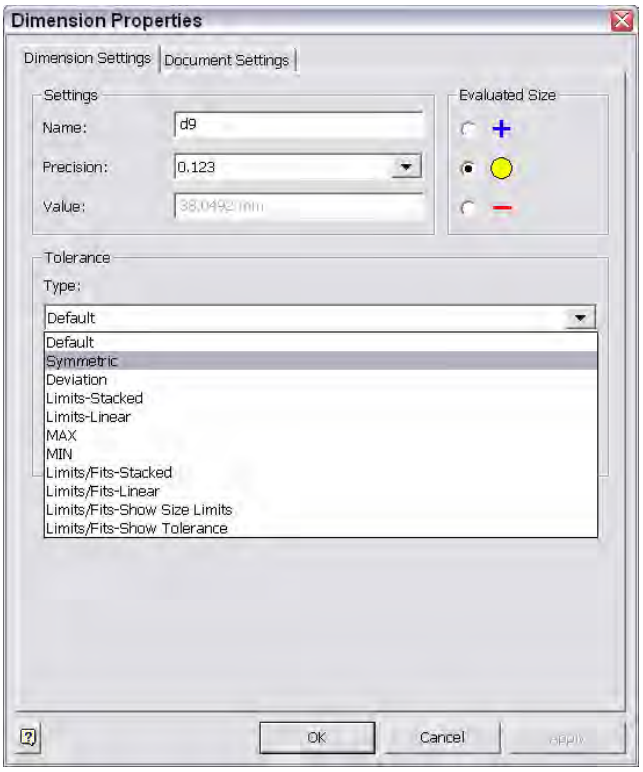
Sketch Dimensions – By right mouse clicking on any sketch dimension you will have an option to select "dimension properties".

The dimension properties dialog box is shown here to the right. Within the dialog box you have the ability to apply a tolerance type for this dimension.

You can drop down the list of tolerance types shown here.

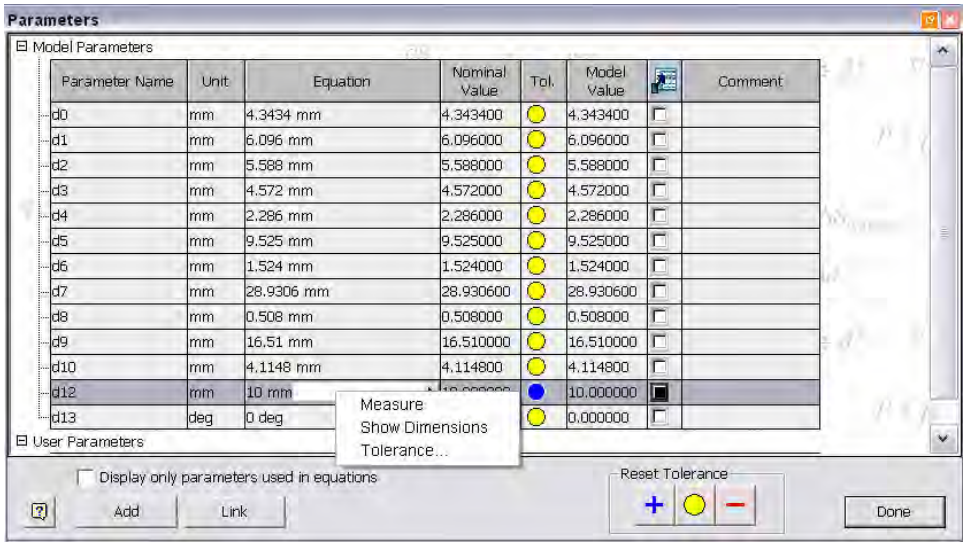
Once you have applied a tolerance type you can then assign the tolerance values.

On the top right of the dialog box you have three symbols representing the Upper, Nominal, and Lower tolerance values. Once you set the appropriate "Evaluated size" this will change the size of the physical model to equal the Upper or Lower tolerance values specified.



Feature Dimensions – The only way to add tolerances to parameters generated as a result of a feature is to go through the Parameters dialog box. You can access this from the Panel Bar with the f_x Parameters button. An example of a feature generated parameter is an Extrusion length.

In the parameter dialog box you can right mouse click in the equation column of the desired parameter and select the "Tolerance" option. This will display the Dimension Property dialog box as discussed in the previous example.

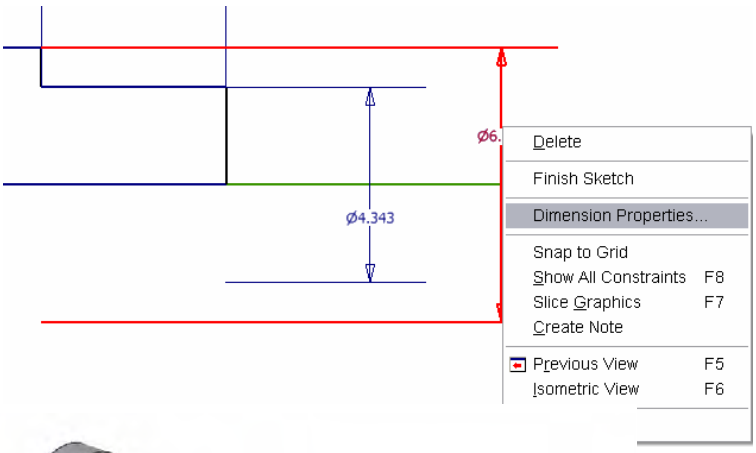


Exercise 1: Sketch Dimension Tolerances

Open the file "1 – Sketch Tolerance.ipt"

Select the Revolve feature in the browser and edit the sketch.

You can then select the dimension shown in the image and right click to select the "Dimension Properties" option.

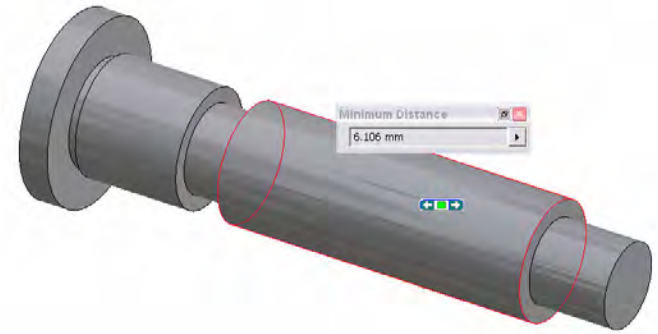


In the dimension properties dialog box you can create a symmetrical tolerance of .010 and set the dimension to the Upper tolerance then select OK.

You will then have to select the Update button 

.

When you measure the surface that has been changed you will notice that the physical model actually represents the Max tolerance condition as specified (6.106 mm).



Tolerance Defaults

Just about all 2D drawings that I see have a standard tolerance block which assigns standard tolerances to dimensions on the drawing depending on the precision of the dimension. An example is illustrated to the right:

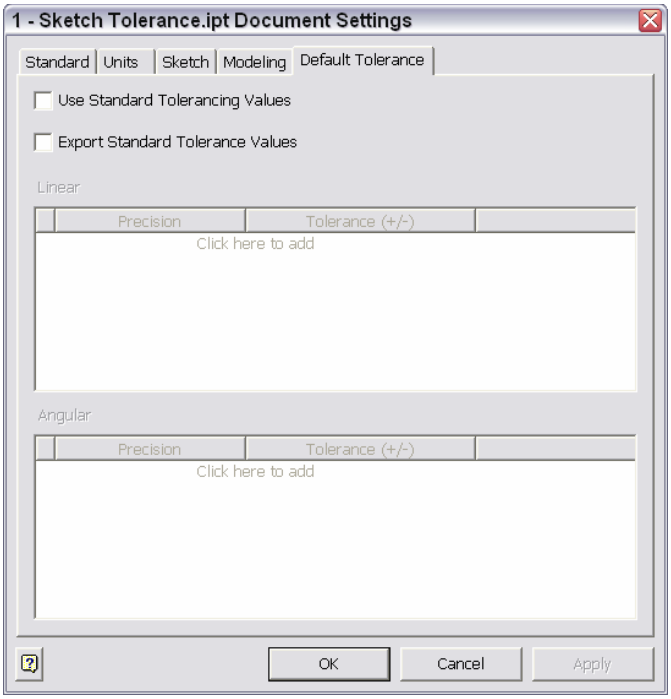
TOLERANCES	
LINEAR ±	ANGULAR ±
.X = **0.1**	X. = **1.0**
.XX = **.06**	.X= **0.1**
.XXX= **.031**	

You can now take advantage of these standard tolerances within the modeling environment. By establishing a standard tolerance like this within your IPT template file you can ensure that all parts created will have these tolerances applied as dimensions and features are created.

To access and specify your default tolerances open up your standard IPT template file.

Note: By opening the template file and applying your tolerances in the Default Tolerance window it will enforce these tolerances as your parts are built in the future.

Once your IPT template file is open go to the Tools menu > Document Settings > and select the Default Tolerance tab.



Use Standard Tolerancing Values- Select check box to use the precision and tolerance values set on this tab when creating dimensions.

This has to be checked in order to add tolerances.

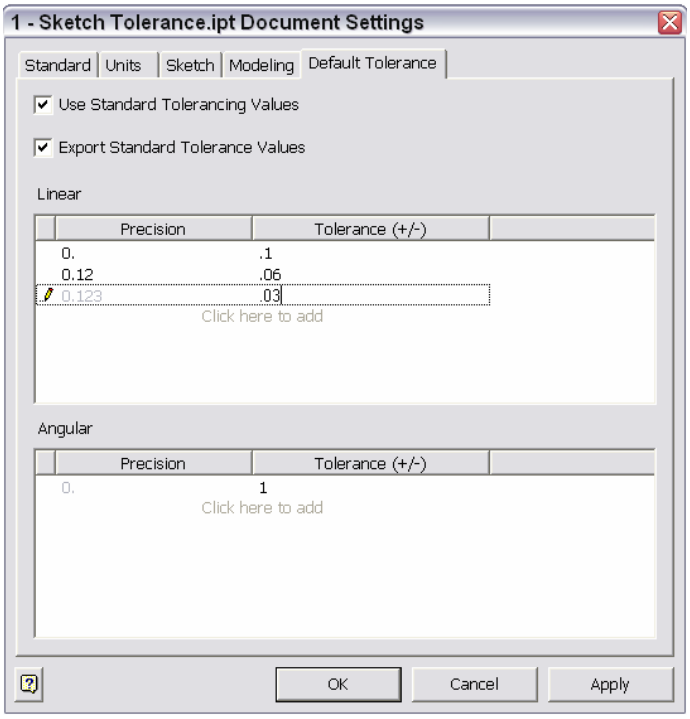
Export Standard Tolerance Values- Select check box to export dimensions to drawings using the precision and tolerance values set on this tab.

If you check this, all custom tolerances will still show in the drawing.

In the Linear/Angular area you can add tolerances based on dimension precision, matching your current standard tolerances.

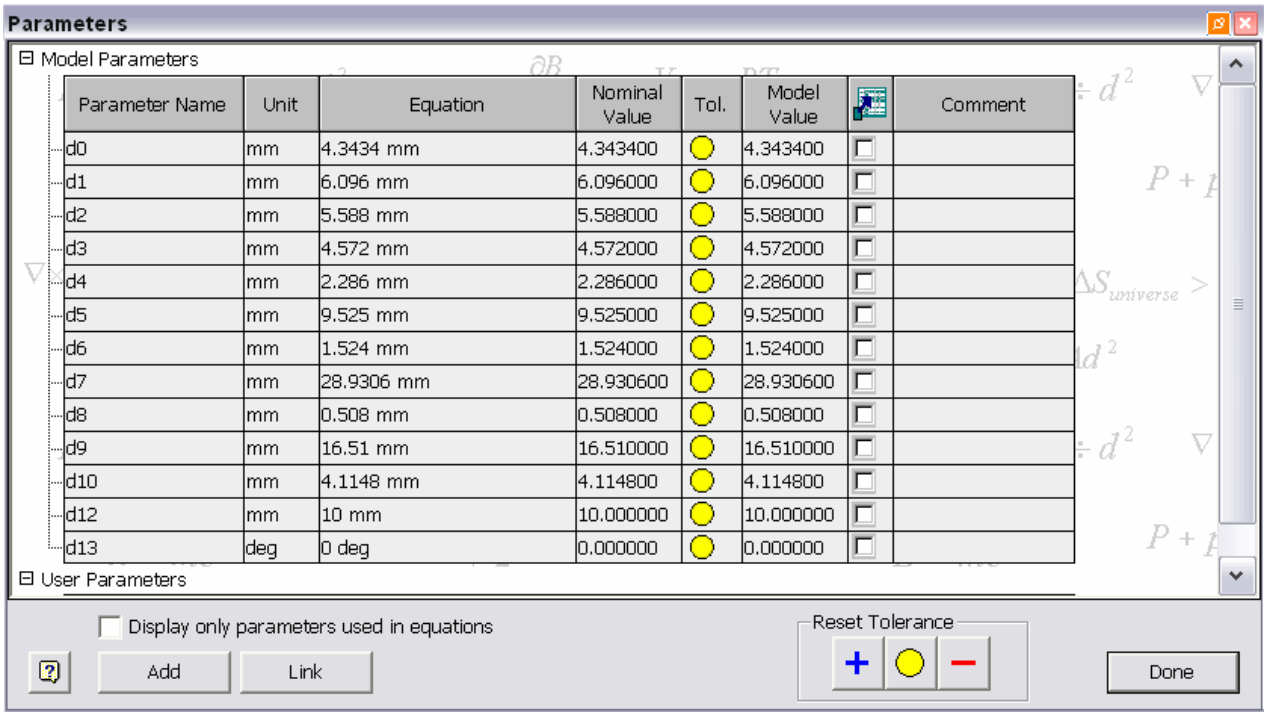
Select OK in the dialog box and save the file.

Now as you add features and sketch dimensions to a part they will all have tolerances assigned to them.



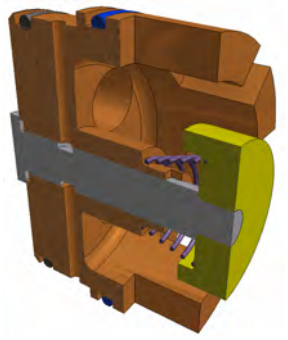
Applying Minimum and Maximum Tolerance Conditions

Minimum and maximum tolerance conditions can be applied to your parts at any time. In order to do this select the parameters button f_x Parameters in the Panel Bar.



In the Parameters dialog box there is a “Tol.” (Tolerance) column. By selecting the column you can select the Upper, Nominal or Lower tolerance for each parameter.

On the bottom of the dialog box there is an option to “Reset Tolerance”. When you select one of these options it will apply to all parameters of your part. This allows you to easily turn your entire part into a Minimum/Maximum condition.



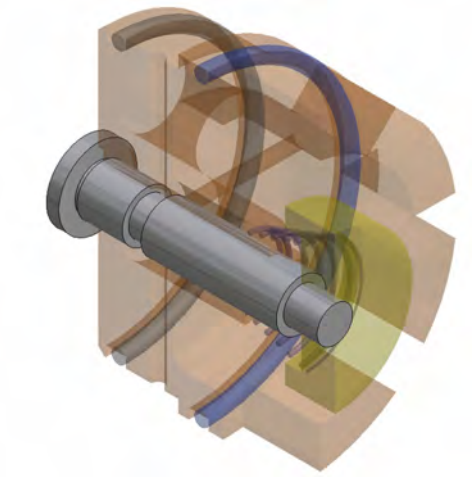
Excercise 2:

Open the file “2 – Apply Tolerance.iam”

This is the Plunger assembly. We will check the upper tolerance for the grey shaft that goes through the Plunger by utilizing the interference detection tool.

Next go ahead and create a half section view of the assembly by using the XY plane and the assembly section tool in the Panel bar.

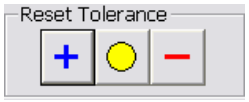
When your section is complete activate the grey shaft (61100500.ipt) by double clicking on it.



Then select the parameters button 

in the Panel bar.

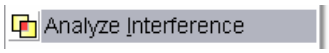
Next we will now change all of the parameters for the shaft to represent the Upper/Max tolerance value. Select the plus sign in the Reset Tolerance area of the dialog box.



This changes all parameter to the Upper tolerance value. Close the dialog box and select the update button 

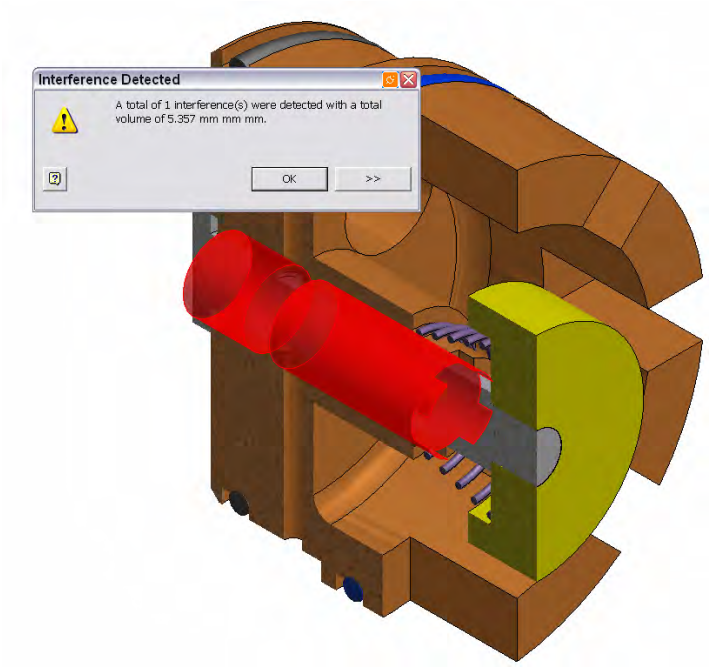
. This changes your model to represent the desired tolerance case.

We can now right click and select Finish Edit taking us back to the assembly.

Next go ahead and select the Analyze Interference tool 

, in the Tools menu.

Select the inner shaft (61100500.ipt) and the outer housing (61411000.ipt) and then select OK.



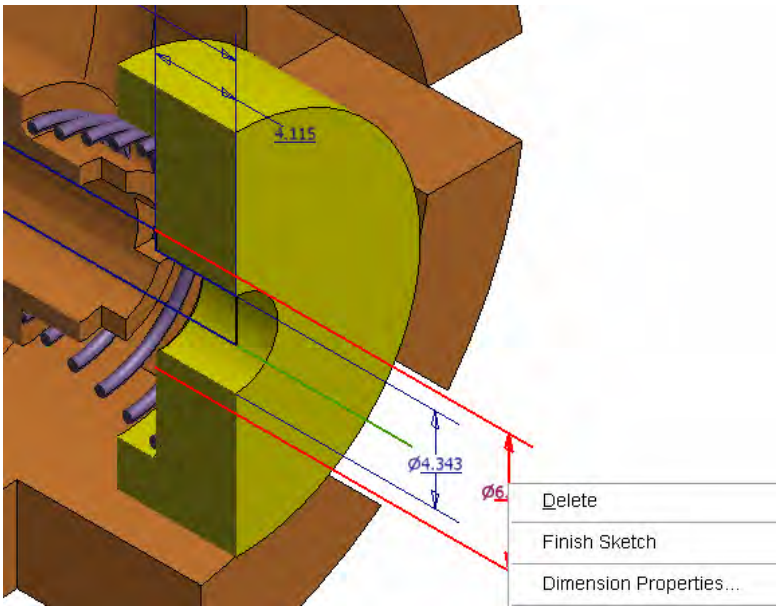
This now shows that if the shaft was manufactured to the maximum tolerance and the housing was manufactured to the nominal tolerance there would be problems.

Note: When the shaft was set to the nominal tolerance there was no interference.

In order to fix this we will have to modify and tighten the tolerance for the shaft diameter.

Activate the inner shaft and edit the sketch for the feature Revolution1 in the browser.

Right click on the dimension which defines the OD of the shaft as shown in the image and select the Dimension Properties option.



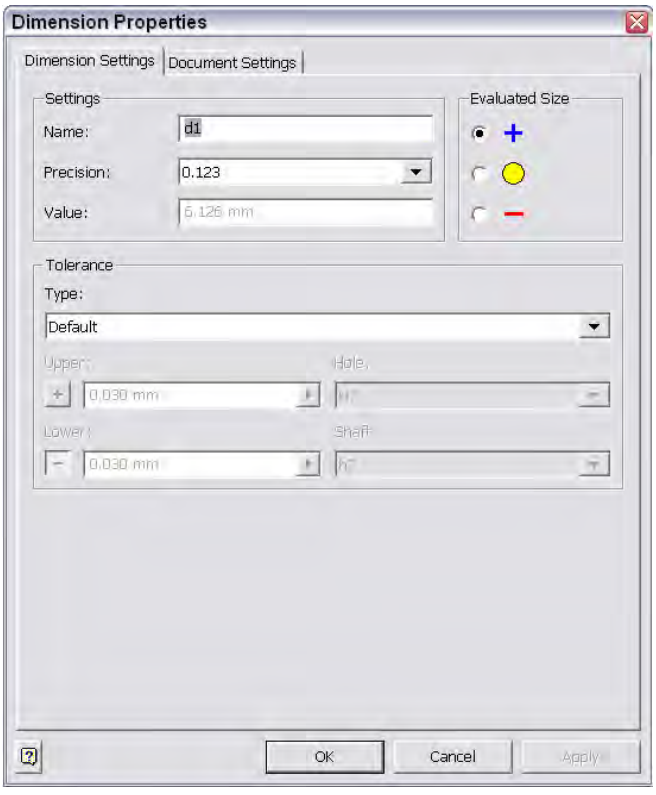
The Dimension Property dialog box allows us to override the Default tolerance. We can see what the current default tolerance is by selecting the Document Settings tab.

The current default tolerance is ± .03 mm. Select the Dimension settings tab so that we can now override that tolerance for this dimension.

Drop down the Tolerance Type list and select Deviation. We will now apply an upper tolerance of .00mm and a lower tolerance of .03mm.

You can now close the dialog box and return to the assembly to re-run the Interference detection.

Try applying tolerances to the housing and checking the worst case tolerance settings on the housing and the shaft.



Retrieving Dimensions in the Drawing

All dimensions with or without tolerances that were created in the model can be re-used in the drawing (IDW) by utilizing the Retrieve dimension tool.

Exercise 3

Open the file 61100500.idw .

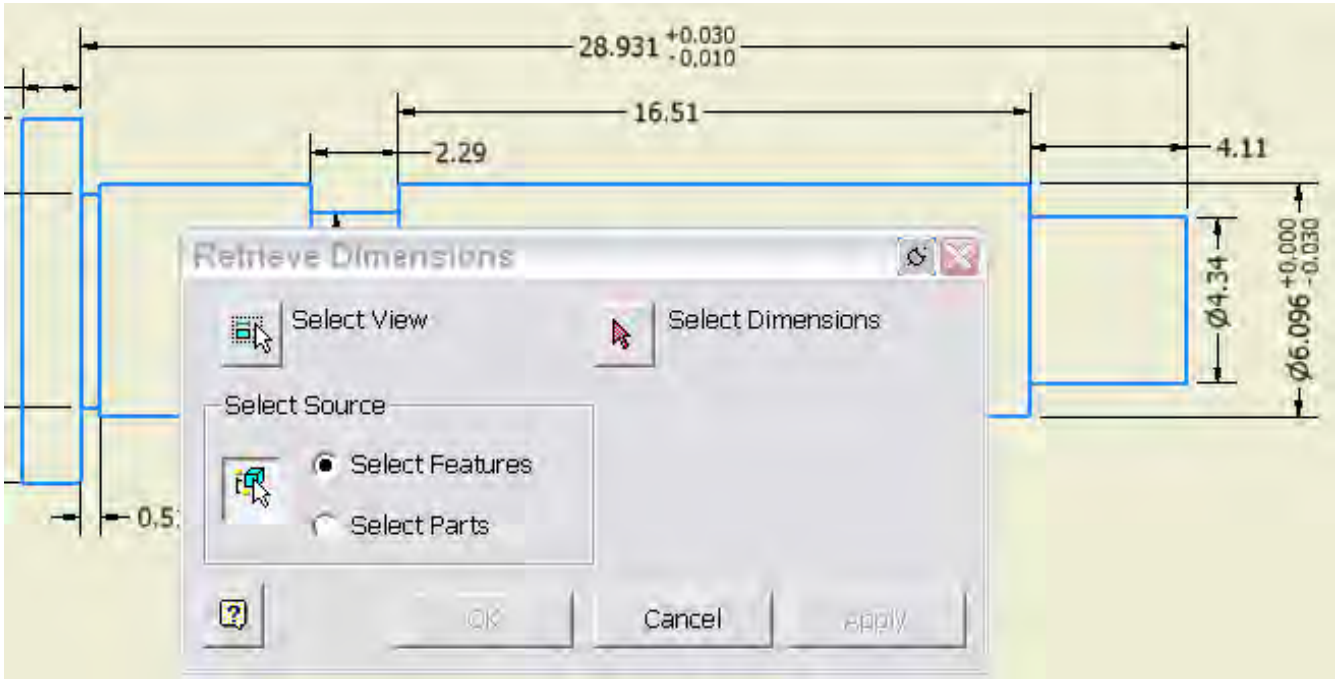
Some standard views of the shaft have already been created. Select the front view of the shaft and right click. In the right click menu select the Retrieve Dimensions option.

In the retrieve dimension dialog box under Select Source you can choose whether you want to retrieve dimensions for the entire part or individual features of the part. By default it is on Select Features so select the Revolve feature in the front view (an object line).

All of the dimensions for the Revolve feature will then show up in the view.

Next by selecting the Select Dimensions button in the dialog box you can choose which dimensions you want to keep on the drawing.

Select the dimensions with tolerances and then OK. By reusing the dimensions with tolerances from the model you have saved time and know that the tolerance values have already been validated with mating components.



Take advantage of the ability to capture tolerances in the design process then use them while validating your assemblies. This will hopefully improve the quality of your product as well as save you time and increased confidence when creating your drawings.

Thank you for attending this class. Feel free to contact me with questions - anthony@ketivtech.com

For Tips and Tricks visit: www.ketivtech.com