

TABLE OF CONTENTS

Introduction 1

 Functional Tolerancing and Annotation 2

 Pull-down Menus 3

 Insert 3

 Functional Tolerancing & Annotation Workbench 4

 Bottom Toolbar 5

 Top Toolbars 6

 GD&T Review 8

 Geometric Characteristic Symbols 8

 Modifying Symbols 9

 Other Symbols 10

Views 11

 View Creation 11

 Offset Section View/Section Cut 20

 Aligned Section View/Section Cut 23

 Orientation 26

 Transfer 30

 Using Axis Systems 32

 Properties 37

 Changing Support 39

Datums 41

 Planar Datums 41

 Tolerancing Advisor 42

 Datum Reference Frames 48

 Manually 58

 Positioning a Datum 63

 Datum Targets 65

 Points 65

 Tolerancing Advisor 65

 Manually 69

 Lines 70

 Tolerancing Advisor 70

 Manually 73

 Areas 74

 Tolerancing Advisor 74

 Manually 83

 Datum Axes and Center Planes 89

 Tolerancing Advisor 89

 Manually 98

Dimensions	101
Creating Dimensions	101
Length/Distance	101
Angle	110
Radius	112
Diameter	114
Coordinate	116
Cumulated	117
Stacked	119
Dimensions for curves	120
Generative Dimensions	122
Setup Parameters	124
Dimension Lines	124
Tolerance	127
Numerical Display	130
Modifying Dimensions	133
Top Toolbar	133
Pull-Down Menu Tools, Options	134
Properties	137
Positioning	152
Tolerancing Advisor	156
Creating Dimensions	156
Modifying Dimensions	166
Propagation Selection	168
Propagation options	168
Geometrical Tolerancing	171
Form Controls	171
Flatness	171
Tolerancing Advisor	172
Manually	177
Straightness	179
Tolerancing Advisor	179
Manually	188
Circularity	193
Tolerancing Advisor	193
Manually	197
Cylindricity	200
Tolerancing Advisor	200
Manually	203

Orientation Controls	205
Perpendicularity	205
Tolerancing Advisor	205
Manually	212
Angularity	215
Tolerancing Advisor	215
Manually	217
Parallelism	219
Tolerancing Advisor	219
Manually	223
Location Controls	225
Position	225
Tolerancing Advisor	225
Manually	244
Concentricity	256
Tolerancing Advisor	256
Manually	258
Symmetry	259
Tolerancing Advisor	259
Manually	262
Runout Controls	263
Circular Runout	263
Tolerancing Advisor	263
Manually	267
Total Runout	268
Tolerancing Advisor	268
Manually	271
Profile Controls	273
Profile of a Surface	273
Tolerancing Advisor	273
Manually	279
Profile of a Line	280
Tolerancing Advisor	280
Manually	285
Unilateral or Unequal Bilateral	287
Unilateral - Outward	287
Unilateral - Inward	287
Bilateral - Unequal	287
Modify	288
Changing Datum Reference Frame	288
Adding a Geometrical Tolerance to a Datum	290
Grouping	291
Positioning	293
Basic Dimensions	295

Annotations	305
Creating Text	305
Modifying Text	310
Text Properties Toolbar	310
Font properties	310
Justification	312
Anchor point	312
Frame	313
Insert Symbol	314
Properties	316
Adding a Leader	328
Links	332
Orientation Link	332
Positional Link	334
Attribute Link	335
Query Object Links	336
Isolate Text	337
Flag Notes	338
Roughness Symbol	341
Weld Symbols	344
Graphic Properties Toolbar	346
Copy Object Format	350
Tolerancing Advisor	351
Text	351
Flag notes	353
Roughness Symbol	354
Geometry for 3D	357
Restricted Area	357
Construction Geometry Creation	361
Construction Geometry Management	370
Thread Representation Creation	374
Geometry Connection Management	380
Note Object Attribute	389
Reports	393

Visualization	399
List Annotation Set	399
3D Annotation Query	401
Filtering	403
Mirror	406
Clipping Plane	408
Captures	409
Displaying Captures	409
Creating Captures	412
Filter Workbench	413
Active Views and the Clipping Plane	416
Current State	418
Creating the Side Capture	420
Creating the Top Capture	421
Creating the 3D All Capture	422
Creating the 3D None Capture	424
Properties	425
Capture Management	426
Problems	429
Problem #01	429
Problem #02	431
Problem #03	433
Problem #04	436
Appendix A	447
Mechanical Design - FT&A - Tolerancing	447
Mechanical Design - FT&A - Display	448
Mechanical Design - FT&A - Constructed Geometry	450
Mechanical Design - FT&A - Manipulators	451
Mechanical Design - FT&A - Dimension	452
Mechanical Design - FT&A - Annotation	453
Mechanical Design - FT&A - Tolerances	454
Mechanical Design - FT&A - View/Annotation Plane	455
Mechanical Design - FT&A - Administration	456

Introduction

CATIA Version 5 Functional 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

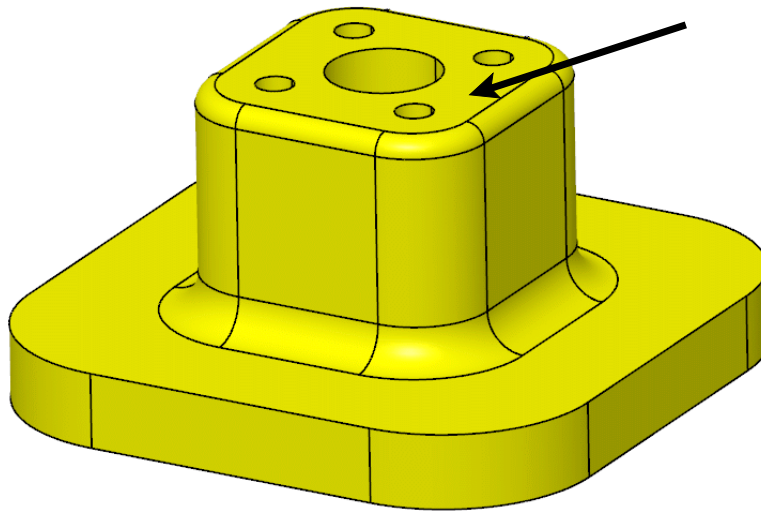
Functional Tolerancing and Annotation

Functional 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. Functional tolerancing and annotation is one set of tools that can help make the transition a reality.

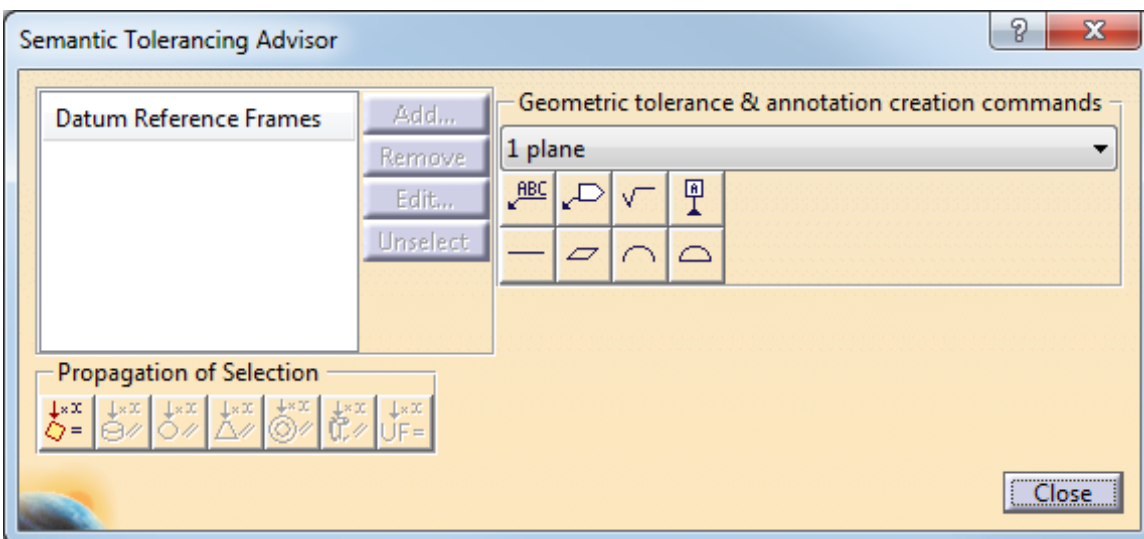
To effectively implement the tools in this course, you must be familiar with the fundamentals of geometric dimensioning and tolerancing (GD&T). It is not the intention of this course to teach GD&T. There is some assistance provided within the functionality of the workbench, but it will still allow you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

Tolerancing Advisor

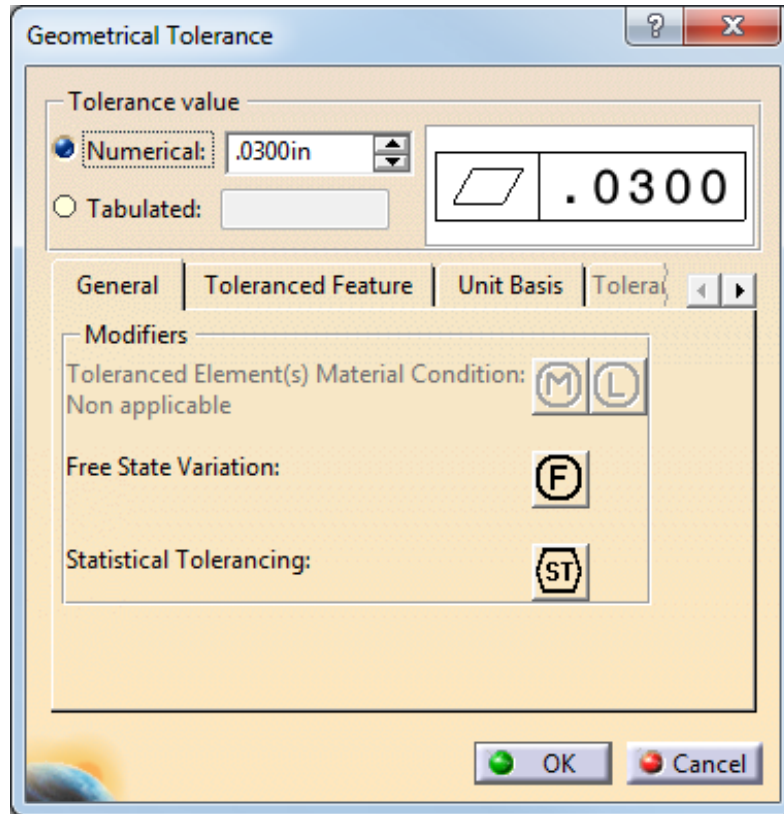
Select the **Tolerancing Advisor** icon, then select the top of the part.  You will create a flatness tolerance for this face.



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.

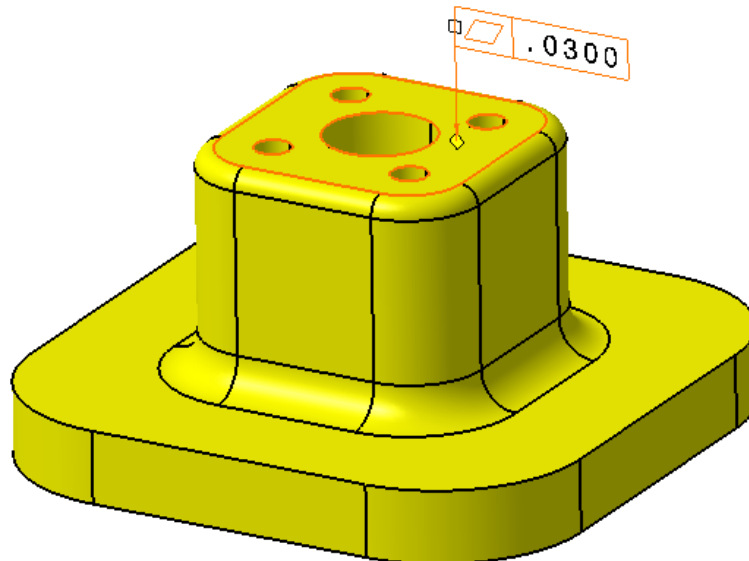


Select the **Flatness Specification** icon.  The *Geometrical Specification* window appears.

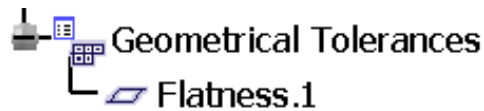


These options will be discussed as they are used throughout the exercises.

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

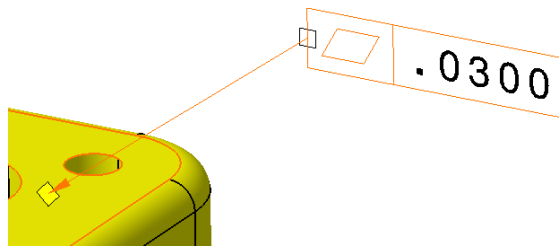


The tolerance appears in the tree as shown below.

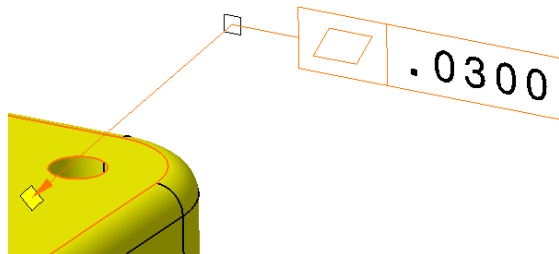


There are many options for working with the leader lines, but they will be covered in more detail when annotations are discussed. For now, you will only move the tolerance and extend the leader.


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.

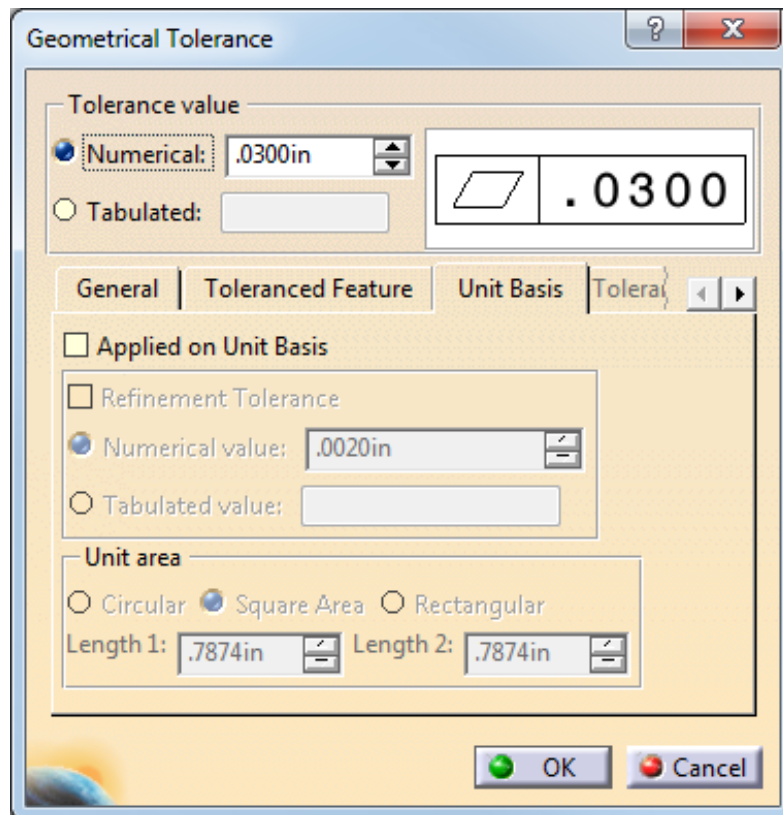


Feel free to move the tolerances to better locations throughout the exercises. Since it is the same procedure every time, it will not be mentioned repeatedly.

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.

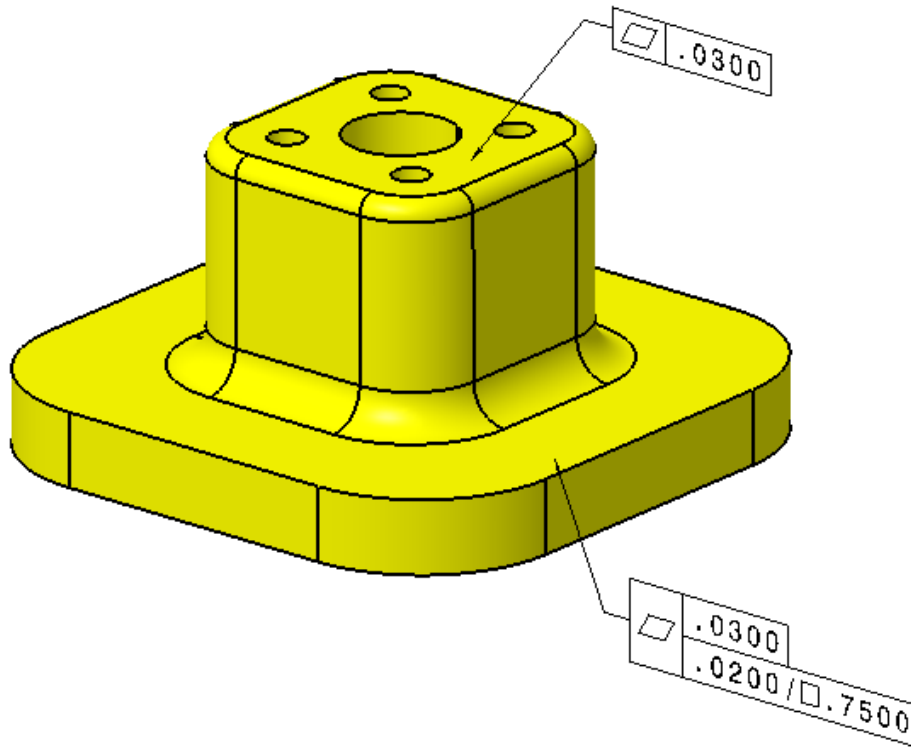
Select the *Unit Basis* tab.



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.

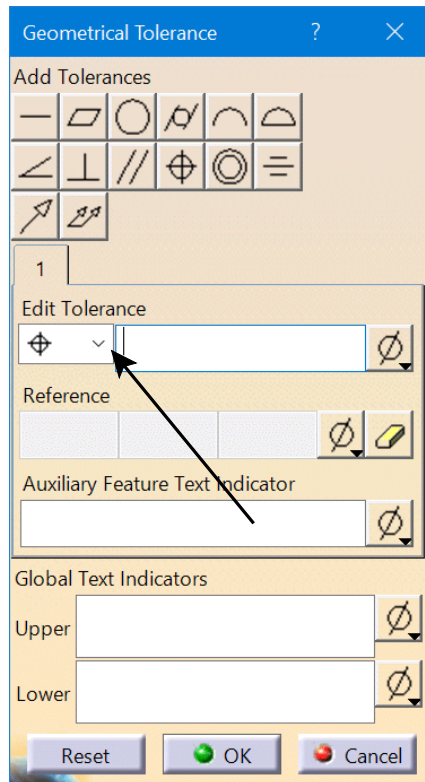
Save and close the document.

Manually


Now, you will create the same geometrical tolerances manually.


Open the original Flatness document again, then select the Geometrical Tolerance 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.

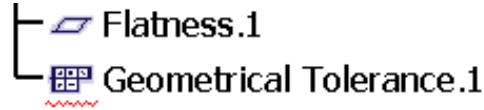


Select the black arrow on the *Edit Tolerance* drop down as shown above. More tolerancing options appear. This method does not filter out inappropriate selections.

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 *Tools, Options* window.

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

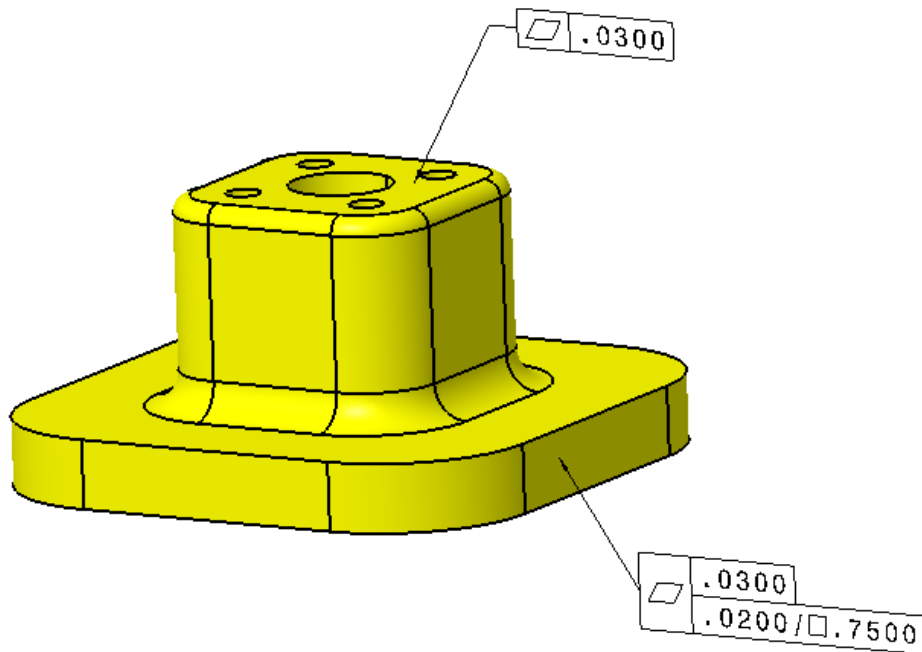
Change the symbol to Flatness with a value of 0.03.  Before selecting *OK*, look at the tolerance in the graphical area. It has a red, squiggly line beneath it. Look in the specification tree as well. It is referred to as a *Geometrical Tolerance* instead of a *Flatness*. There is also a red, squiggly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

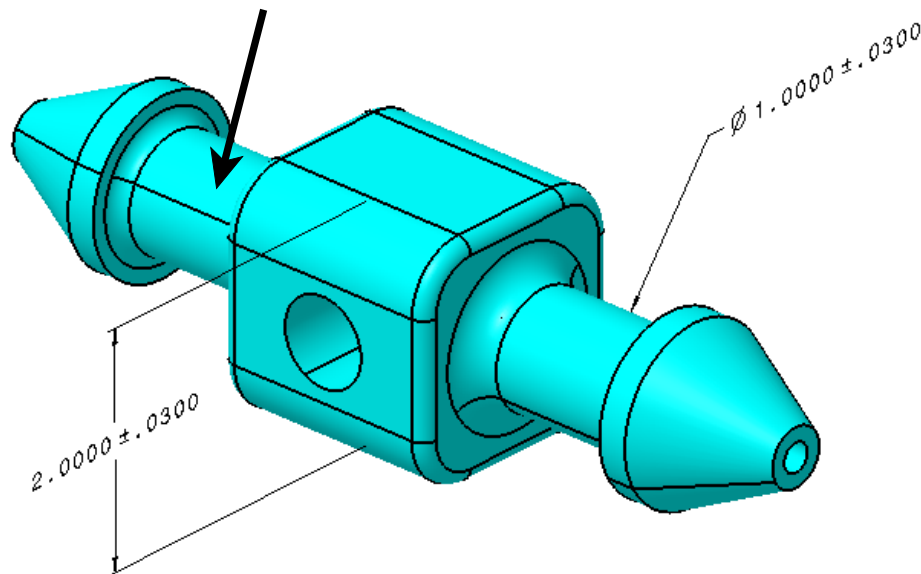
If applied to a center plane, it is controlled similar to a surface. Every line element of the plane must exist within the tolerance zone specified by two parallel planes that are separated by the tolerance value.

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

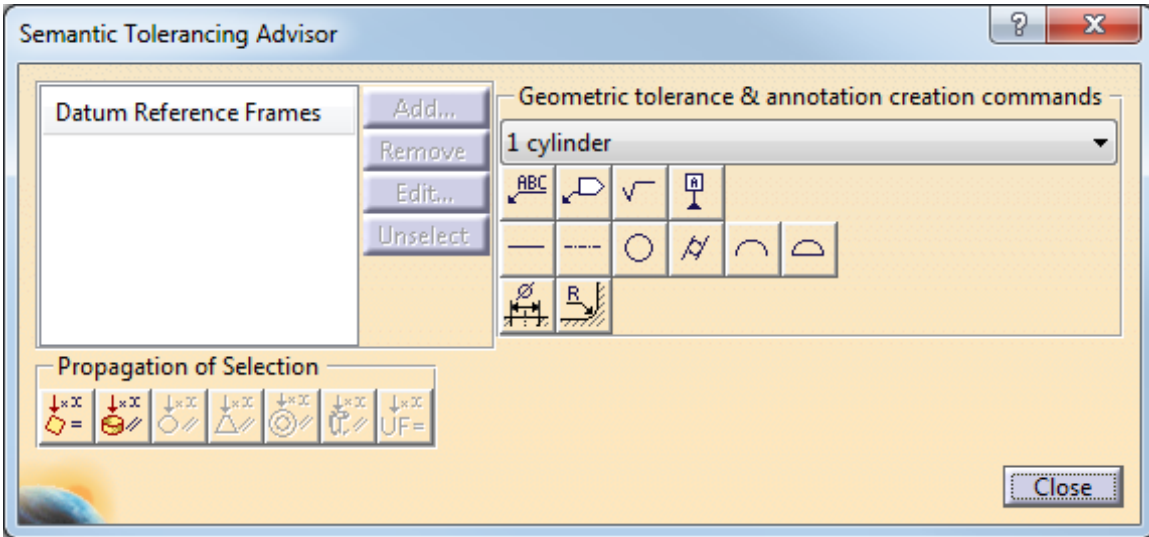
Tolerancing Advisor

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

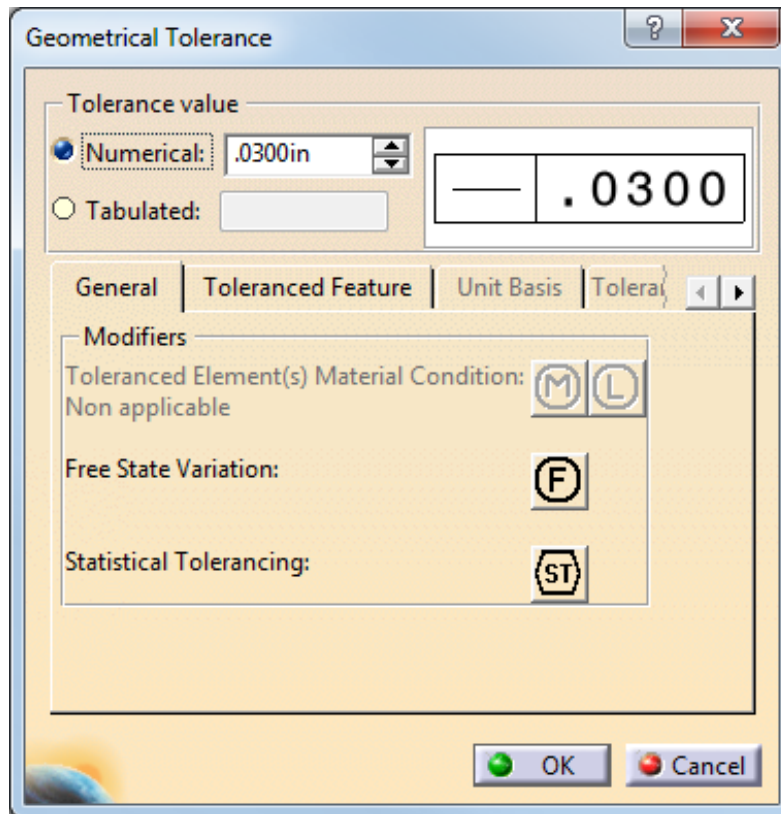
Select the Tolerancing Advisor icon, then select the cylindrical surface indicated below.  You will create a straightness tolerance for this surface. It is not a feature of size, so the tolerance will be applied to the line elements of the surface, not to its axis, or centerline.



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

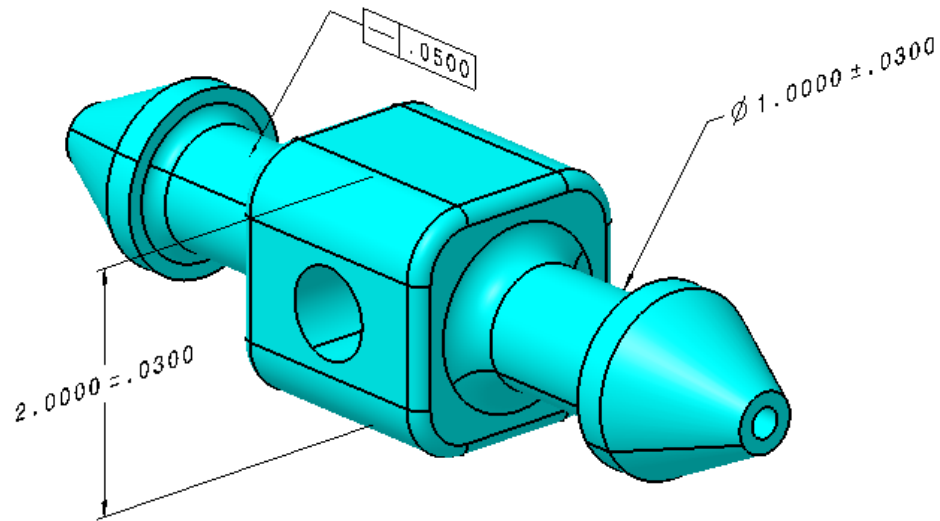


Select the **Straightness Specification** icon.  The *Geometrical Specification* window appears.

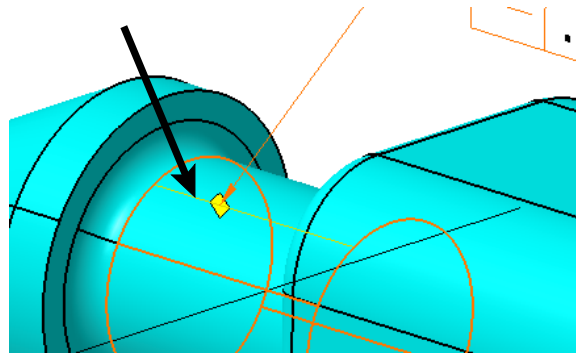



Change the *Numerical* value to **0.05** and select **OK**. The tolerance appears, and the Tolerancing Advisor remains active.


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



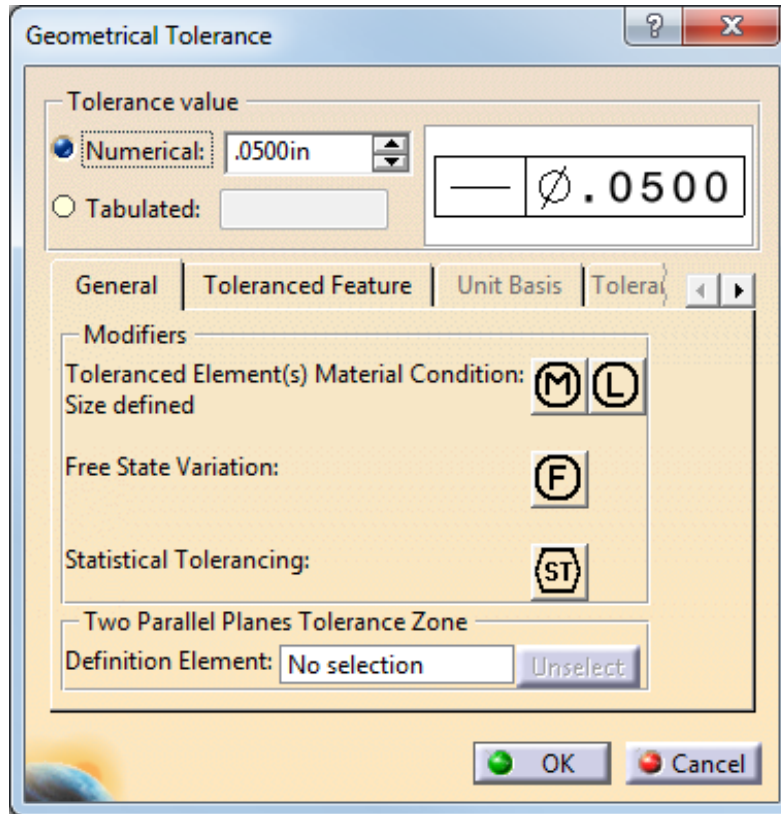
Select the tolerance, then press and hold the first mouse button on the yellow diamond. Two, yellow lines appear. These signify the paths that the arrowhead can be moved along for the current specification.



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.

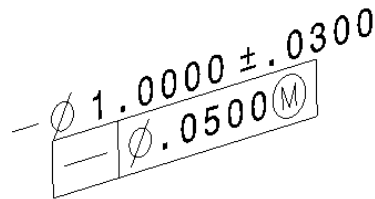
Select the **Axis Straightness Specification** icon.  The *Geometrical Specification* window appears.


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.

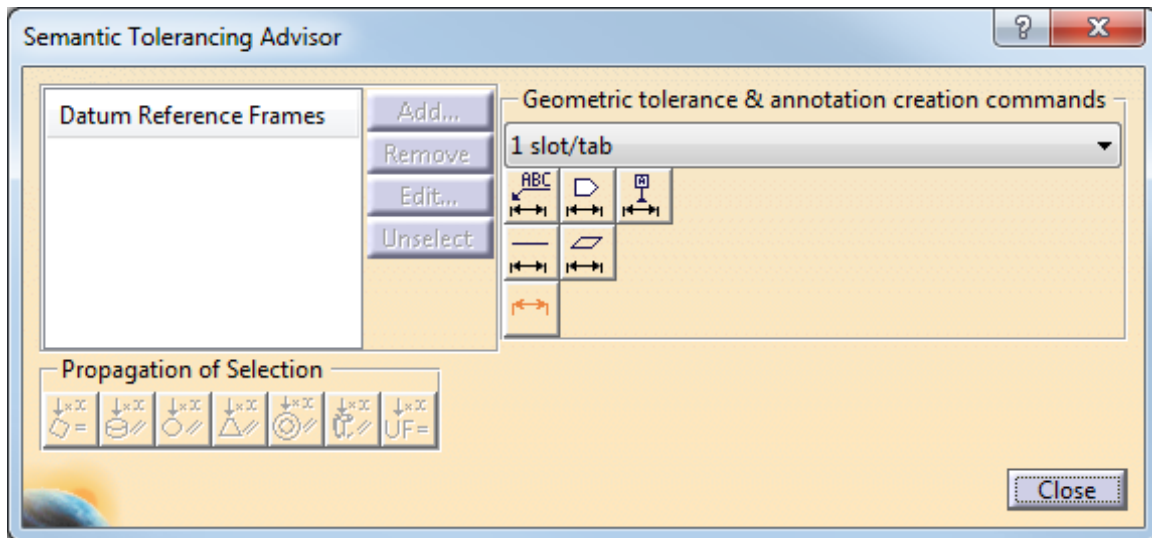


Select the Maximum Material Condition icon. This allows for extra tolerance while still ensuring the function of assembly.


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*.



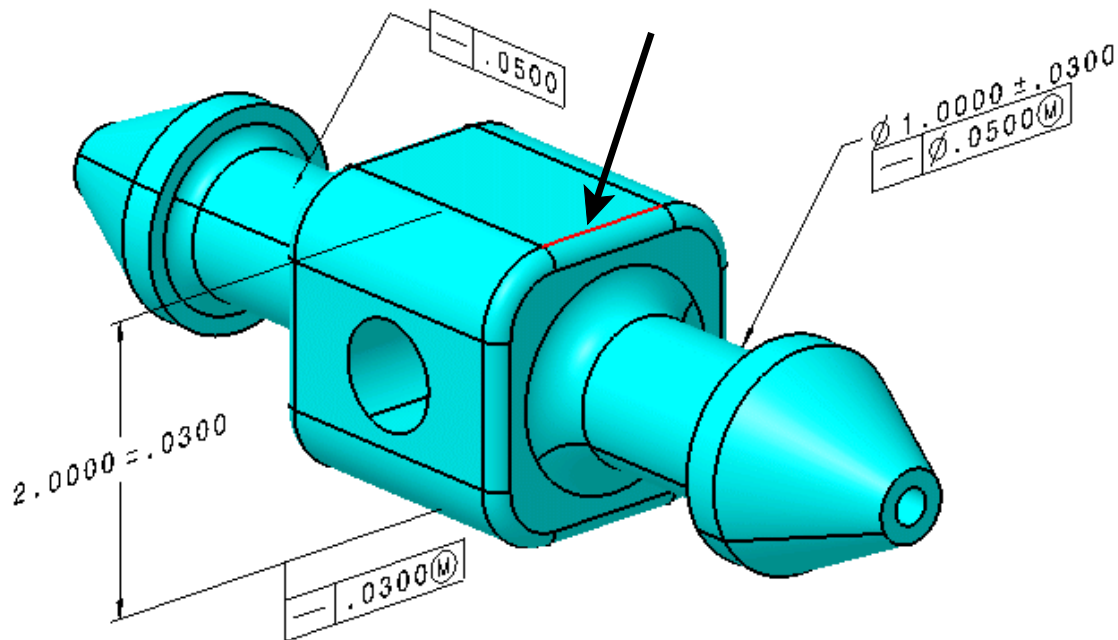
Select the **Straightness Specification** icon.  The *Geometrical Specification* window appears.

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.

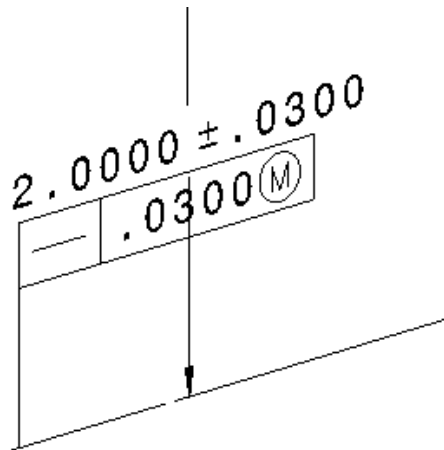
Select **Close**, then **double-click** on the straightness tolerance just created. A direction for the tolerance zone must be defined since it is being applied to a plane.

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.



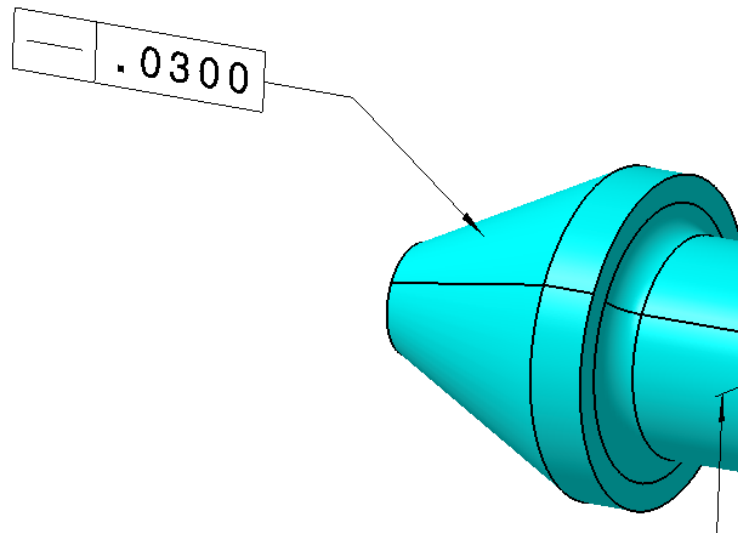
Select the tolerance, then press the third mouse button on the yellow diamond at the end of the leader and select *Remove Leader/Extremity* from the contextual menu. The leader is removed.

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

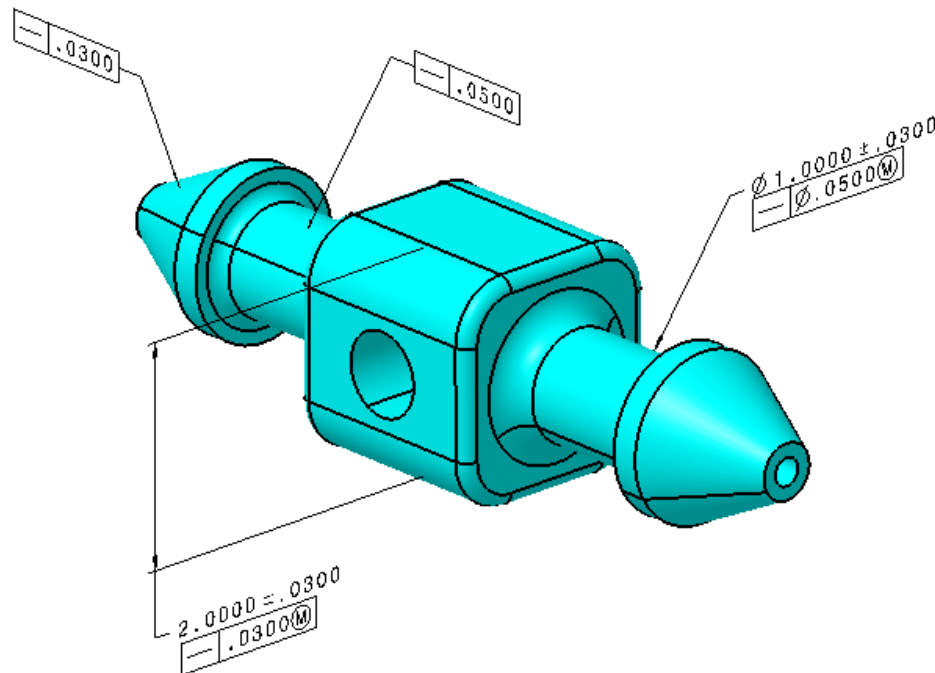
A straightness tolerance can also be applied to a conical shape.

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.



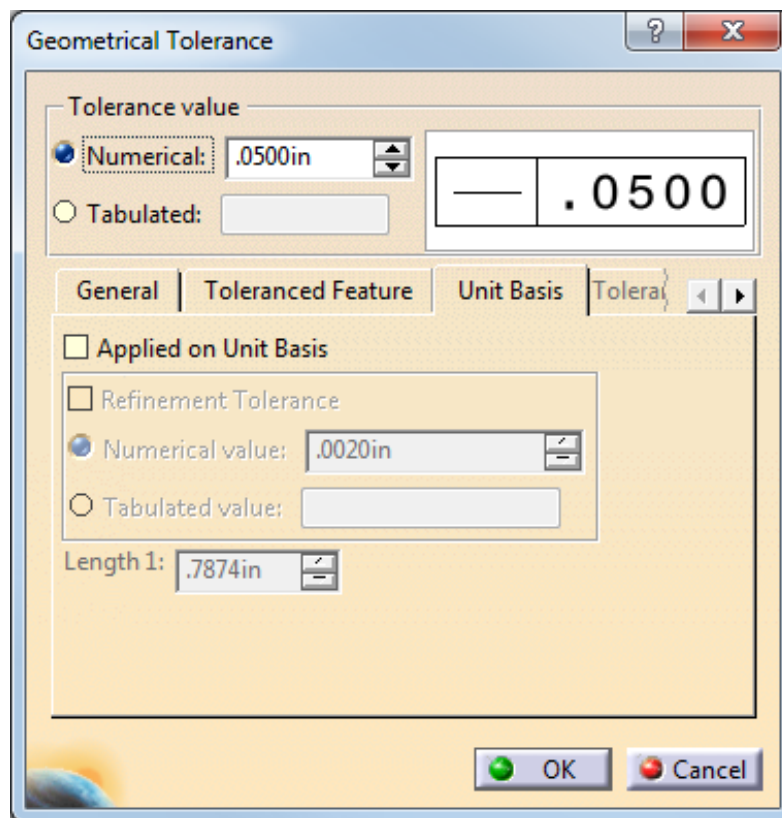
Save and close the document.

Open the 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.



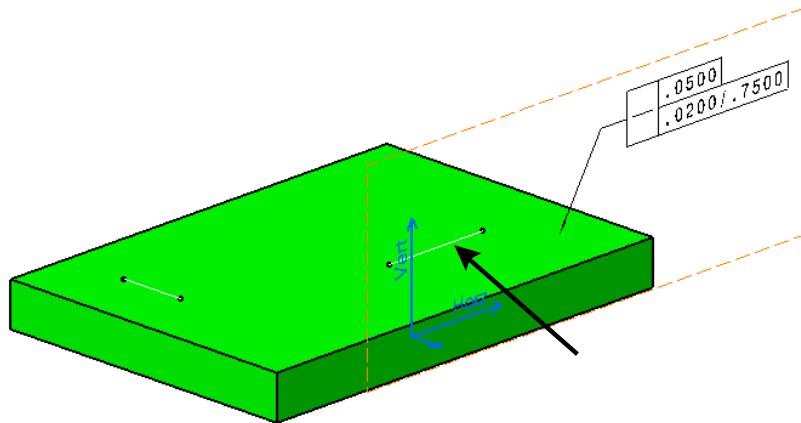
The *Unit Basis* tab specifies a refinement tolerance for a smaller length of the plane along with a total variation, or it can be used by itself. In this case, you will specify a refinement stating that for a 0.75 length, the maximum variation can only be 0.02.


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.

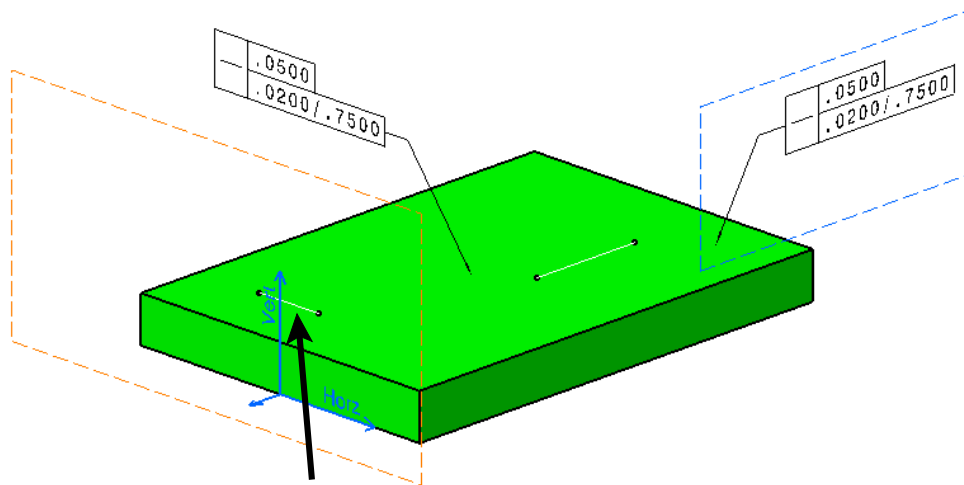
No definition element has been selected.
 Hence, the created geometrical tolerance applies to all the lines defined by all the possible intersection planes. This is not explicitly allowed by GD&T standards. Make sure it is really the specification you want to define.

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.




Create a view normal to the existing view, then create another straightness tolerance as shown below using the other line as the direction. 



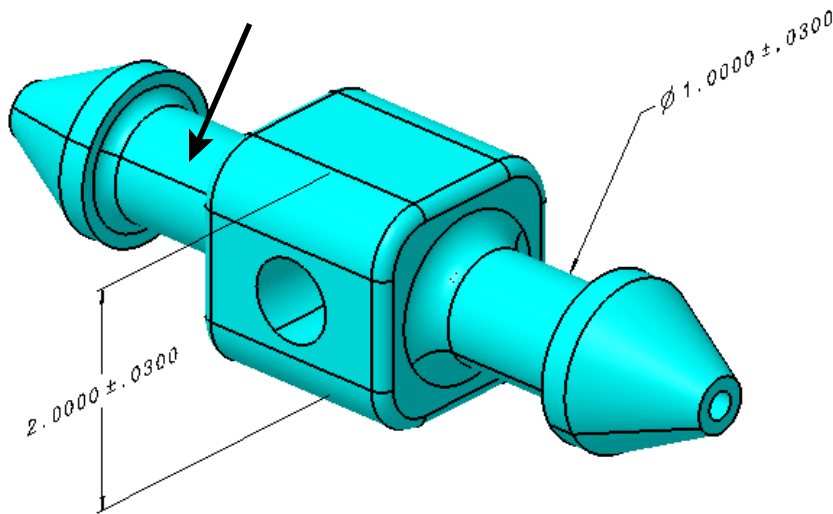
Save and close the document.


Manually

Now, you will manually create the same geometrical tolerances.


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

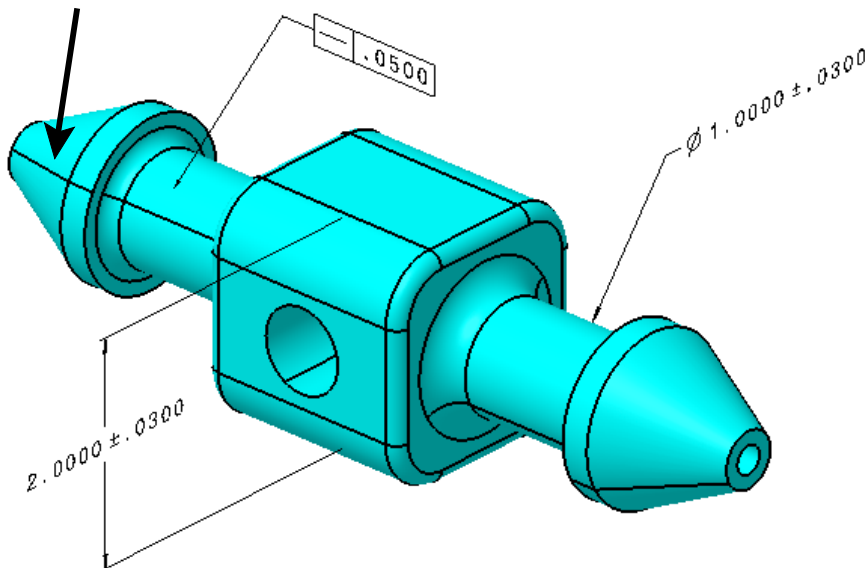
Select the cylindrical surface shown below. The *Geometrical Tolerance* window appears.





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.

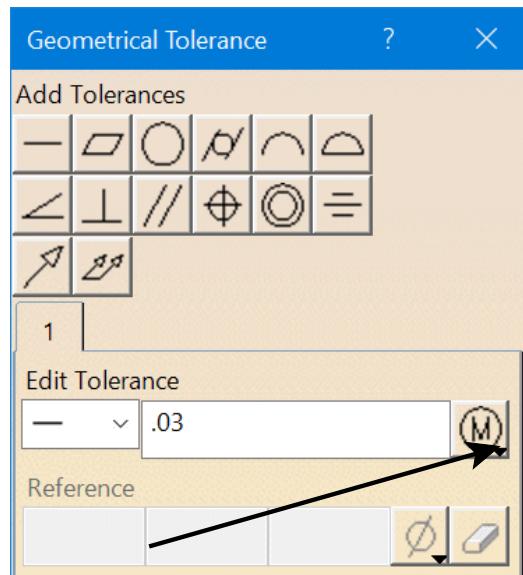



Change the symbol to **Straightness** with a value of 0.03 and select **OK**.  The tolerance appears.


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.


Select before the value in the *Edit Tolerance* field, then select the symbol drop down 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 symbol drop down again and choose the **Maximum Material Condition** symbol.  The symbol is inserted after the value.

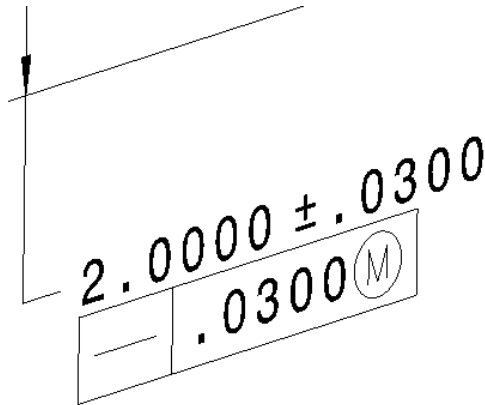
Select **OK**. The tolerance appears, but it has an exclamation point in the specification tree. The Geometrical Tolerance option does not currently allow an axis straightness tolerance to be defined. As a result, it cannot be positioned.

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.


Close the document.

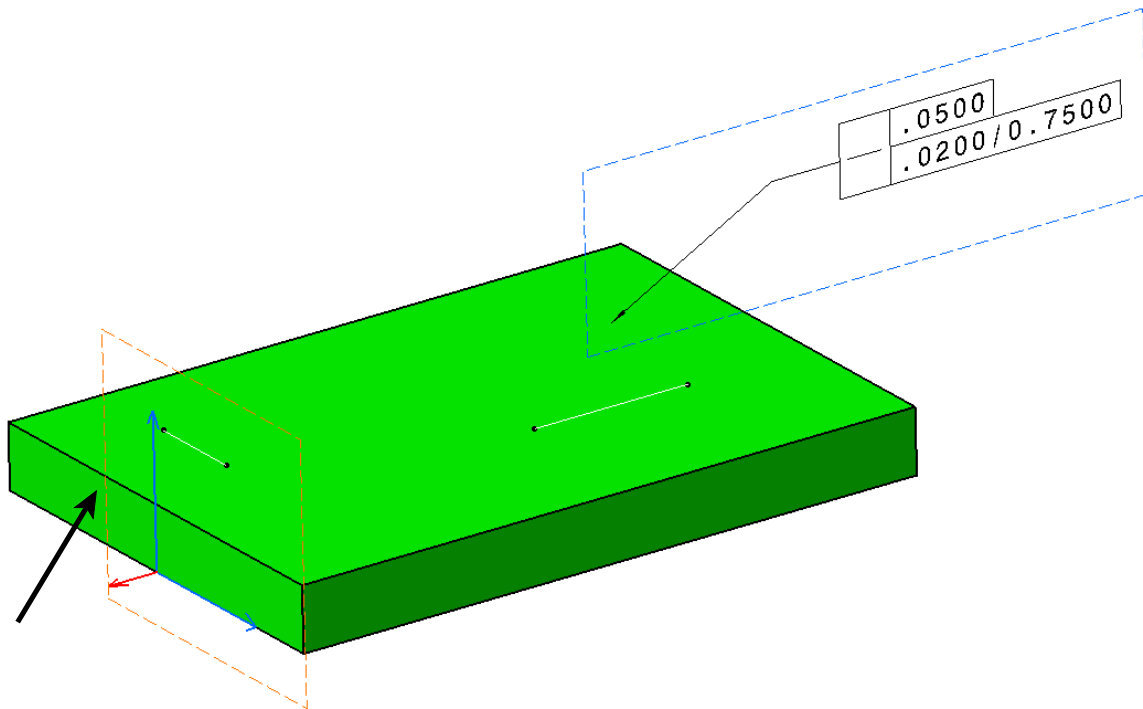
Open the **Straightness - Unit Basis** document. You will manually create the tolerances that were created earlier.

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

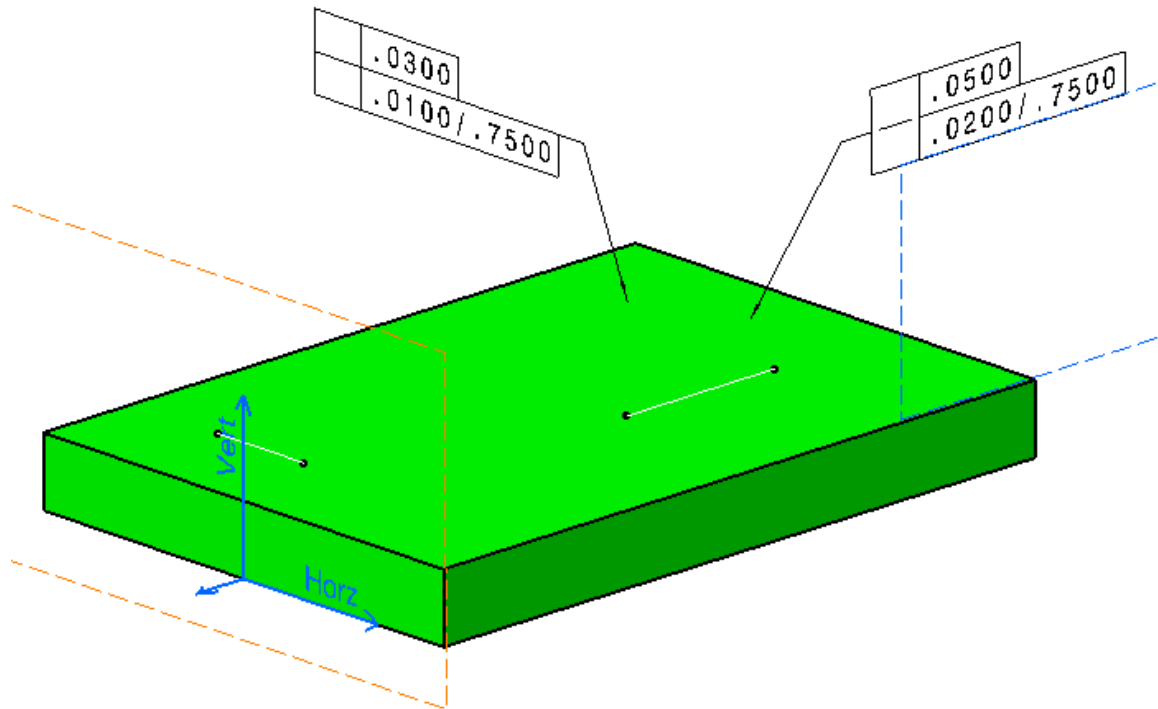
Change the symbol to **Straightness** with a value of **0.05** and select **OK**.  The tolerance appears.

To define the other tolerance correctly, a different view is required. CATIA will not automatically make a new view when the tolerance is created manually.

Select the **View From Reference** icon, then create a front view on the face that is shown below.  The view is created.



Create the second tolerance shown below. It should be created in the same matter as the previous tolerance.



Close the document.

Circularity

Circularity tolerances control how round a surface must be in order to meet the design requirements.

If applied to a surface other than a sphere, all points of the surface that are intersected by a plane normal to the axis of the surface must exist within the tolerance zone specified by two concentric circles that are separated by the tolerance value.

If applied to a sphere, all points of the surface that are intersected by any plane passing through the center point of the sphere must exist within the tolerance zone specified by two concentric circles that are separated by the tolerance value.

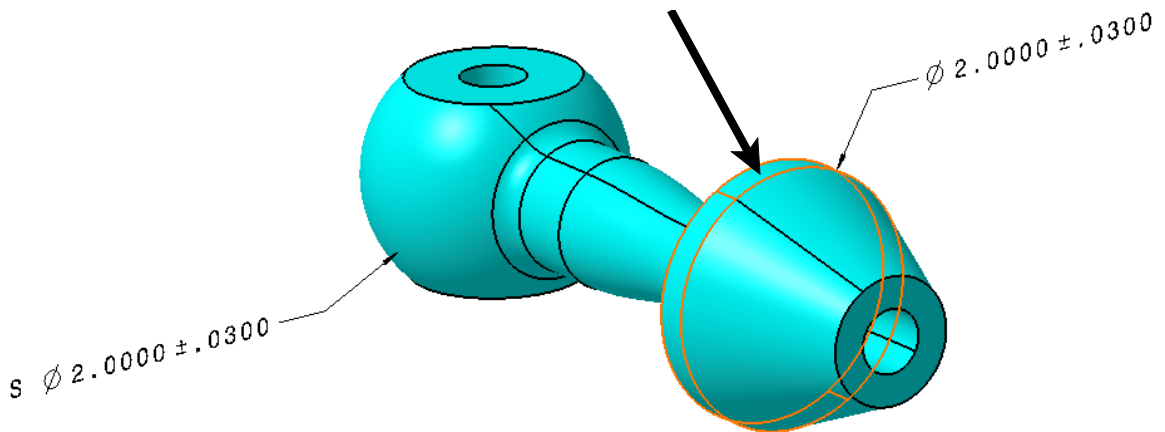
Open the Circularity document. A view and two dimensions have already been created.

Tolerancing Advisor

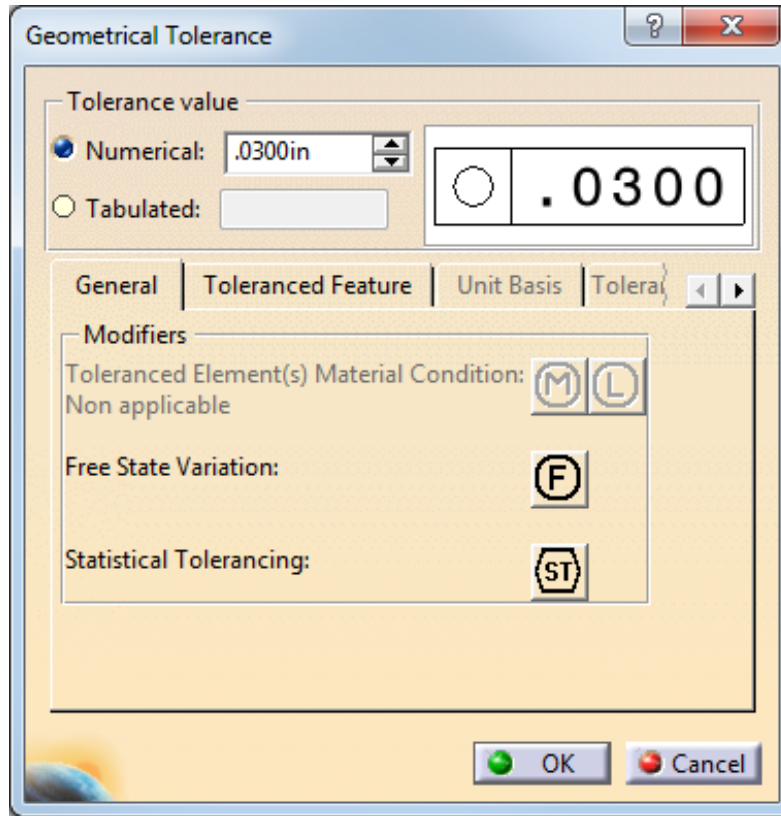
As stated before, the Tolerancing Advisor provides guidance to ensure that the geometrical tolerances are valid.

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

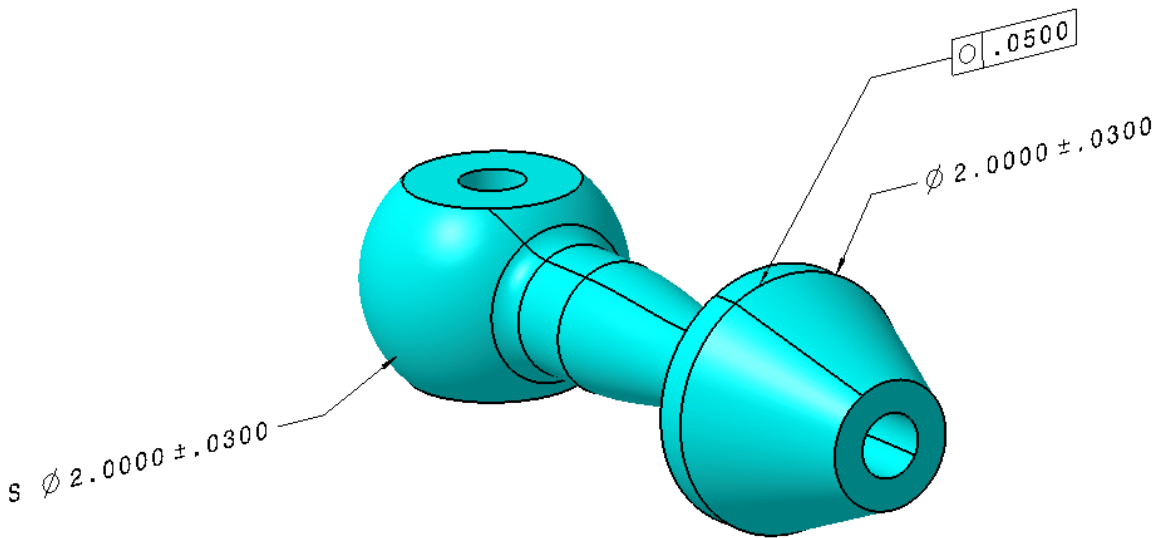
Select the cylindrical surface shown below. You will create a circularity tolerance for this surface.



Select the **Circularity Specification**.  The *Geometrical Specification* window appears.



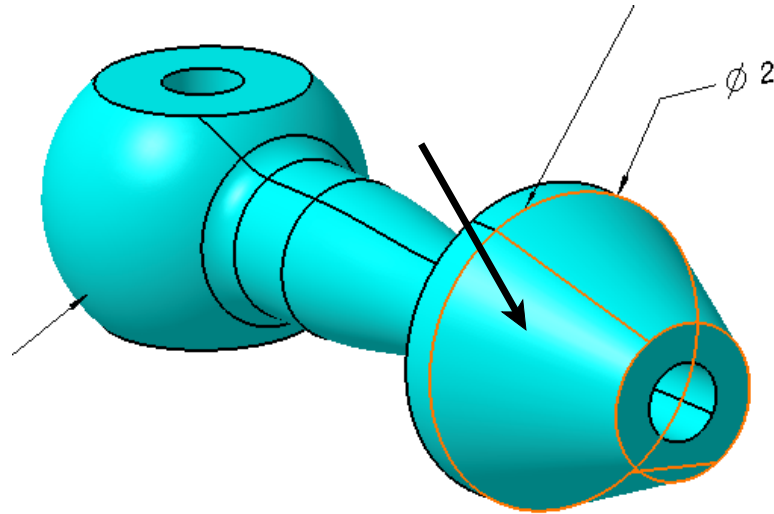
Change the *Numerical* value to **0.05** and select **OK**, then select **Close**. The tolerance appears. Position it as shown below.



Clicking and dragging on the yellow diamond at the end of the leader will enable it to slide in a circular path around the surface, similar to the linear path for straightness.

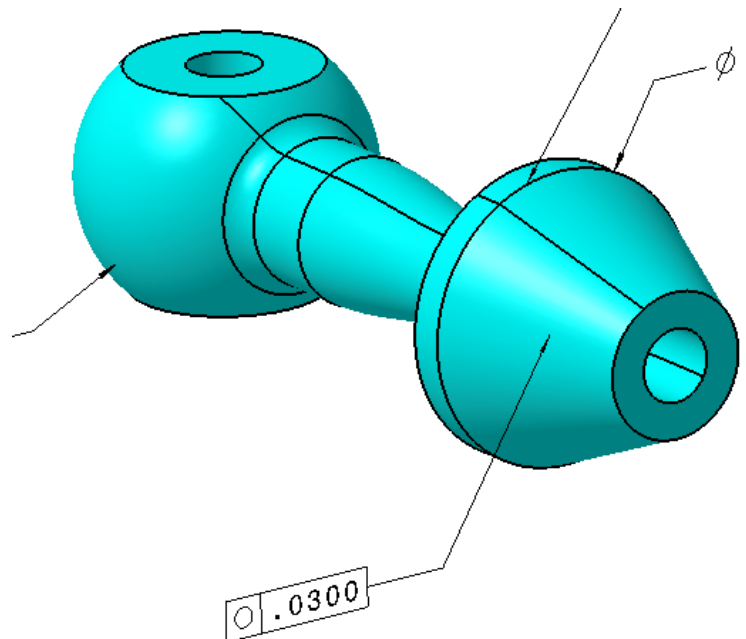
Select the **Tolerancing Advisor** icon.  The *Semantic Tolerancing Advisor* window appears. This time, you will select a cone-shaped surface.


Select the **conical surface shown below**. The **Cone Angle Creation** icon is available now instead of the diameter and radius options.



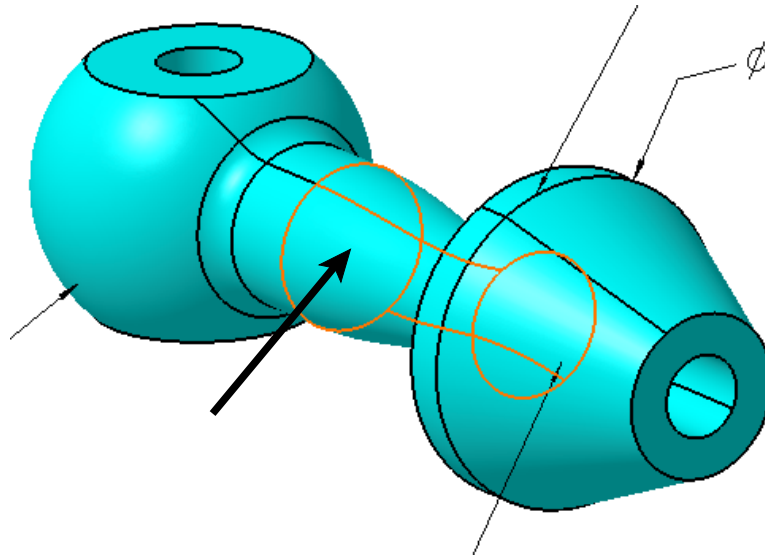
Select the **Circularity Specification**.  The *Geometrical Specification* window appears.

Change the *Numerical* value to **0.03** and select **OK**, then select **Close**. The tolerance appears.




Select the **Tolerancing Advisor** icon again.  The *Semantic Tolerancing Advisor* window appears. You will place a circularity tolerance on a surface of revolution.

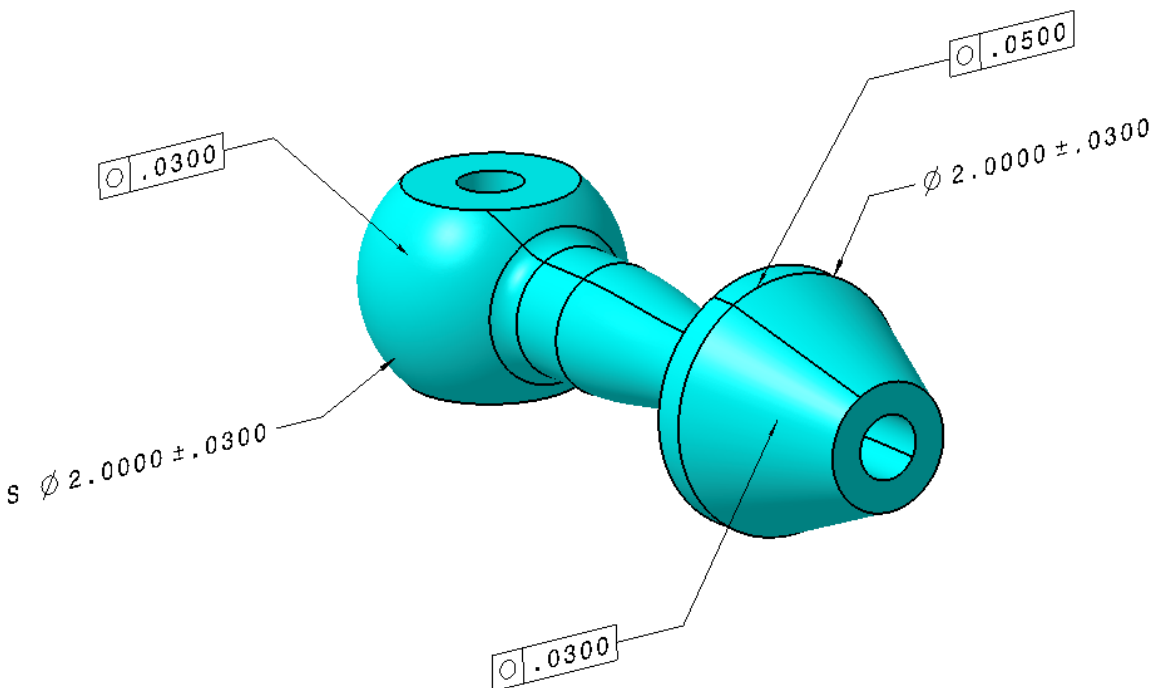
Select the surface shown below. The circularity option does not appear. This is a limitation of the Tolerancing Advisor. The circularity tolerance can be created with the Geometric Tolerance icon, but it will be considered non-semantic.



Select the spherical surface instead, then select the **Circularity Specification icon.**

 The *Geometrical Specification* window appears. You will place a circularity tolerance on the sphere.

Change the *Numerical* value to 0.03 and select *OK*, then select *Close*. Position the tolerances as shown.



Save and close the document.

Manually

Now, you will create the same geometrical tolerances manually.

Open the *Circularity* document, then select the *Geometrical Tolerance* icon. 

Ensure you open the original document, not the one you saved. Nothing will happen until an element is selected.

Select the cylindrical surface, then change the symbol to *Circularity*. 

Enter 0.05 in the *Tolerance* field and select *OK*. The tolerance appears. It is identical to the one created with the Tolerancing Advisor.

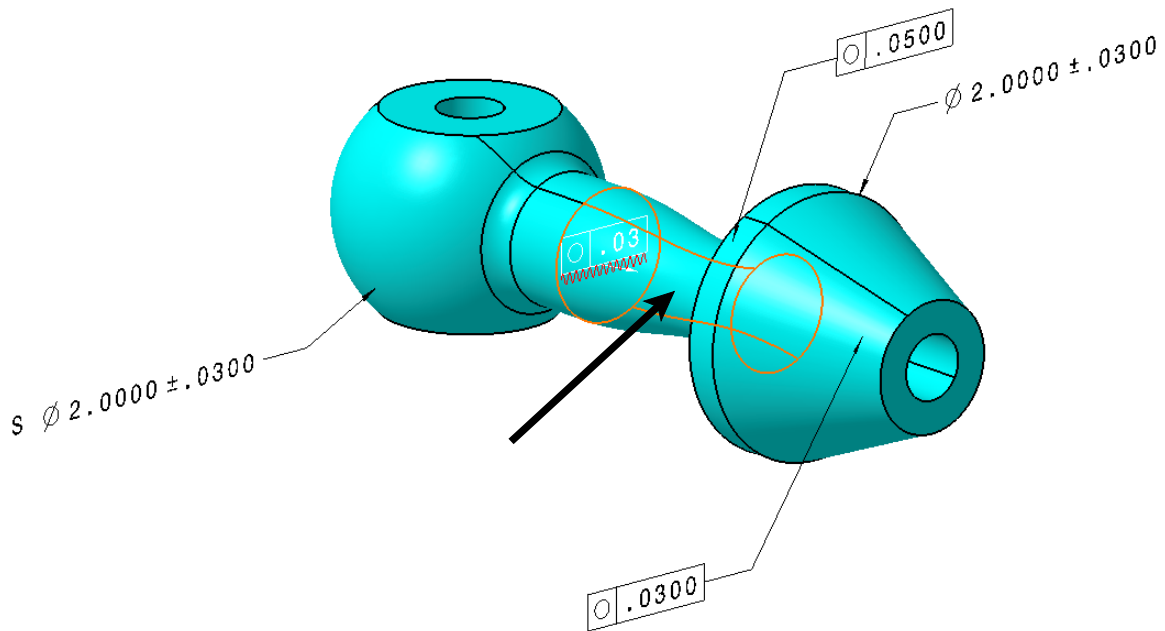
Select the *Geometrical Tolerance* icon again, then select the conical surface. 


The *Geometrical Tolerance* window appears.

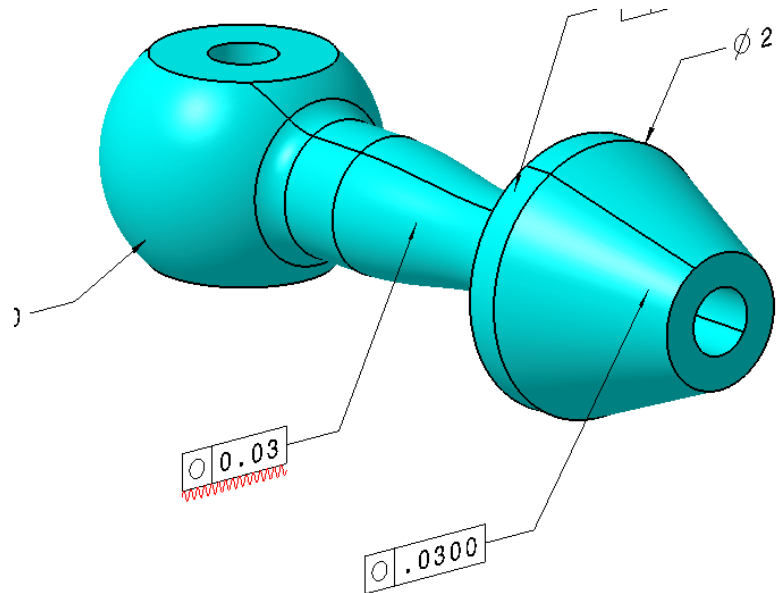
Change the symbol to *Circularity* with a value of 0.03 and select *OK*.  The tolerance appears.

Select the *Geometrical Tolerance* icon, then select the surface shown below. 

The *Geometrical Tolerance* window appears.



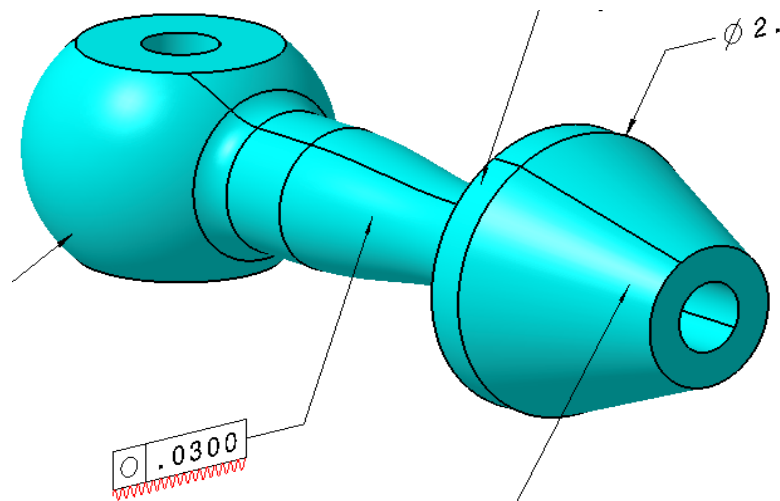
Change the symbol to **Circularity** with a value of **0.03** and select **OK**.  The tolerance is considered non-semantic, but at least it was created this time. It appears with the red, squiggly line beneath it, and will be referred to as a geometrical tolerance in the specification tree.




The value did not adjust to .0300 as it should have because it is non-semantic. You can force it to read correctly by entering the exact value you want to see.

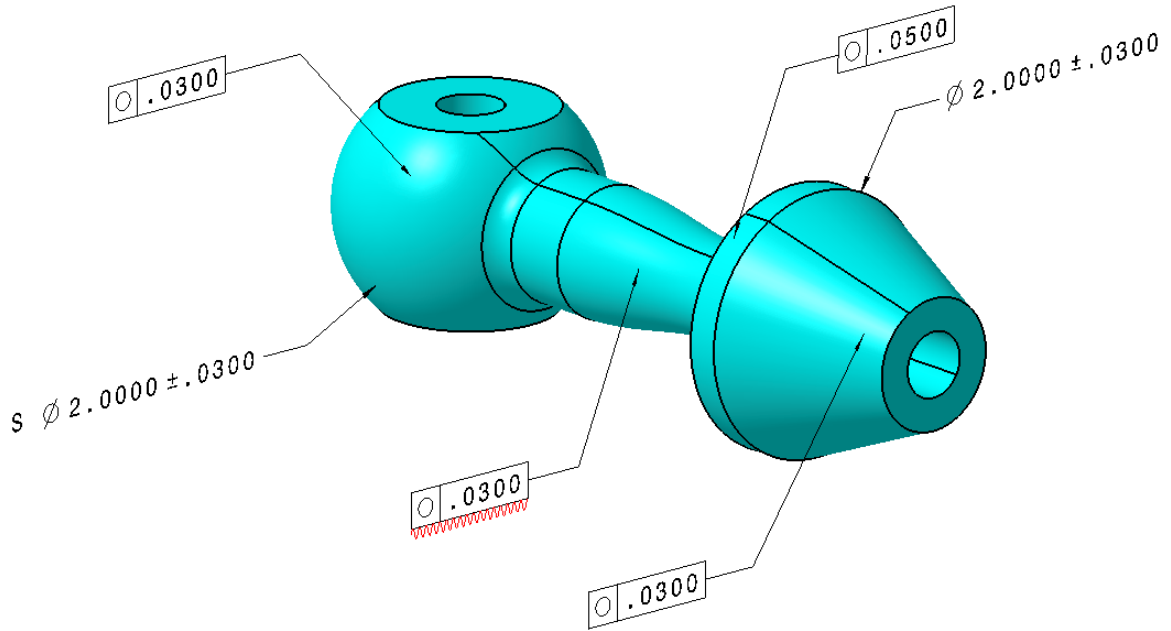
Double-select on the new geometrical tolerance. The *Geometrical Tolerance* window appears.

Change the value to .0300 and select OK. Now the value reads properly.



Select the **Geometrical Tolerance** icon, then select the spherical surface.  The *Geometrical Tolerance* window appears.

Change the symbol to **Circularity** with a value of **0.03** and select **OK**.  Position the tolerances as shown below.



Close the document.