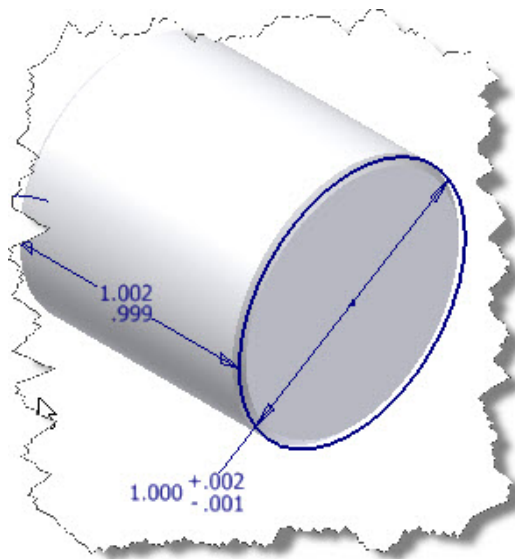


Tolerance Modeling with Inventor

The ability to work in a virtual Digital Prototype is certainly an invaluable tool for assembly ideation and visualization. Analysis tools such as Inventor's ability to check for potential interferences and drive assembly locational constraints through a range of sizes and positions can help a designer ensure that valid form, fit and function considerations are honored. This virtual prototyping in turn reduces design cycle times, time to market, and ultimately, cost. I am confident using these techniques made me a better Product Designer.

However, performing these analyses on parts modeled to a nominal size is insufficient and error-prone. Designers will place allowable tolerances on parts based on manufacturing processes, material properties, and cost factors. How, then, do I capture and incorporate these factors into my design verification?



Defining the Tolerance Model

Dimensional tolerancing is often achieved through the addition of tolerances to individual dimensions and/or the use of a global tolerance table on a drawing. By adding the requisite dimensional tolerances to the driving parametric dimensions within the model itself, a designer will be much better equipped to physically simulate and evaluate the impact of the change on the model and its related componentry. This process allows you to:

- Apply and set tolerances on a global or per-dimension basis to min, max, median, or nominal settings to evaluate fit in each state. This approach recognizes and accounts for the fact that not all machining processes will manufacture to an identical tolerance within the same part, or batch of parts.
- Account for tolerance stack-up across a series of features or parts.
- Highlight and underline all dimensions whose tolerances are not currently set to nominal.
- Automatically insert tolerances into a detail drawing.
- Reduce if not eliminate physical prototypes, revisions, scrap and rework time.

Albeit a very powerful concept, I am going to use a very simple, straightforward example here.

This process starts at the sketch level as we first add dimensions. Upon placing the relevant dimensions, in the Edit Dimension dialog, enter a unique Parameter name (highly recommended) and its corresponding nominal value as shown below in Figure 1. (Note: On older Inventor releases, you may have to already have pre-defined a User Parameter, or renamed a Model Parameter, for this purpose.)

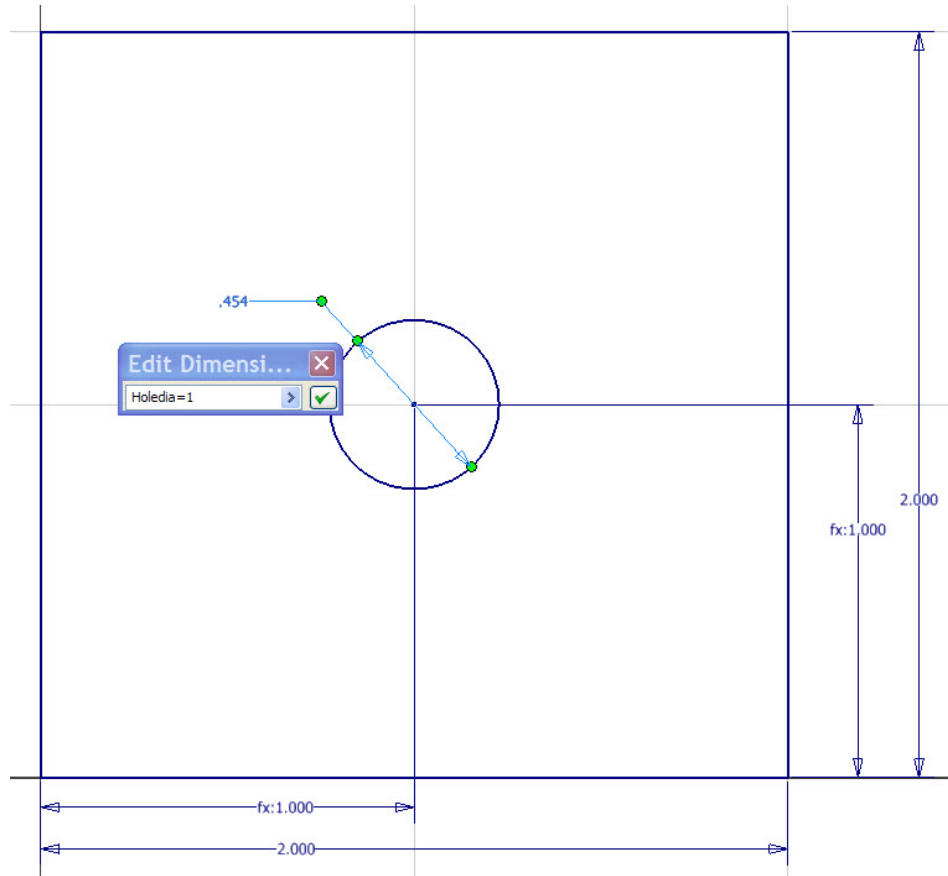


Figure 1

As mentioned, tolerances can be created on a global (document-wide) or added (or overwritten) on a per-dimension basis.

GLOBAL TOLERANCING:

To add a global tolerance, select “Document Settings” from the Tools menu tab as shown below in Figure 2.

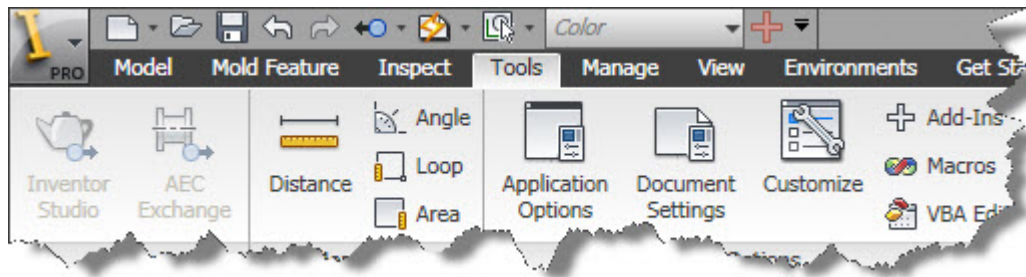


Figure 2

In the Document Settings dialog box, select the Default Tolerance tab (Figure 3).

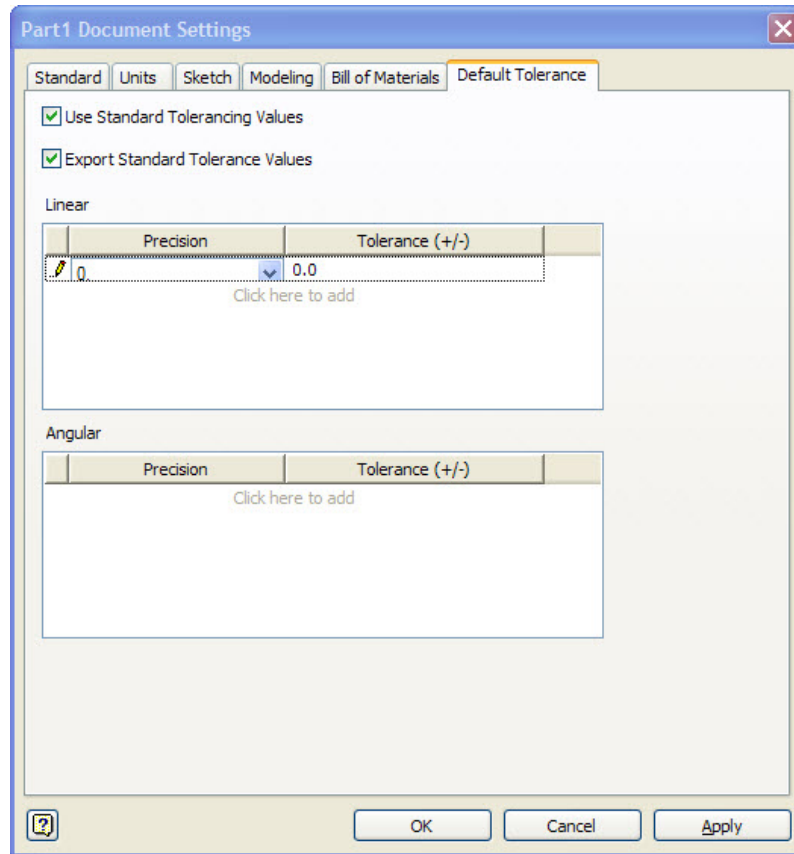


Figure 3

In this dialog box, the “Use Standard Tolerancing Values” checkbox allows us to use the precision and tolerance values defined on this tab when creating dimensions, and the “Export Standard Tolerance Values” checkbox is used to export dimensions to drawings using the precision and tolerance values set here.

In the Linear and/or Angular fields, use the pulldown menus under Precision to select a value and enter the corresponding tolerance next to it as shown in Figure 4. Add as many as needed.

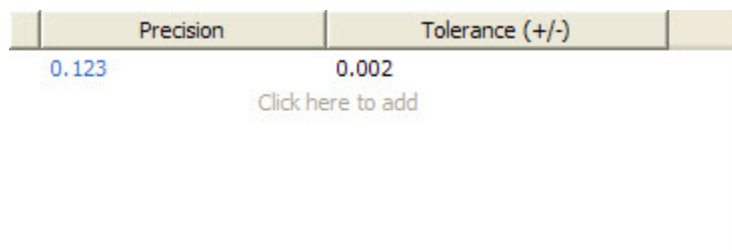


Figure 4

(TIP: Consider adding frequently-used values to your company templates.)

PER-DIMENSION TOLERANCING:

To add (or override) tolerances on a per-dimension basis, right-click on a dimension and select “Dimension Properties” from the context menu (Figure 5).

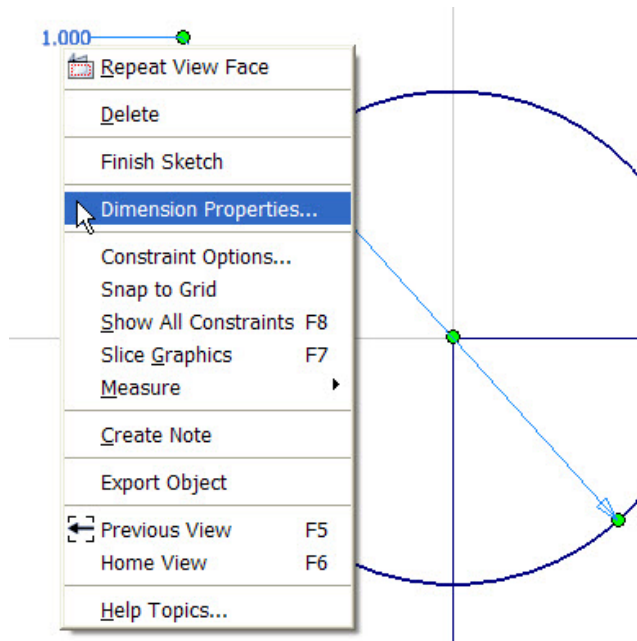


Figure 5

This may also be accomplished while in the Edit Dimension dialog box by clicking the right-arrow and selecting “Tolerance” as seen in Figure 6. Additionally, this method is also available when using the Edit Model Dimension function on the drawing file.

(TIP: Consider that this method is available any time a dimension is made visible, not only in the sketch editing environment, but also when using the “Show Dimensions” option on any feature, making this a practical workflow for Extrusion depth, Revolve angle, or any Loft or Sweep dimensions.)

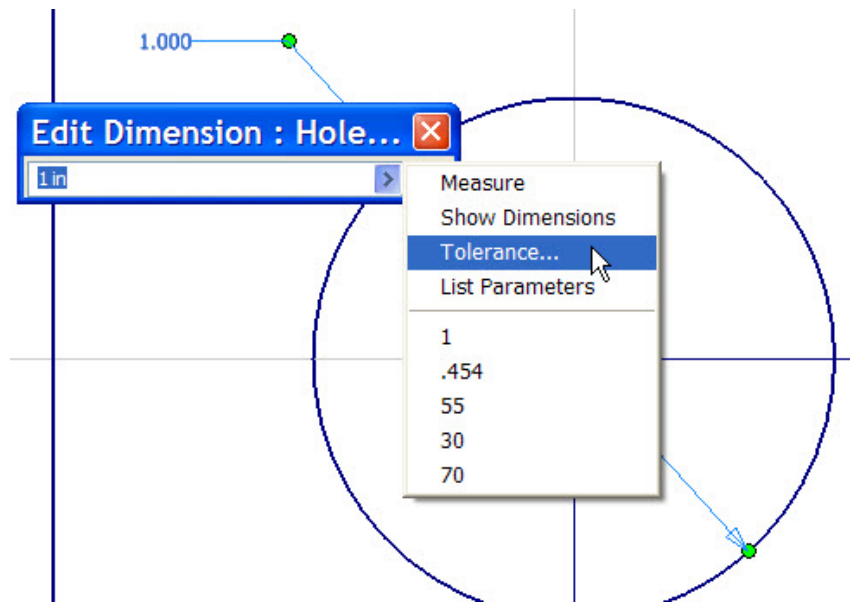


Figure 6

In the Dimension Properties dialog box, select the Tolerance Type desired from the following choices (Figure 7).

- **DEFAULT** – No tolerance specified.
- **SYMMETRIC** – Uses the same upper and lower tolerance, i.e., +/- .002.
- **DEVIATION** – Uses differing upper and lower tolerance, i.e., +.002/-.001
- **LIMITS – STACKED** – Displays the upper and lower allowable value in a stacked format.
- **LIMITS – LINEAR** - Displays the upper and lower allowable value in a hyphenated side-by-side fashion.
- **MAX, MIN** – Denotes the dimension value as Maximum or Minimum and labels it as such.
- **LIMITS/FITS** – Tolerancing method used on shafts and holes. Stacked and Linear display options are available, and, when selected, a user may also enter corresponding Hole and Shaft tolerances.

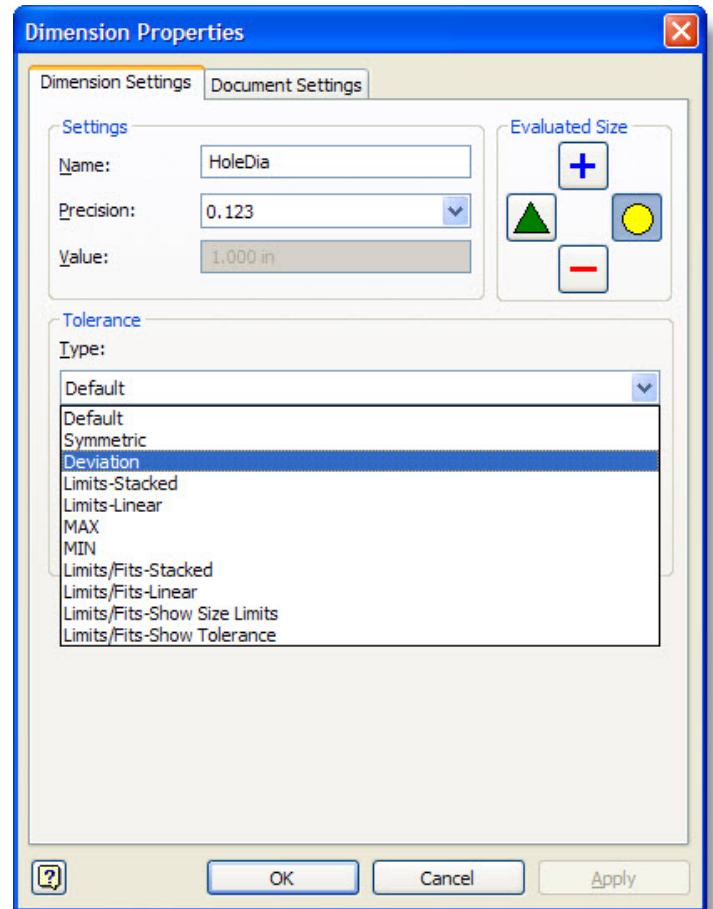


Figure 7

Enter the appropriate values as shown in Figure 8.

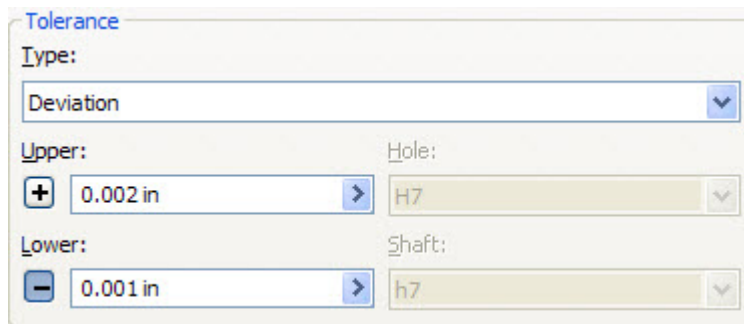


Figure 8

Now that we have attached tolerances to our dimensions, using either of these methods, how do we actually incorporate them into the calculated dimensional value? There are a couple of ways to accomplish this. The first method is to reenter the Dimension Properties dialog box shown earlier in Figure 7. In the upper right corner we will find an Evaluated Size selection area as shown below.

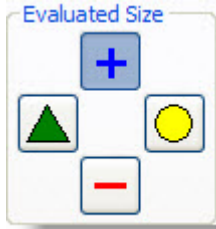


Figure 9

Clockwise from the top these allow me to set the dimension to its Upper (max), Nominal, Lower (min), or Median (min/max average) size.

An alternate method is from with the Parameters dialog box. The Parameters dialog box is accessed from the Manage menu tab (Figure 10).

(TIP: This is useful for changing multiple dimensions.)

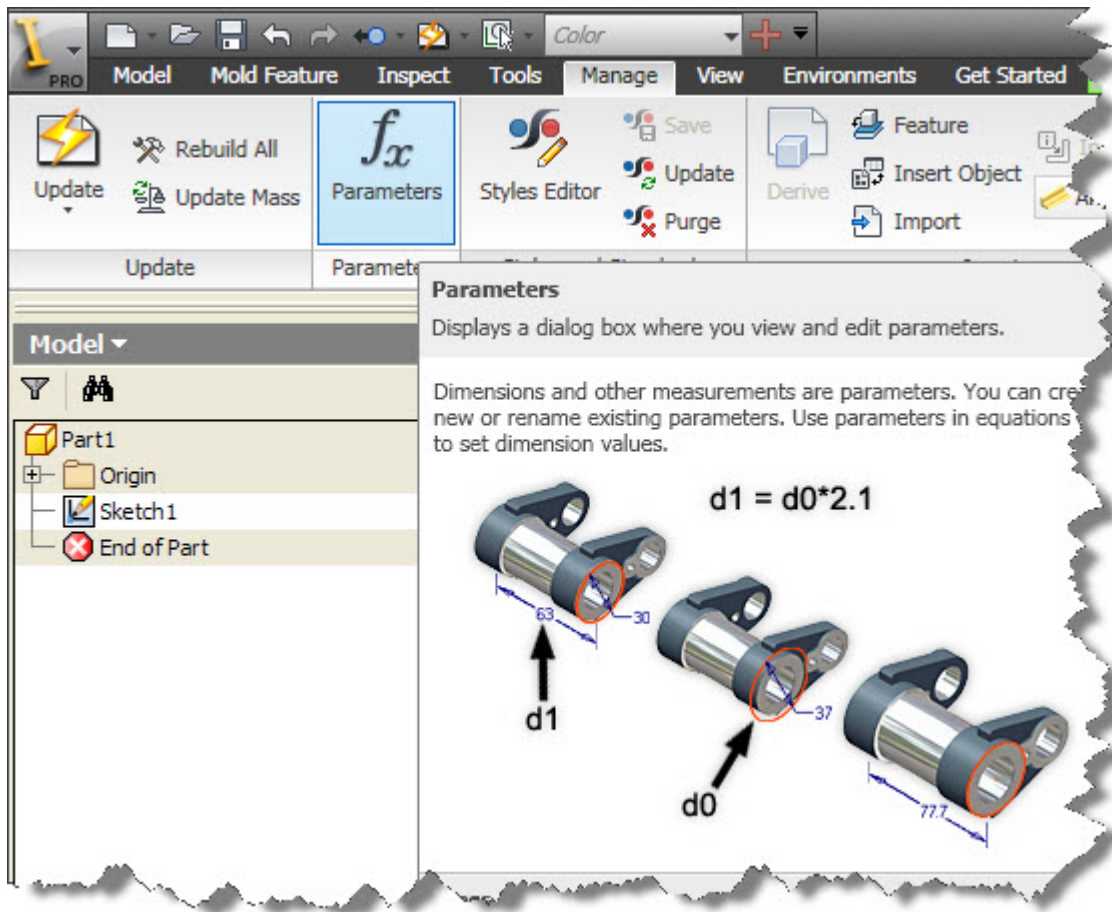


Figure 10

The Parameters dialog box, as seen below, also affords the ability to toggle any dimension to its required value. The Reset Tolerance selection in the lower part of this dialog box provides the ability to globally set all dimensions at once. This is especially useful to reset all values back to their Nominal state after analysis.

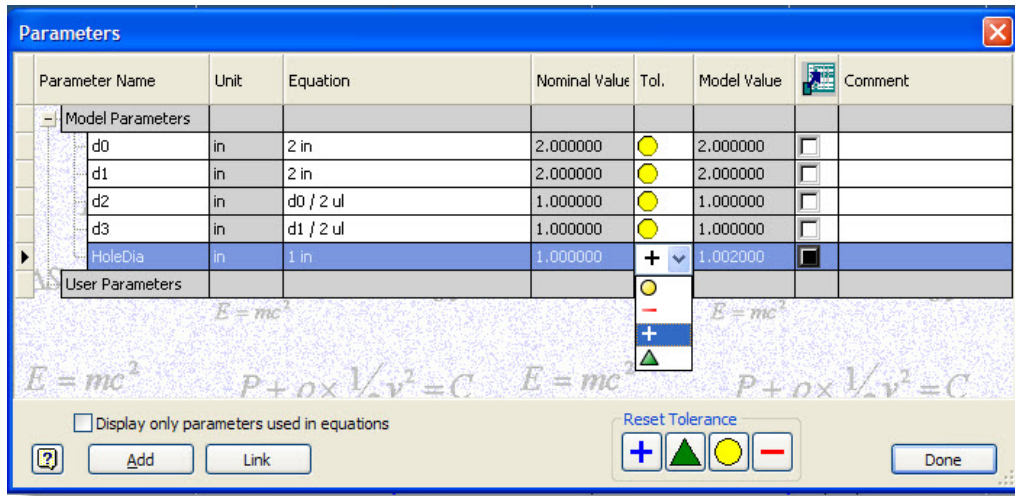


Figure 11

Dimension Display Options in the Sketch and Drawing Environments

There are several display options available to the user in the Sketch environment that can also carry through to the drawing. Right-click anywhere in the Sketch background and you will see a “Dimension Display” option (Figure 12).

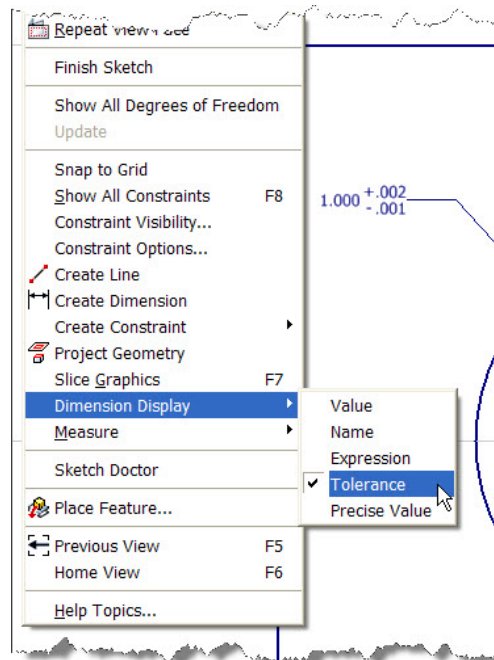


Figure 12

The options are as follows:

- **Value:** Shows the nominal value of the dimension.
- **Name:** Displays the default or user-defined name of the dimension, such as “d5” or in my particular example, “HoleDia”.
- **Expression:** Displays both Name and current Value, i.e., “HoleDia=1”.
- **Tolerance:** Displays the nominal dimension value including the tolerance added. Note: if there are in fact no tolerances placed on the dimension, this will look identical to the “Value” setting.
- **Precise Value:** Shows and underlines the evaluated dimension value per the current Upper, Lower, or Median setting, as shown below.

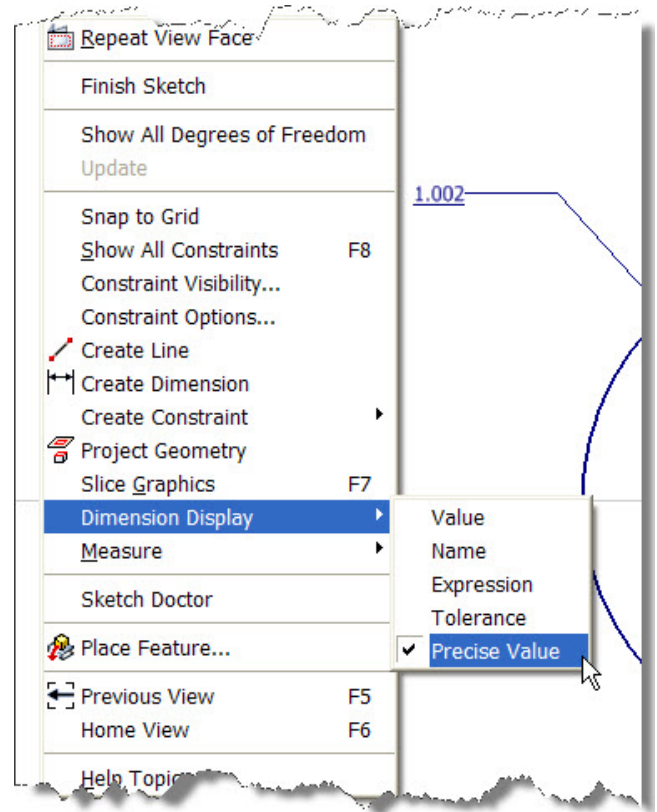
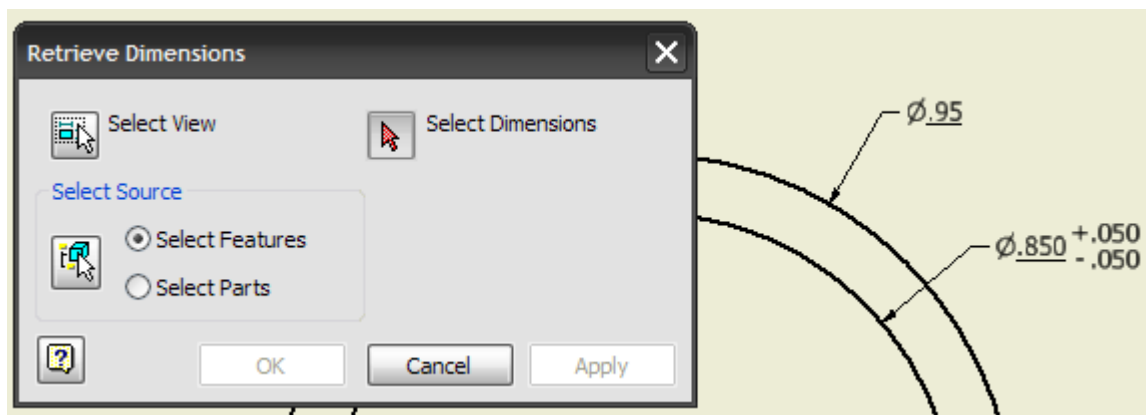


Figure 13

It is useful to note that these tolerance and display settings will carry through to the drawing when dimensions are retrieved, and when tolerances are defined and selected on the model, they will accurately be reflected when using any of Inventor’s Measure commands.



Performing an Interference Check

To initiate an Interference Check in Inventor’s assembly environment, select “Analyze Interference” from the Inspect menu tab as shown:

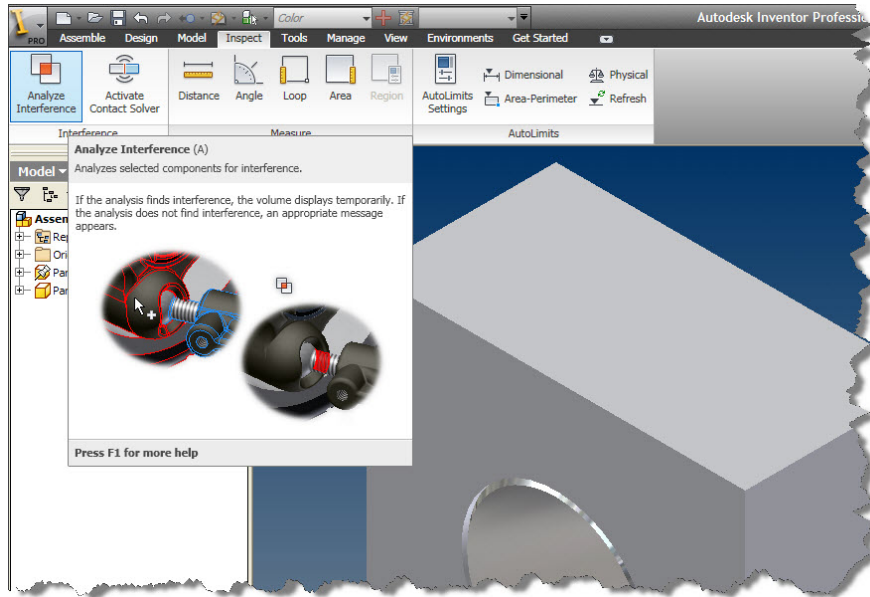


Figure 14

Two discrete sets of objects can be selected at this point. There can be any number of parts in any set. It is also sometimes convenient to pre-select some or all the parts in the assembly before invoking the command.

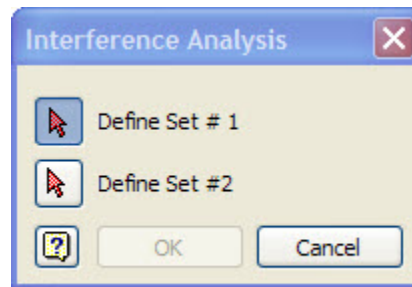


Figure 15

If any interference is detected, Inventor will highlight the offending bodies and bring up a dialog box stating the number and total volume of interference(s) found.

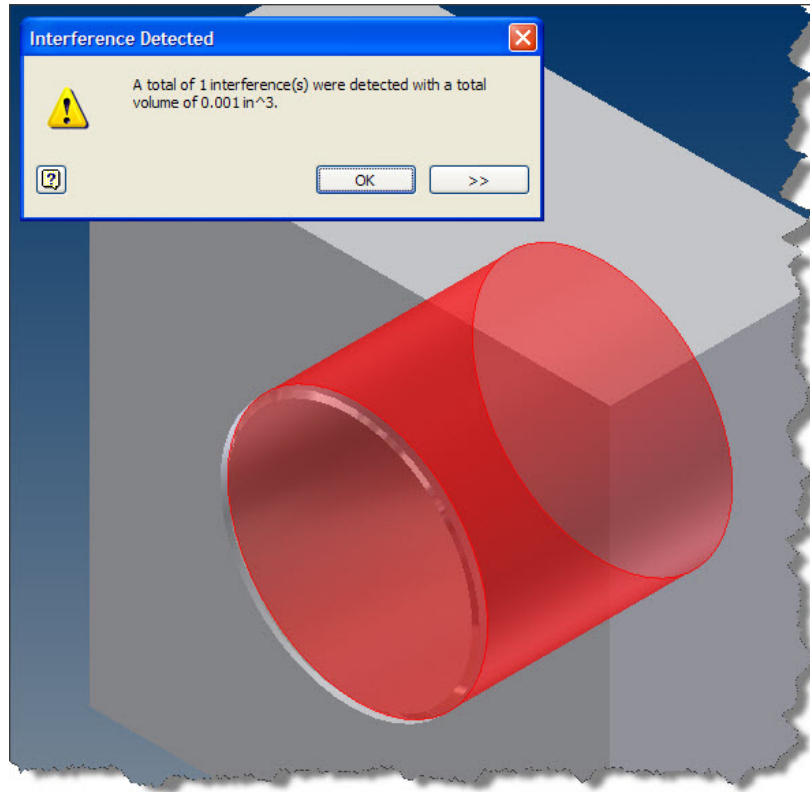


Figure 16

By expanding the dialog box we see a thorough listing of the exact parts and locations where conflicts are found.

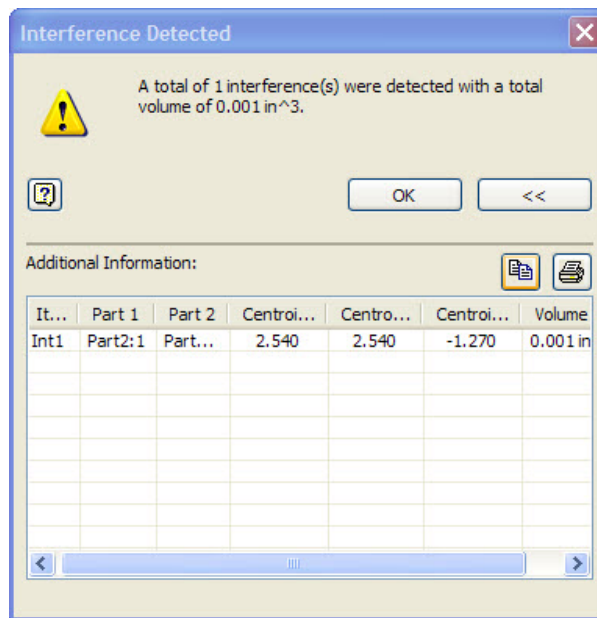


Figure 17

By reiterating this process through all the potential best- and worst-case scenarios, a designer may thoroughly evaluate and validate designs with minimal time investment.

SUMMATION

We've seen that users can take advantage of Tolerance Modeling to accurately predict whether designed assemblies will fit together, perform, and last. We've seen that draftsmen can automatically incorporate this data within detail drawings to ensure that design intent is correctly conveyed to manufacturing. The benefits in terms of reduced design cycle times, change orders, scrap and capital expenditure are self-evident.

Are you putting your Digital Prototypes to work for you?

About the Author: Sam Hochberg

Sam Hochberg, Senior Mechanical Application Engineer/Consultant, IMAGINiT Chicago

Sam comes from a manufacturing and design background, and has been using 3D CAD modelers as a Product Designer since 1981. While working at Motorola in the mid-80's, he was tasked with devising and instituting an in-house CAD training program, and also became involved in application support. He has also taught CAD part- and full-time at the collegiate level, and has worked as an Application Engineer at Computervision, PTC, and Autodesk resellers since 1987. He brings this combination of experience into his current pre- and post-sales support position, and in training customers believes in not just showing "how" but explaining "why."

- ATC Certified Instructor
- Inventor Certified Expert
- Manufacturing Solutions Implementation Certified Expert
- AutoCAD and AutoCAD Mechanical PSE Certified



All brand names, product names, or trademarks belong to their respective holders.
 IMAGINiT Technologies is not responsible for typographical or graphical errors that may appear in this document.

© 2009 IMAGINiT Technologies