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Introduction

CATIA Version 6 3D Tolerancing and Annotation

Upon completion of this course, the student should have a full understanding of the following topics:

- Creating annotation views
- Applying GD&T datums and controls
- Creating annotations
- Creating dimensions
- Creating construction geometry
- Working with note object attributes
- Creating reports
- Utilizing visualization tools
- Creating captures

3D Tolerancing and Annotation

3D tolerancing and annotation is used to define characteristics of parts and products in a 3D environment. By utilizing these tools, two dimensional drawings may not need to be created. Many companies have expressed an interest in going to a *paperless* environment, but find it difficult to accomplish. 3D tolerancing and annotation is one set of tools that can help make the transition a reality.

To effectively implement the tools in this course, you must be familiar with the fundamentals of geometric dimensioning and tolerancing (GD&T). It is not the intention of this course to teach GD&T. There is some assistance provided within the functionality of the workbench, but it will still allow you to improperly tolerance and annotate a design.

Geometrical Tolerancing

Geometrical tolerancing is the primary method used to accurately describe a part's design intent. When used properly, geometrical tolerancing can increase the tolerance zones to ensure that no part is rejected that will actually meet the design intent. Coordinate tolerancing is ambiguous, and does not give a full tolerance range for acceptable parts.

A good understanding of the fundamentals of geometrical dimensioning and tolerancing (GD&T) should be possessed before using these tools on a design. The Tolerancing Advisor will assist in the proper syntax of geometric tolerancing, but there is no way for CATIA to know the design intent. It is not the purpose of this course to teach GD&T, but rather to demonstrate how to apply it with the tools that are available in CATIA.

Many of the examples shown in this section are not finished parts. Instead, they are small examples of how to use the tools. You should make yourself aware of your company's procedures and standards in order to meet their criteria. The intention of this section is to introduce the various methods available for applying geometrical tolerances.

Form Controls

Form tolerances control flatness, straightness, circularity, and cylindricity. They are applied to a single element or feature, and are not related to datums. The first form control to be discussed is flatness.

Flatness

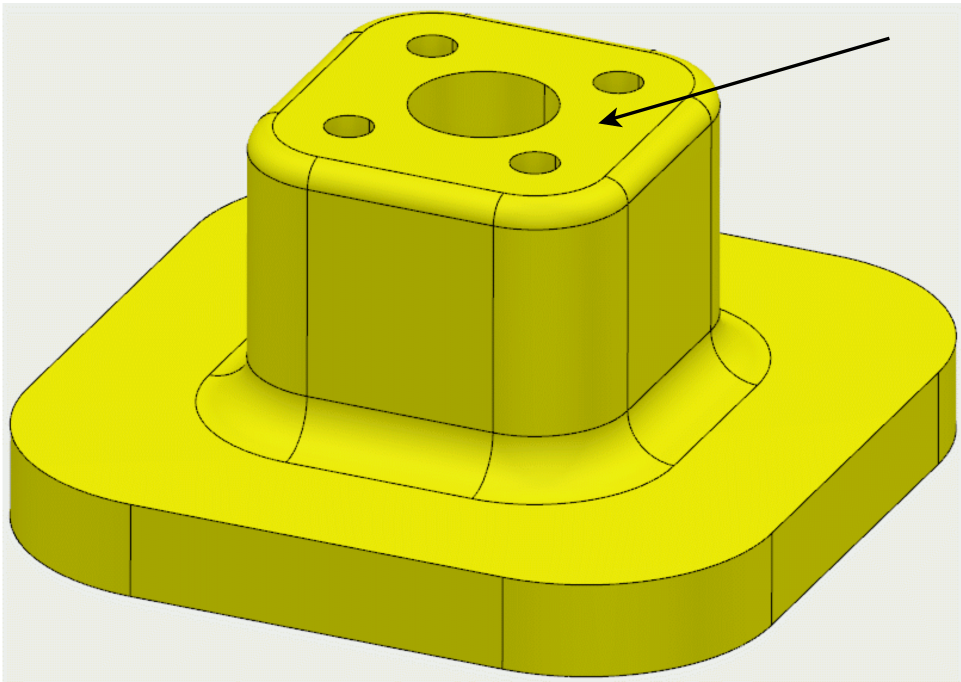
Flatness controls how flat a surface must be in order to meet the design requirements. All elements of the surface have to exist within the tolerance zone specified by two parallel planes that are separated by the tolerance value.

Open the 3DTA - Flatness document. A view already exists.

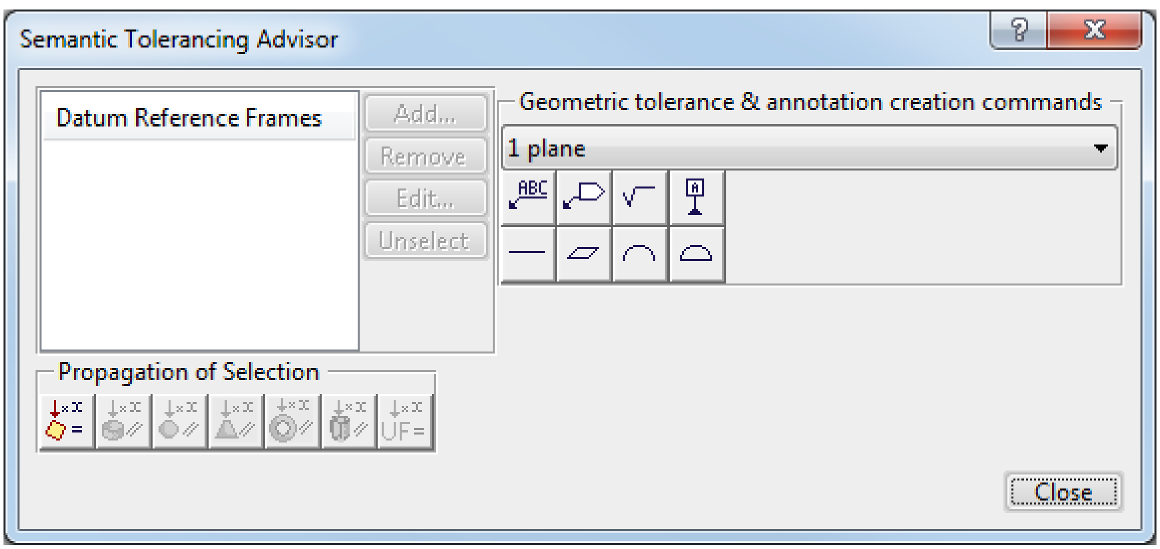
A flatness tolerance can be added by using either the Tolerancing Advisor or the Geometrical Tolerance icon. The Tolerancing Advisor provides guidance and will prevent the creation of invalid tolerances. However, it also requires certain steps to be followed, which can make it a slower process. Both methods will be demonstrated in the following exercises.

Tolerancing Advisor

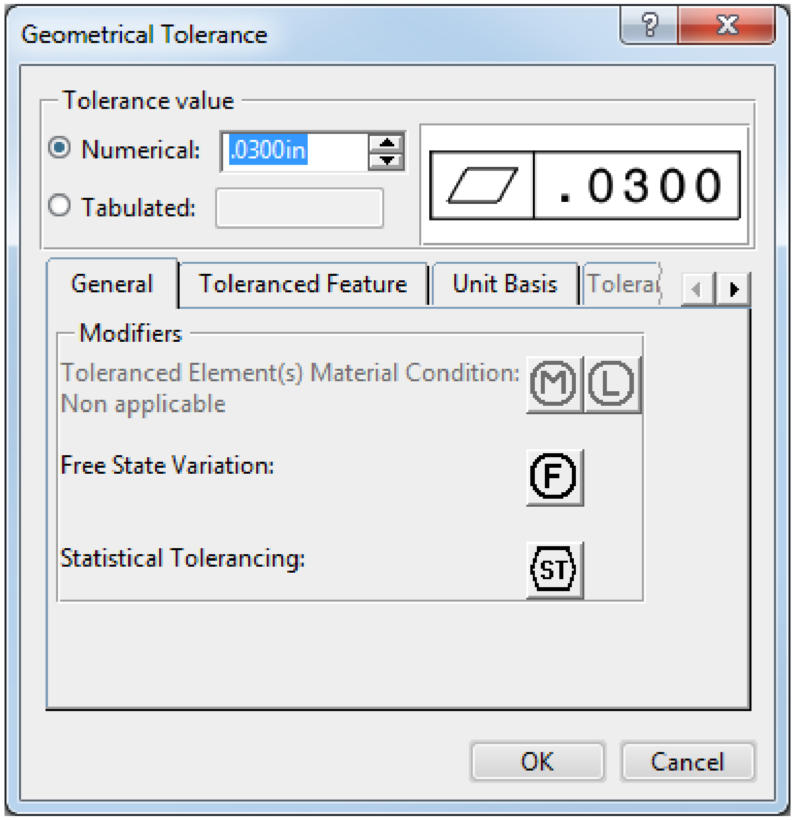
Select the Tolerancing Advisor icon, then select the top of the part.  You will create a flatness tolerance for this face.



Your window should appear as shown. The Tolerancing Advisor filters out the options that are not valid for a single surface. Only the pertinent options will be discussed in each exercise.

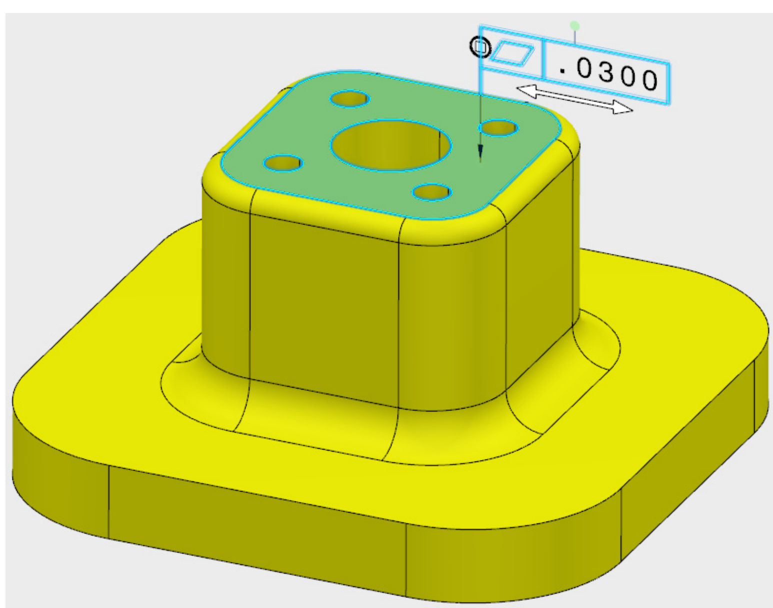


Select the Flatness Specification icon.  The *Geometrical Specification* window appears.



These options will be discussed as they are used throughout the exercises.

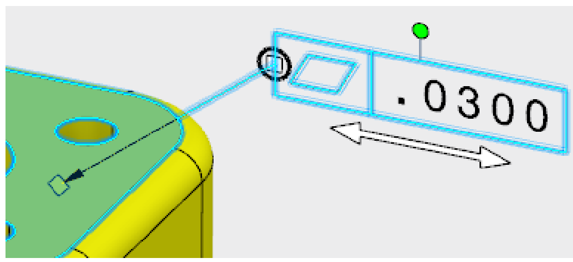
Change the *Numerical* value to 0.03 and select *OK.* The tolerance appears. The Tolerancing Advisor remains active, and the flatness specification is highlighted in the window.



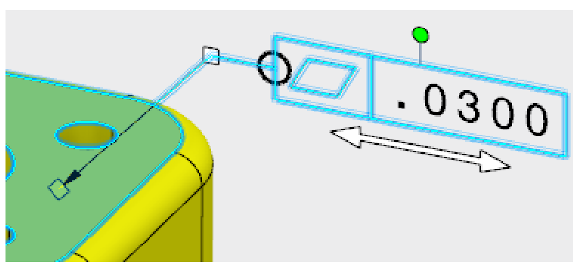
The tolerance appears in the tree as shown below.

There are many options for working with the leader lines, but they will be covered in more detail when annotations are discussed. For now, you will only move the tolerance and extend the leader.

Click and drag the tolerance to the right. Notice the white square at the left side of the tolerance and the yellow diamond at the end of the leader. These allow you to modify the leader.



Click and drag the white square to the left. The tolerance should now appear as shown below.

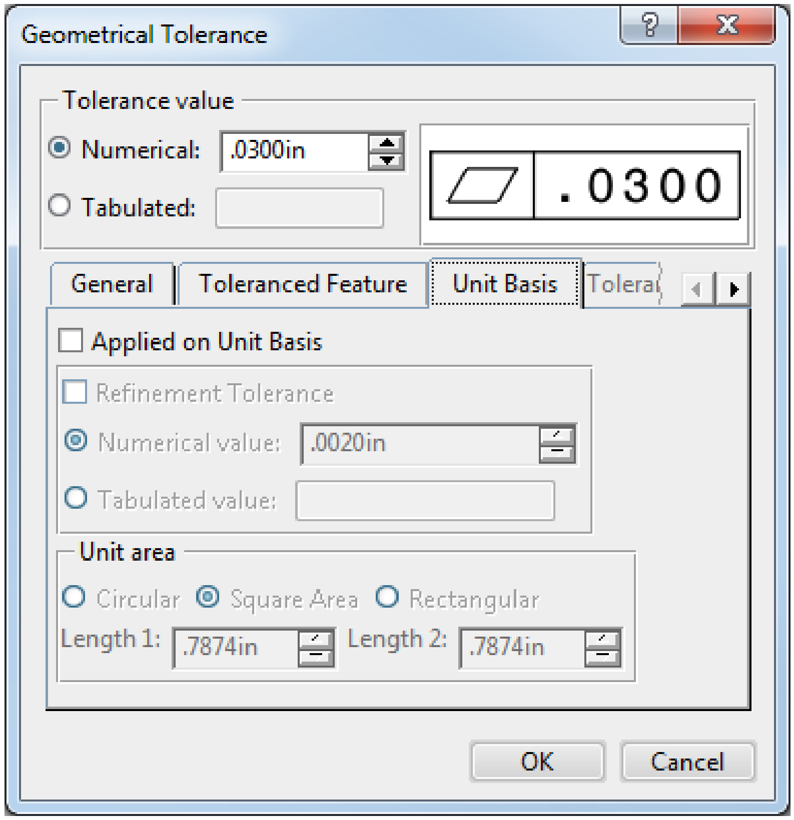


Feel free to move the tolerances to better locations throughout the exercises. Since it is the same procedure every time, it will not be mentioned repeatedly.

Select the Tolerancing Advisor icon if it is not still active, then select the bottom face of the part.  You will have to rotate the part up in order to select the bottom. The same options appear in the window.

Select the Flatness Specification icon, then change the *Numerical* value in the *Geometrical Specification* window to 0.03.  This time, you will specify a refinement on a unit basis.

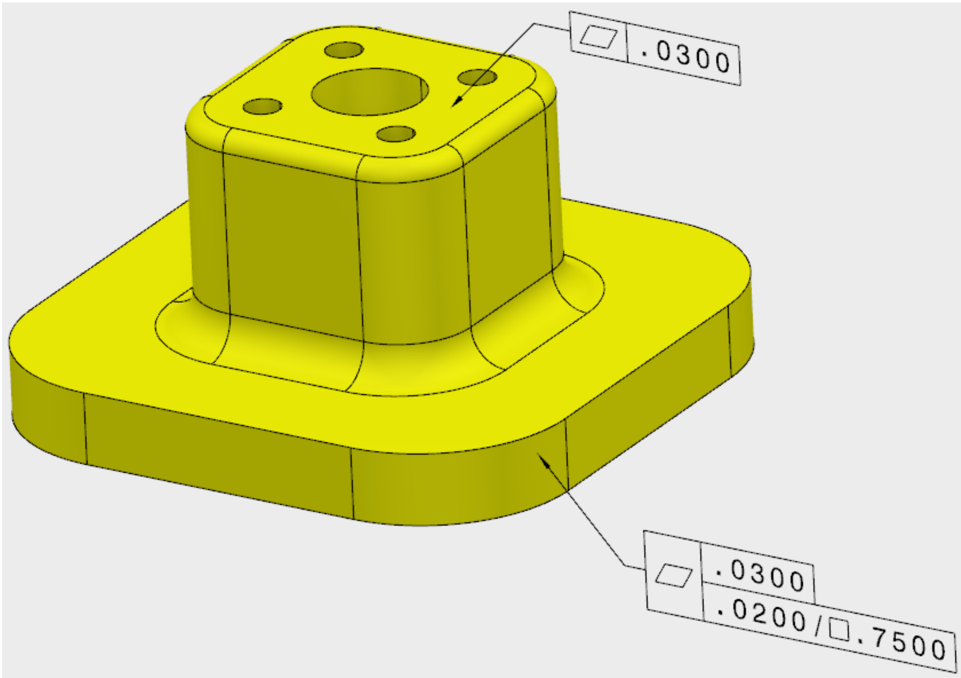
Select the *Unit Basis* tab.



Here, you can specify a refinement tolerance for a smaller area of the surface along with a total variation, or you can use it by itself. In this case, you will specify a refinement stating that for a 0.75 by 0.75 square area, the maximum variation can only be 0.02.

Select the *Applied on Unit Basis* and *Refinement Tolerance* options, then change the *Numerical value* of the refinement to 0.02 and *Length 1* to 0.75 and select *OK*. The tolerance appears.

Position the tolerance as shown below. It is stating that the maximum variation across the entire surface can only be 0.03 inches, and there can only be a maximum variation of 0.02 inches within a 0.75 inch square area.



Caution should be given for a unit basis tolerance without a total variation because a gentle bow in the bottom of the part could meet a unit base tolerance but have a huge variation across the entire surface.

Save and close the document.

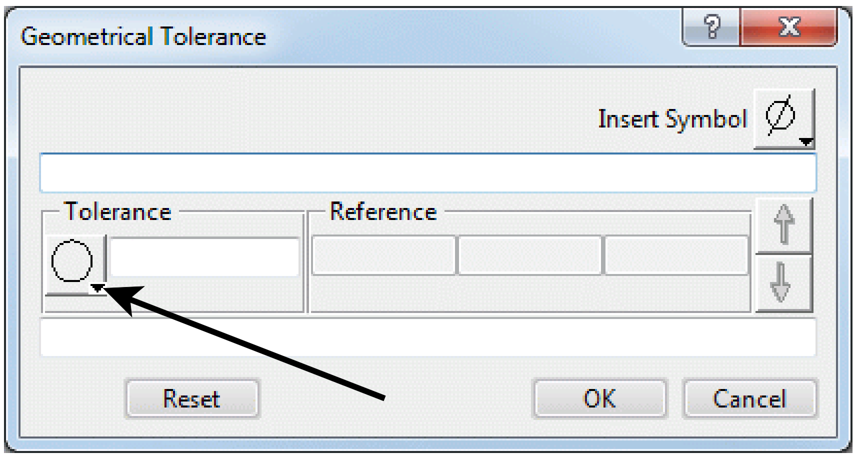
Manually

Now, you will create the same geometrical tolerances manually.

Open the original 3DTA - Flatness document again.

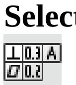
Select the Geometrical Tolerance icon.  It is located in the *Annotation* section under the Datum Feature icon. Nothing will happen until an element is selected.

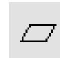
Select the top face of the part. The *Geometrical Tolerance* window appears. Text can be entered above and below the feature control frame, values can be added for the *Tolerance*, and datums can be added in the *Reference* fields. In addition, a *Tools Palette* toolbar appears with propagation options. These were discussed previously.

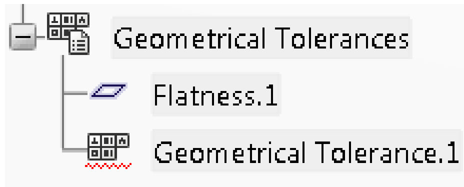


Select the black arrow on the symbol icon as shown above. More tolerancing options appear. This method does not filter out inappropriate selections.

Select Flatness, then enter 0.03 in the *Tolerance* field and click *OK*.  The tolerance appears. Essentially, it is identical to the tolerance created previously with the Tolerancing Advisor. The only difference is that a red, squiggly line appeared beneath the tolerance until *OK* was selected. This is the symbol used for non-semantic annotations. Non-semantic means that CATIA considers them invalid due to either syntax or associativity. Once creation of the tolerance was finalized, the red, squiggly lines were removed because CATIA saw the tolerance as valid. The red, squiggly lines can be turned off in the *Preferences*.

Select the Geometrical Tolerance icon again, then select the bottom face of the part.  The *Geometrical Tolerance* window appears.

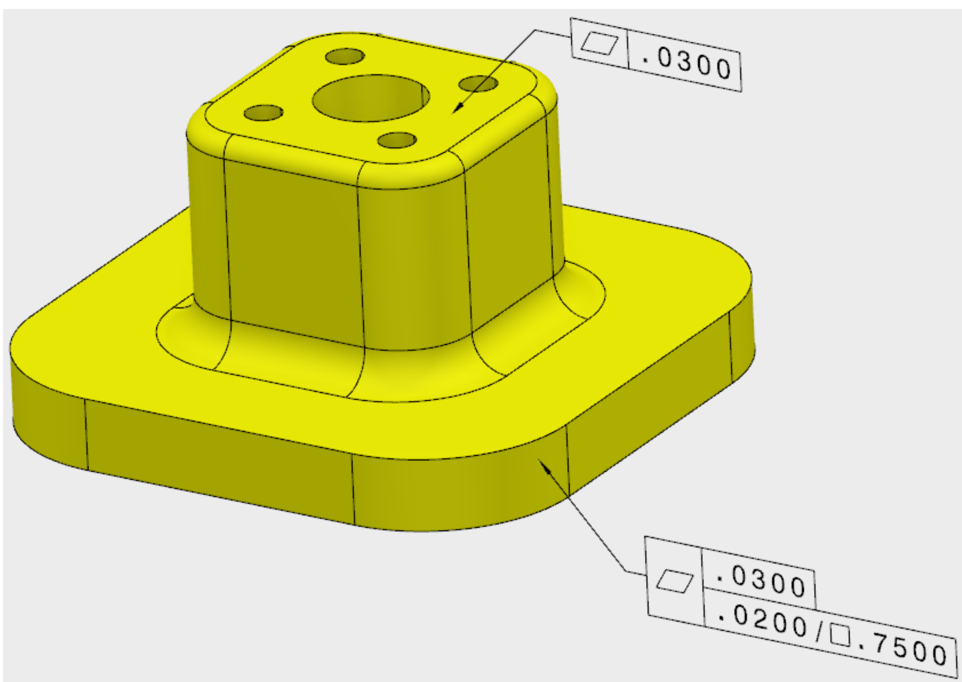
Change the symbol to Flatness with a value of 0.03.  Before selecting *OK*, look at the tolerance in the graphical area. It has a red, squiggly line beneath it. Look in the specification tree as well. It is referred to as a *Geometrical Tolerance* instead of a *Flatness*. There is also a red, squiggly line beneath it in the tree to denote it as non-semantic.



Select *OK*. The tolerance is now referred to as a *Flatness* in the tree.

Double-select on the new tolerance, then select the *Unit Basis* tab from the *Geometrical Specification* window. This is the same window that appears when using the Tolerancing Advisor.

Turn on the *Applied on Unit Basis* and *Refinement Tolerance* options, change the *Numerical value* to 0.02 and *Length 1* to 0.75, then select *OK*. This tolerance is now identical to the tolerance that was created with the Tolerancing Advisor.



Close the document.

Straightness

Straightness tolerances can be applied to surface elements or to the axis or center plane of features of size.

If applied to a surface, it controls how straight a line element of the surface must be in order to meet the design requirements. All line elements of the surface have to exist within the tolerance zone specified by two parallel lines that are separated by the tolerance value.

If applied to an axis, or centerline, of a cylindrical feature of size, it controls the straightness of the axis. The axis must exist within the tolerance zone specified by a cylinder whose diameter is equal to the tolerance value.

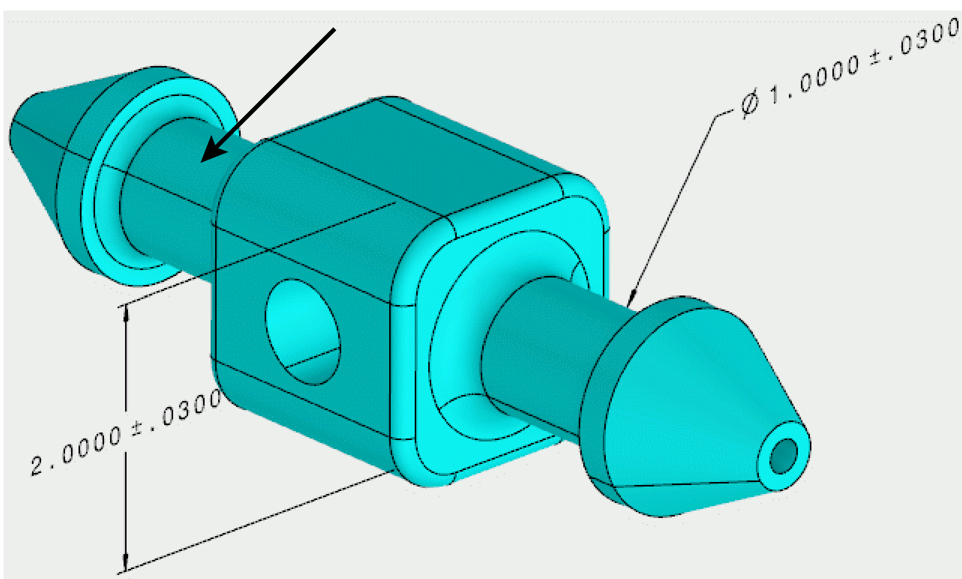
If applied to a center plane, it is controlled similar to a surface. Every line element of the plane must exist within the tolerance zone specified by two parallel planes that are separated by the tolerance value.

Open the Straightness document. Two views and two dimensions already exist.

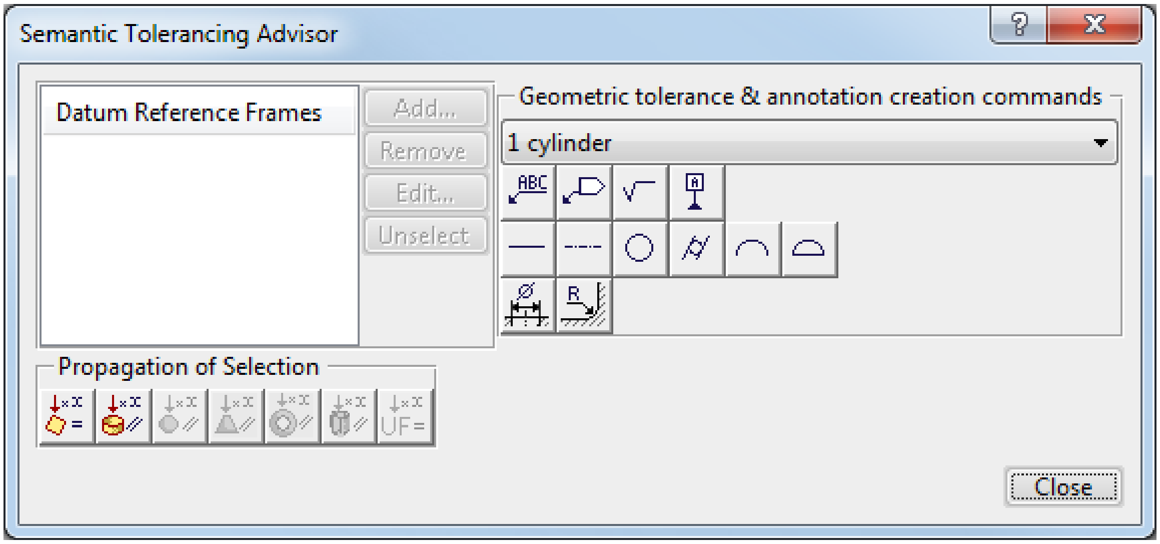
Tolerancing Advisor

As stated before, the Tolerancing Advisor ensures that only valid geometrical tolerances are created.

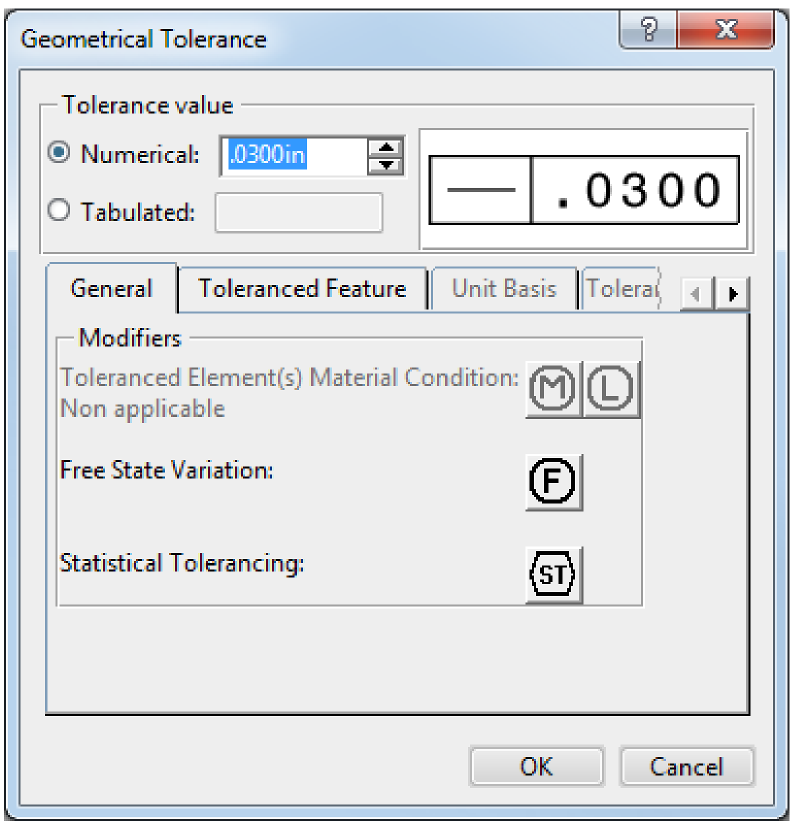
Select the Tolerancing Advisor icon, then select the cylindrical surface indicated below.  You will create a straightness tolerance for this surface. It is not a feature of size, so the tolerance will be applied to the line elements of the surface, not to its axis, or centerline.



The *Semantic Tolerancing Advisor* window expands, and the options that were not valid for the current selection have been filtered away.

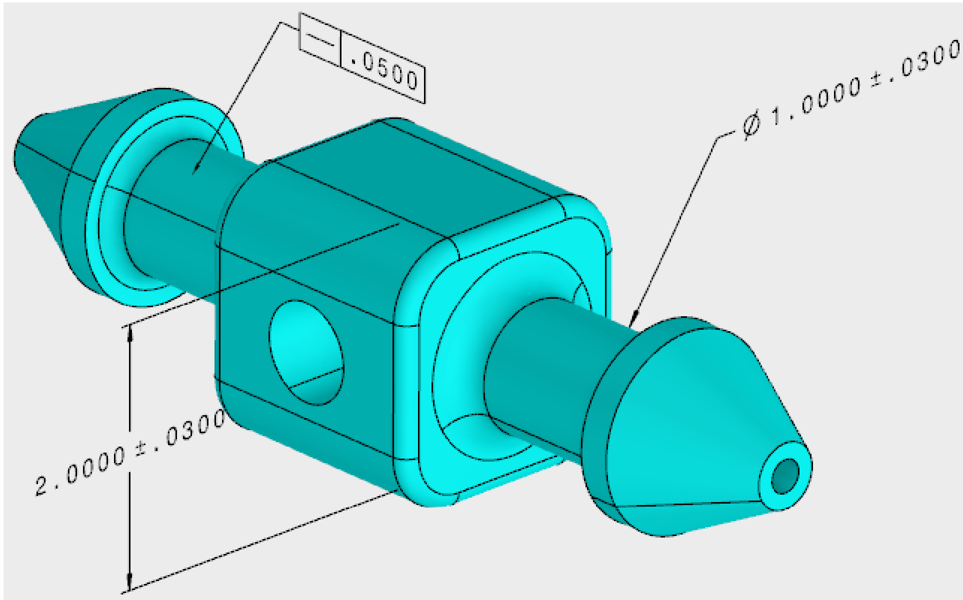


Select the Straightness Specification icon. The *Geometrical Specification* window appears.

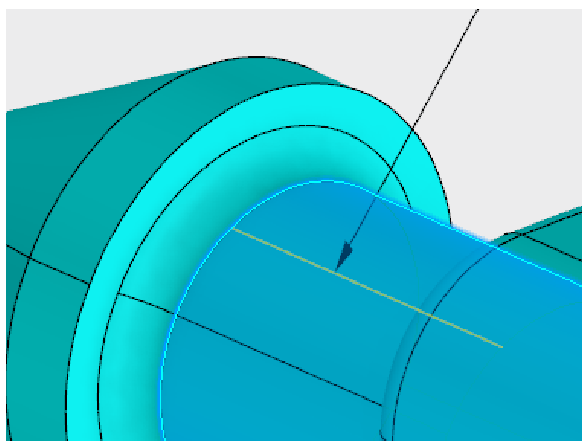


Change the *Numerical* value to 0.05 and select *OK*. The tolerance appears, and the Tolerancing Advisor remains active.

Select *Close*, then position the tolerance as shown below.



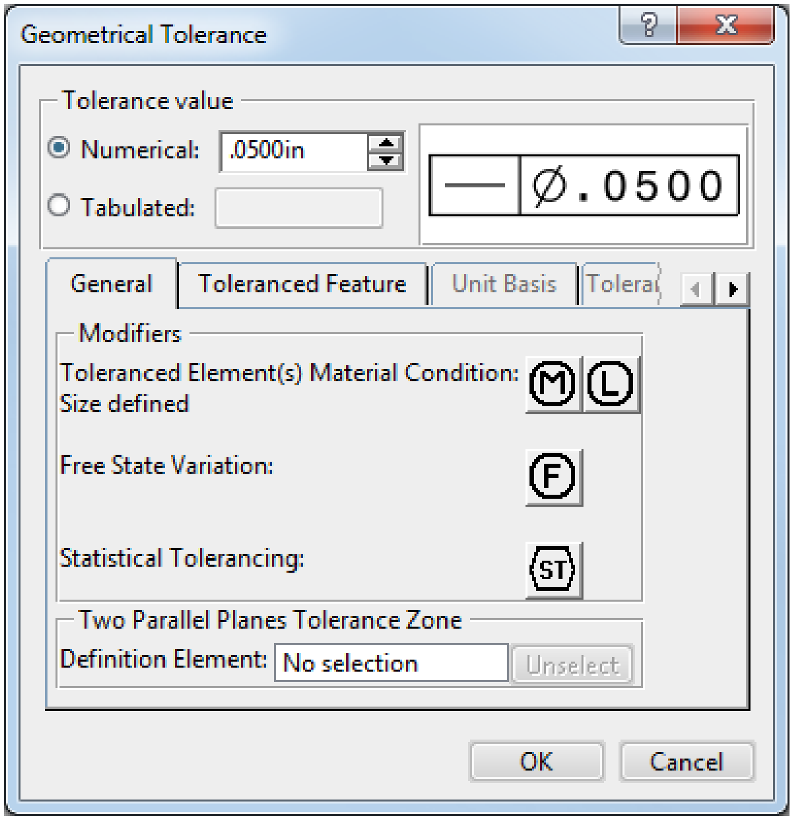
Select the tolerance, then press and hold the first mouse button on the yellow diamond. Two, yellow lines appear. These signify the paths that the arrowhead can be moved along for the current specification.



Select the Tolerancing Advisor icon, then select the 1.0000 dimension.  This time, you will select an existing dimension and add the straightness specification to the feature of size.

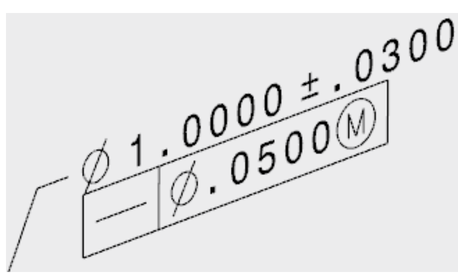
Select the Axis Straightness Specification icon. The *Geometrical Specification* window appears.

Change the *Numerical* value to 0.05. The diameter symbol automatically appears in the feature control frame since CATIA knows that it is a cylindrical tolerance zone. Also, the material condition icons are now available.

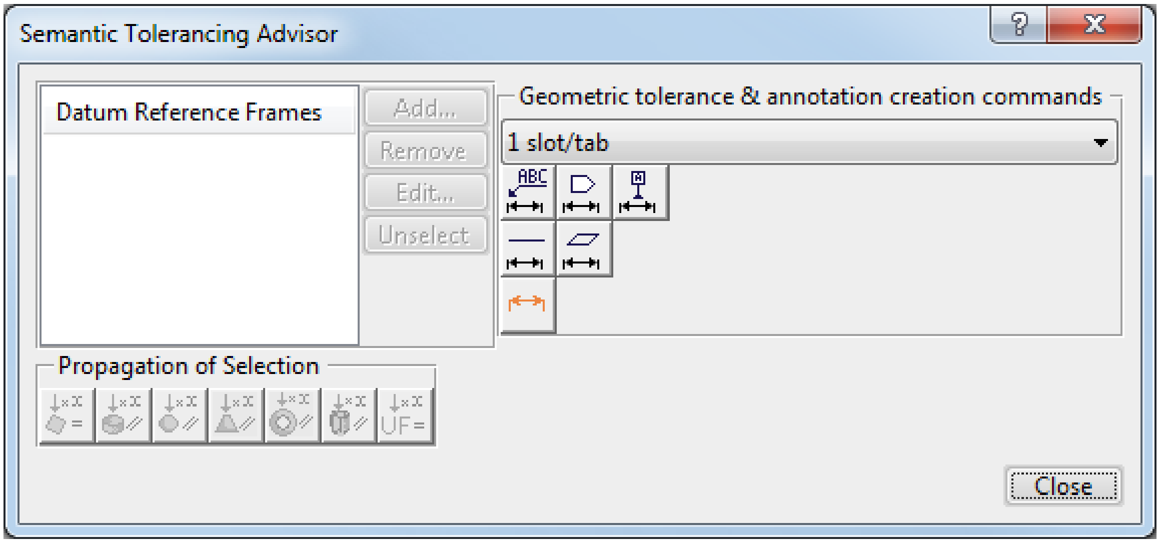


Select the Maximum Material Condition icon.  This allows for extra tolerance while still ensuring the function of assembly.

Select *OK*, then select *Close*. The straightness tolerance appears beneath the dimensional tolerance and has a positional link to it. When the dimension moves, the tolerance will move with it. The straightness tolerance also exists in the same view as the dimension.

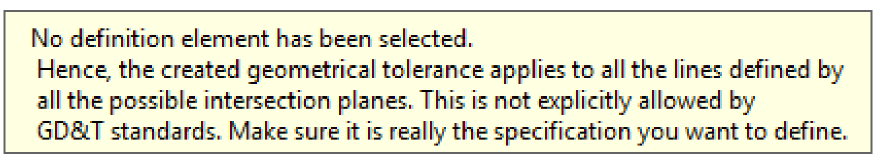


Select the Tolerancing Advisor icon, then select the 2.0000 dimension. Selecting this dimension is similar to selecting two parallel faces. Therefore, it is referred to as a *Tab/slot*.



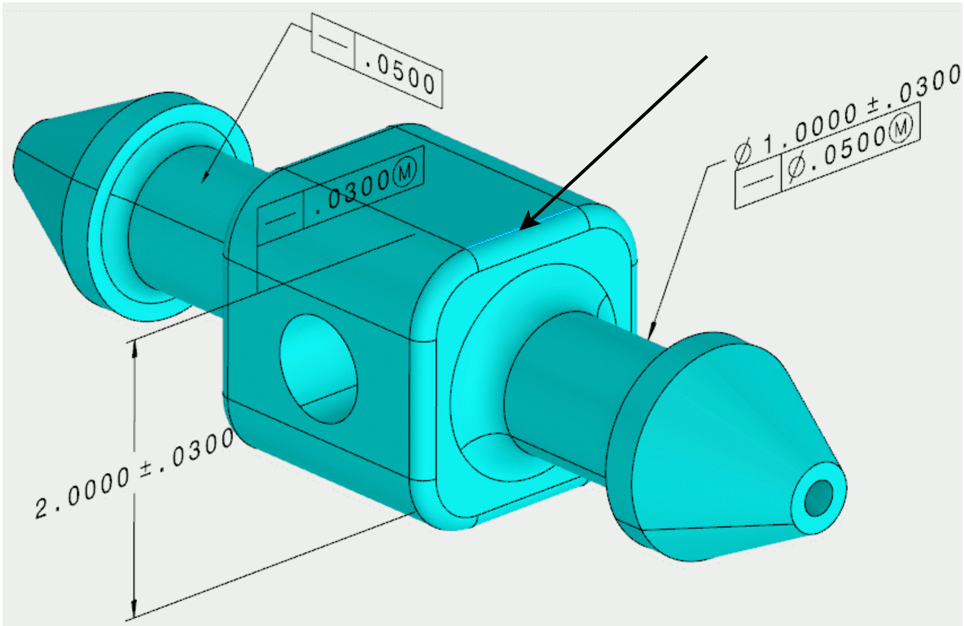
Select the Straightness Specification icon. The *Geometrical Specification* window appears.

Change the *Numerical* value to 0.03 and select the Maximum Material Condition icon, then click *OK*. A small message appears in the lower-right corner of the CATIA window because no direction has been specified.


No definition element has been selected.
Hence, the created geometrical tolerance applies to all the lines defined by all the possible intersection planes. This is not explicitly allowed by GD&T standards. Make sure it is really the specification you want to define.

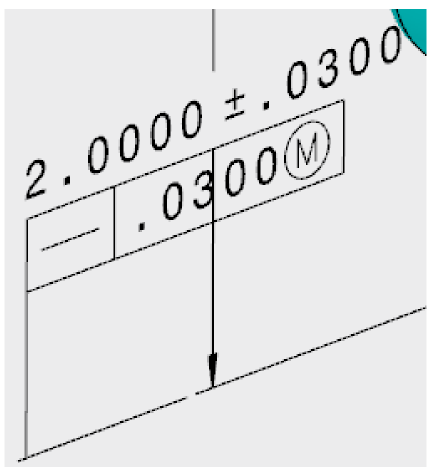
Select *Close*, then double-click on the straightness tolerance just created. A direction for the tolerance zone must be defined since it is being applied to a plane.

Select in the *Definition Element* field and select the edge shown below.



Select *OK.* Normally, the tolerance will be located with the dimension.

Position the tolerance so that it is beneath the dimension.

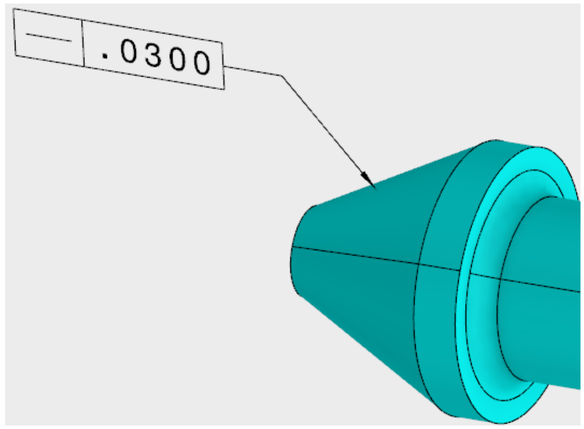


Select the tolerance, then press the third mouse button on the yellow diamond at the end of the leader and select *Remove Leader/Extremity* from the contextual menu. The leader is removed.

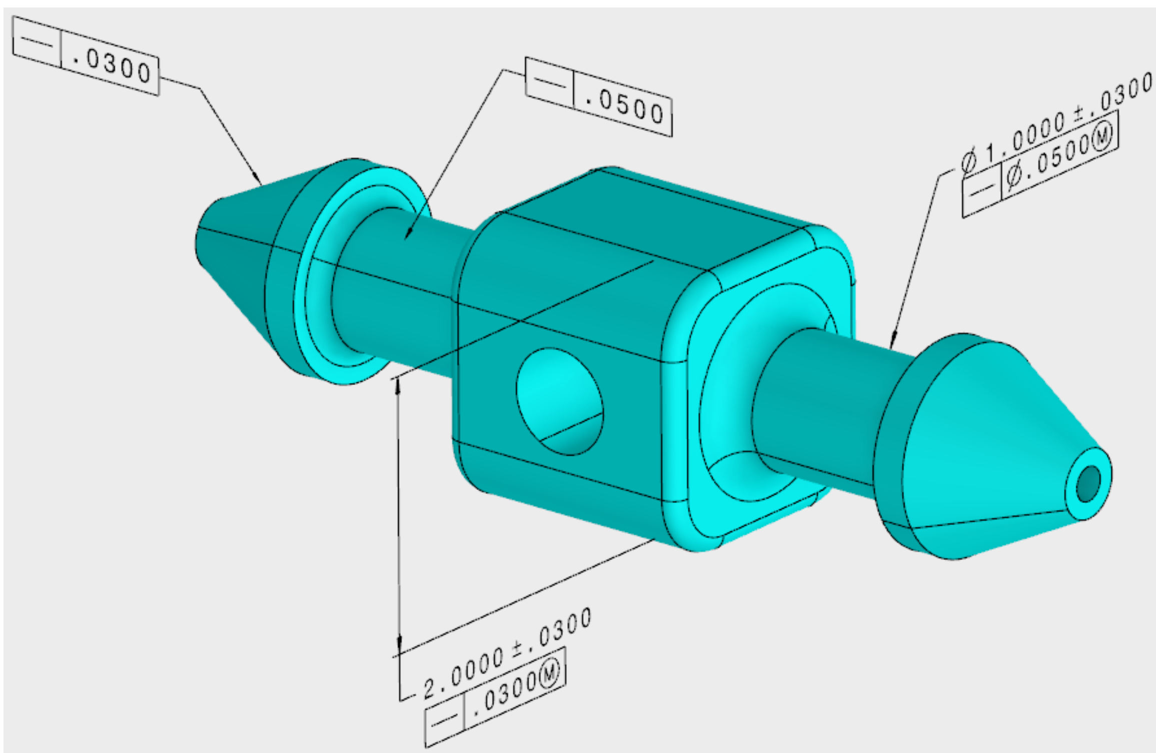
Change the dimension's leader to Two Parts, then move the dimension below its bottom extension line.  The tolerance follows.

A straightness tolerance can also be applied to a conical shape.

Using the Tolerancing Advisor, create the straightness tolerance shown below. Instead of a diameter or radius option in the *Semantic Tolerancing Advisor* window, the Cone Angle Creation icon appears.



Your model should similar to this.



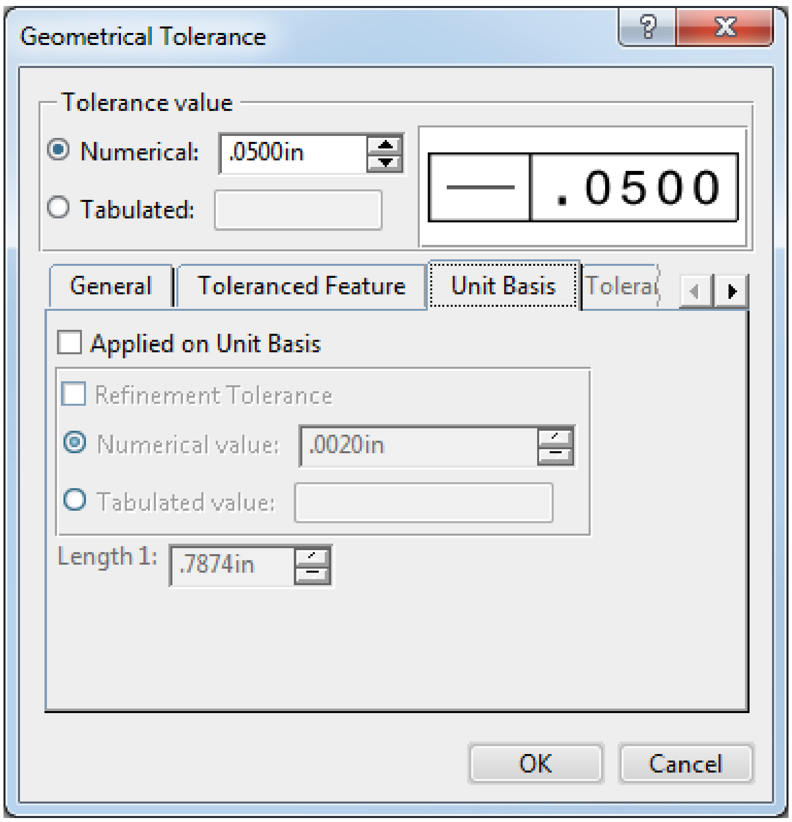
Save and close the document.

Open the 3DTA - Straightness - Unit Basis document. You will define some straightness tolerances, then refine them with unit basis tolerances.

Select the Tolerancing Advisor icon.  The *Semantic Tolerancing Advisor* window appears.

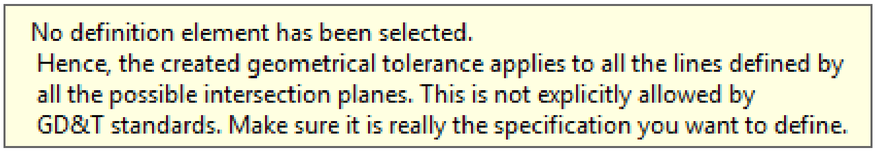
Select the top face of the part, then choose the Straightness Specification icon. The *Geometrical Specification* window appears. A view was automatically created since there were none beforehand.

Change the *Numerical* value to 0.05 and select the *Unit Basis* tab. The options here are very similar to the flatness options, except that there is only one length definition available.



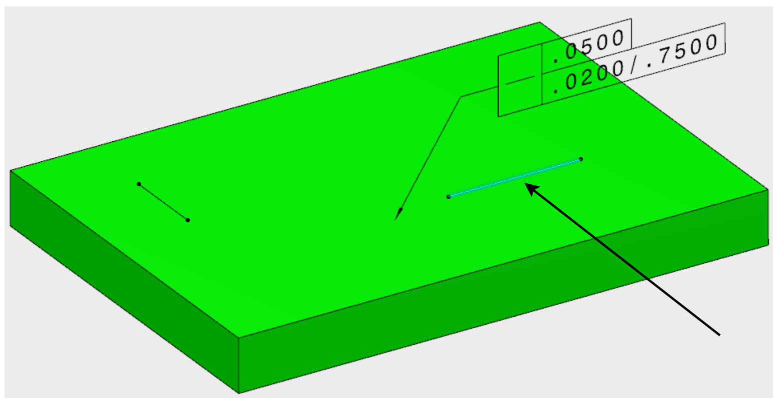
The *Unit Basis* tab specifies a refinement tolerance for a smaller length of the plane along with a total variation, or it can be used by itself. In this case, you will specify a refinement stating that for a 0.75 length, the maximum variation can only be 0.02.

Select the *Applied on Unit Basis* and *Refinement Tolerance* options, change the *Numerical value* of the refinement to 0.02 and *Length 1* to 0.75, then select *OK*. A small message appears in the lower-right corner of the CATIA window. It is the same message as before.

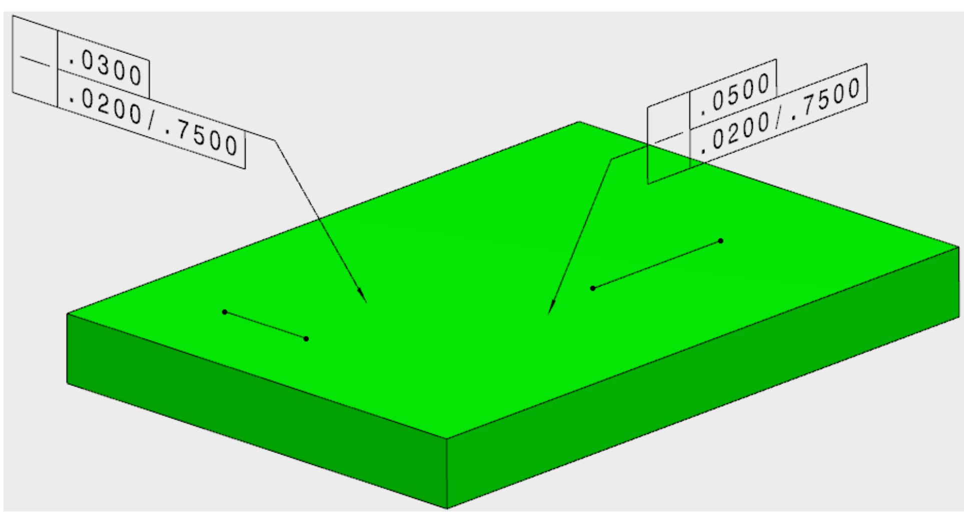

No definition element has been selected.
Hence, the created geometrical tolerance applies to all the lines defined by all the possible intersection planes. This is not explicitly allowed by GD&T standards. Make sure it is really the specification you want to define.

Select *Close*, then double-click on the tolerance just created. When using a plane, a tolerance direction for the straightness must be specified.

Under the *General* tab, select in the *Definition Element* field and choose the line indicated below, then select *OK* and position the tolerance as shown here.



Activate the *Side View*, then create another straightness tolerance as shown below using the other line as the direction. 



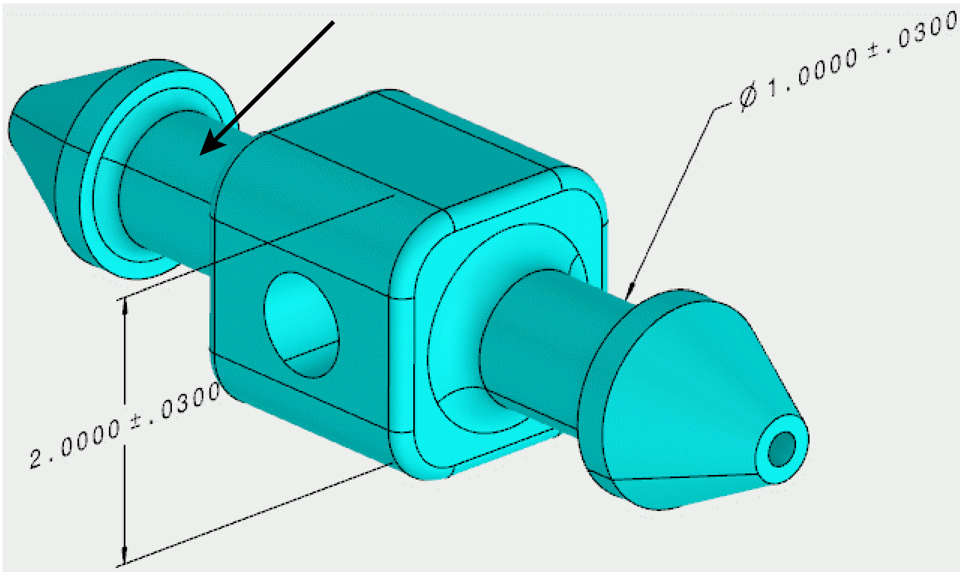
Save and close the document.

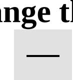
Manually

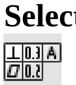
Now, you will manually create the same geometrical tolerances.

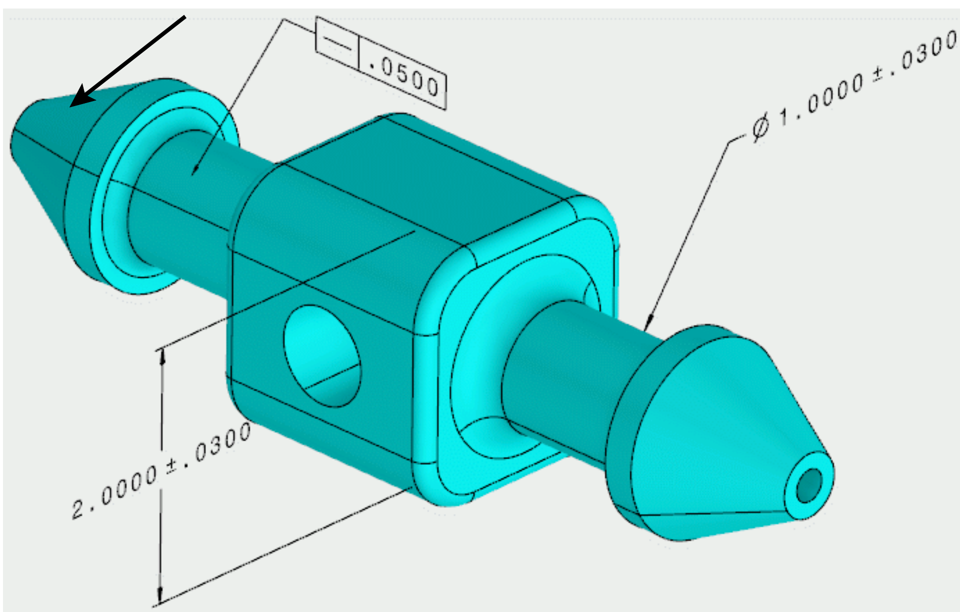
Open the original 3DTA - Straightness document again, then select the Geometrical Tolerance icon.  Nothing will happen until an element is selected.

Select the cylindrical surface shown below. The *Geometrical Tolerance* window appears.



Change the specification to Straightness, then enter 0.05 for the *Tolerance* and click *OK*.  This tolerance is identical to the one created with the Tolerancing Advisor.

Select the Geometrical Tolerance icon, then select the conical surface shown below.  The *Geometrical Tolerance* window appears.

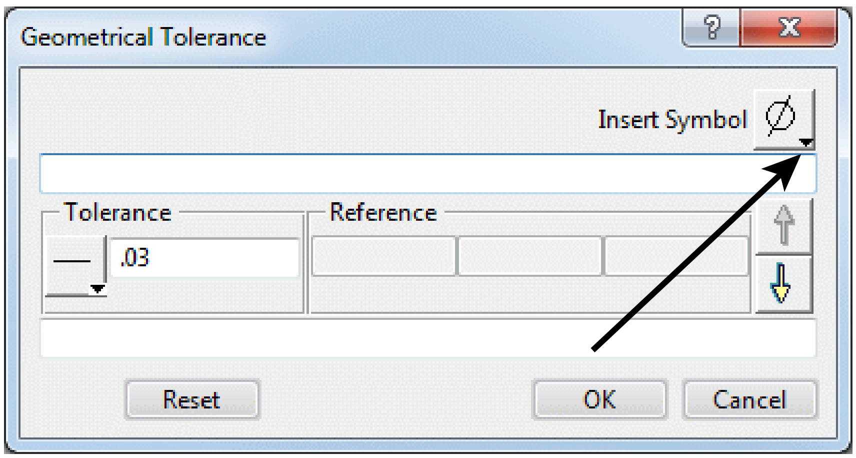


Change the symbol to Straightness with a value of 0.03 and select *OK*. The tolerance appears.

Select the Geometrical Tolerance icon, then select the 1.0000 dimension.  The *Geometrical Tolerance* window appears.

Change the symbol to Straightness. This time, a diameter symbol will be included with the value since CATIA will not automatically add it like the Tolerancing Advisor does.

In the *Tolerance* field, select to the left of the value and click the black arrow on the *Insert Symbol* icon as shown below. A menu with various symbols appears.



Select the Diameter symbol.  The diameter symbol is inserted before the value.

Change the value to 0.05, then select the black arrow on the *Insert Symbol* icon and choose the Maximum Material Condition symbol.  The symbol is inserted after the value.

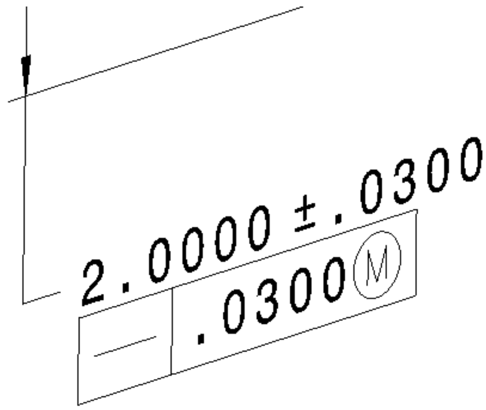
Select *OK*. The tolerance appears, but it has an exclamation point in the specification tree. The Geometrical Tolerance option does not currently allow an axis straightness tolerance to be defined. As a result, it cannot be positioned.

Select the Geometrical Tolerance icon, then select the 2.0000 dimension.  The *Geometrical Tolerance* window appears.

Change the symbol to Straightness with a value of 0.03.

Add the Maximum Material Condition symbol after the value in the *Tolerance* field and select *OK.* The tolerance appears.

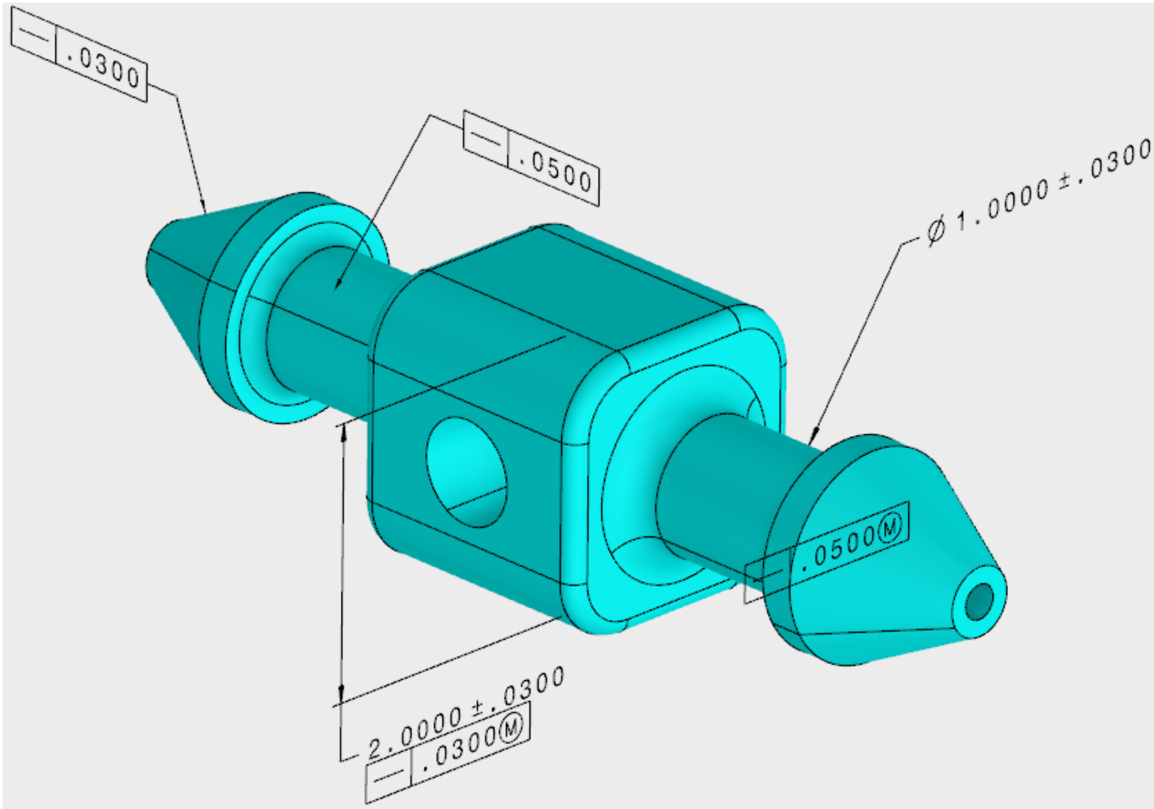
Change the leader of the 2.000 dimension to Two Parts, then position the tolerance beneath it as shown here.



There are a couple of things to note here: 1) an axis straightness tolerance was unable to be defined, and 2) straightness on the center plane did not require a tolerance direction.

Double-select the last straightness tolerance. The *Geometrical Specification* window appears.

Select in the *Definition Element* field at the bottom of the window and pick the edge shown below, then select *OK.* In order to define the tolerance direction, the tolerance must be edited after it is created.



Close the document.