

TABLE OF CONTENTS

Introduction	1
Functional Tolerancing and Annotation	2
Pull-down Menus	3
Insert	3
Functional Tolerancing and Annotation Workbench	4
Bottom Toolbar Changes	5
3D Grid Toolbar	5
Grouping Toolbar	5
Top Toolbar	6
Text Properties	6
Dimension Properties	7
Numerical Properties	7
Position and Orientation	7
GD&T Review	8
Geometric Characteristic Symbols	8
Modifying Symbols	9
Other Symbols	10
Views	11
Front View/Annotation Plane	11
Section View/Annotation Plane	16
Section Cut/Annotation Plane	17
Offset Section View/Section Cut	18
Aligned Section View/Section Cut	21
Orientation	24
Transfer	28
Using Axis Systems	30
Properties	35
Changing Support	37
Datums	39
Planar Datums	39
Tolerancing Advisor	40
Datum Reference Frames	46
Manually	54
Positioning a Datum	58
Datum Targets	60
Points	60
Tolerancing Advisor	60
Manually	64
Lines	66
Tolerancing Advisor	66
Manually	68
Areas	69
Tolerancing Advisor	69
Manually	78

Datum Axes and Center Planes	84
Tolerancing Advisor	84
Manually	92
Dimensions	95
Creating Dimensions	95
Length/Distance	95
Angle	104
Radius	106
Diameter	108
Coordinate	110
Cumulated	111
Stacked	113
Dimensions for curves	114
Generative Dimensions	116
Setup Parameters	118
Dimension Lines	118
Tolerance	121
Numerical Display	124
Modifying Dimensions	127
Top Toolbar	127
Pull Down Menu Tools, Options	129
Properties	132
Positioning	147
Tolerancing Advisor	150
Creating Dimensions	150
Modifying Dimensions	160
Propagation Selection	162
Geometrical Tolerancing	165
Form Controls	165
Flatness	165
Tolerancing Advisor	166
Manually	171
Straightness	173
Tolerancing Advisor	173
Manually	182
Circularity	188
Tolerancing Advisor	188
Manually	193
Cylindricity	196
Tolerancing Advisor	196
Manually	199

Orientation Controls	201
Perpendicularity	201
Tolerancing Advisor	201
Manually	207
Angularity	210
Tolerancing Advisor	210
Manually	212
Parallelism	214
Tolerancing Advisor	214
Manually	218
Location Controls	221
Position	221
Tolerancing Advisor	221
Manually	235
Concentricity	247
Tolerancing Advisor	247
Manually	249
Symmetry	250
Tolerancing Advisor	250
Manually	253
Runout Controls	255
Circular Runout	255
Tolerancing Advisor	255
Manually	259
Total Runout	261
Tolerancing Advisor	261
Manually	264
Profile Controls	265
Profile of a Surface	265
Tolerancing Advisor	265
Manually	269
Profile of a Line	270
Tolerancing Advisor	270
Manually	274
Unilateral or Unequal Bilateral	275
Unilateral - Outward	275
Unilateral - Inward	275
Bilateral - Unequal	275
Modify	276
Changing Datum Reference Frame	276
Adding a Geometrical Tolerance to a Datum	278
Grouping	279
Positioning	281
Basic Dimensions	283

Annotations	293
Creating Text	293
Modifying Text	298
Top Toolbar	298
Font properties	298
Justification	300
Anchor point	300
Frame	301
Insert Symbol	302
Properties	304
Adding a Leader	315
Links	319
Orientation Link	319
Positional Link	321
Attribute Link	322
Query Object Links	323
Isolate Text	324
Flag Notes	325
Roughness Symbol	328
Weld Symbols	330
Graphical Properties Toolbar	332
Copy Object Format	335
Tolerancing Advisor	337
Text	337
Flag notes	339
Roughness Symbol	341
Geometry for 3D	343
Restricted Area	343
Construction Geometry Creation	346
Construction Geometry Management	354
Thread Representation Creation	357
Geometry Connection Management	362
Note Object Attribute	371
Reports	375

Visualization	381
List Annotation Set	381
3D Annotation Query Switch	382
Filtering	384
Mirror	387
Clipping Plane	388
Captures	389
Displaying Captures	389
Creating Captures	392
Naming Convention	392
Tolerancing Capture Workbench	393
Viewpoints	395
Active View State	397
Clipping Plane	398
Current State	400
Creating the Side Capture	402
Creating the Top Capture	403
Creating the 3D All Capture	404
Creating the 3D None Capture	405
Properties	406
Capture Management	408
Problems	411
Problem #01	411
Problem #02	413
Problem #03	415
Problem #04	418
Appendix A	427
Mechanical Design - FT&A - Tolerancing	427
Mechanical Design - FT&A - Display	428
Mechanical Design - FT&A - Constructed Geometry	429
Mechanical Design - FT&A - Manipulators	430
Mechanical Design - FT&A - Dimension	431
Mechanical Design - FT&A - Annotation	432
Mechanical Design - FT&A - Tolerances	433
Mechanical Design - FT&A - View/Annotation Plane	434

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 useful to define characteristics of your parts and products in a 3D environment. Using these tools allows you to avoid having to create two dimensional drawings. Many companies have expressed an interest in going to a *paperless* environment but find it difficult to accomplish this task. This is one set of tools that can help make this move a reality.

It is important to understand that to fully utilize 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 you GD&T. It is assumed that you are already knowledgeable of these fundamentals since proper use of these tools requires it. There is some assistance provided within the functionality of the workbench but it will allow you to improperly tolerance and annotate a design.

Geometrical Tolerancing

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

It is important to note that you should have a good understanding of the fundamentals of geometrical dimensioning and tolerancing (GD&T) before using these tools on your design. The tolerancing advisor will help guide you in the proper syntax of geometric tolerancing but there is no way for CATIA to know your design intent. It is not the intention of this course to teach you GD&T but rather how to apply it with the tools available in CATIA.

Many of the examples shown in this section are not finished parts but rather small examples of how to use the tools. You should make yourself aware of your company's procedures and standards in order to meet the criteria set by the company. 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 that will be discussed is flatness.

Flatness

Flatness controls how flat a surface has to 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 separated by the tolerance value.

Open the Flatness document. Notice that a view already exists in this part.

There are two main methods for putting a flatness tolerance on your part. You can either do it directly using the geometrical tolerance option or you can use the tolerancing advisor. The tolerancing advisor will guide you through putting geometrical tolerances on your part and it will keep you from creating an invalid tolerance. However, it can be frustrating if you have a good understanding of the fundamentals of GD&T and just want to quickly put the tolerance on your part. There are certain procedures you have to follow in order to create the tolerances using the tolerancing advisor. In this course both methods will be shown and performed in order for you to get a good feel for both options.

Tolerancing Advisor

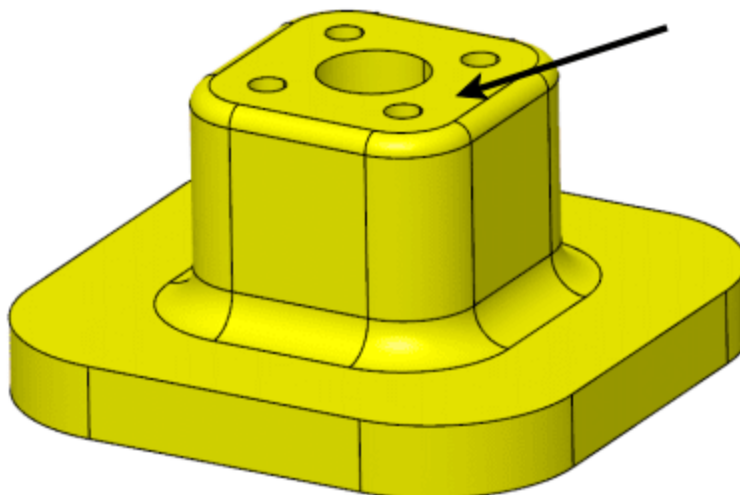
As stated before the tolerancing advisor will help guide you to make sure you create valid geometrical tolerances.

Select the Tolerancing Advisor icon.

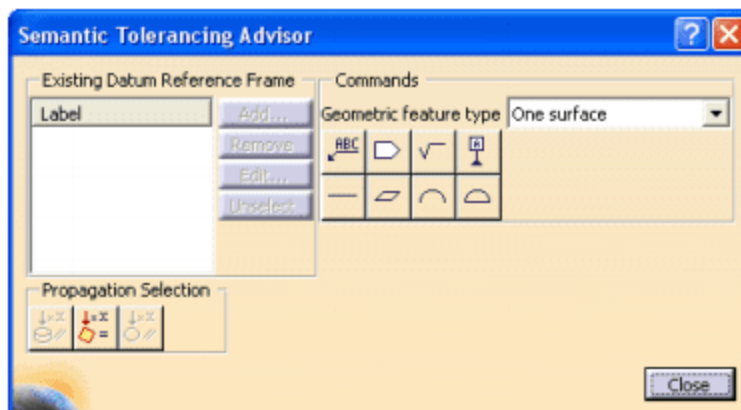


The *Semantic Tolerancing Advisor* window appears.

Select the top of the part as shown below. You are going to create a flatness tolerance for the top face of the part.



Notice that the *Semantic Tolerancing Advisor* window changes.



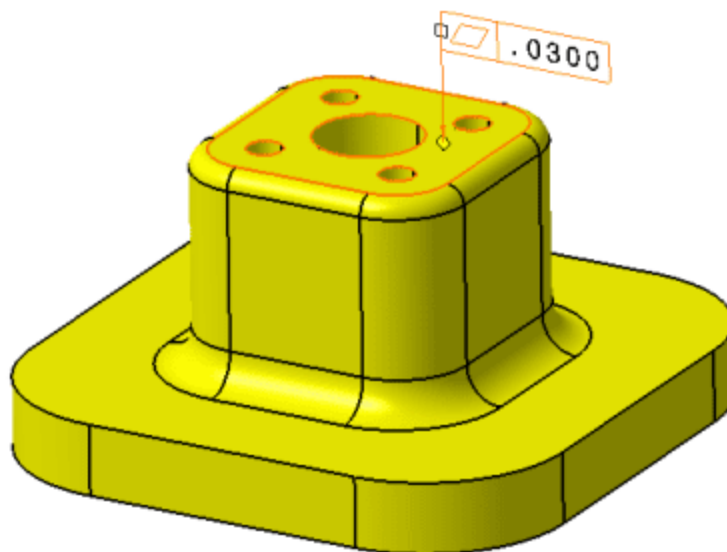
Based on your selection of one surface the options available are shown. The advisor filters out the options that are not valid for a single surface. These symbols are the standard geometrical tolerance symbols and they should look familiar. You will be using all of these as you progress through the exercises so only the pertinent options will be discussed in each exercise.

Select the Flatness Specification.  The Geometrical Specification window appears.

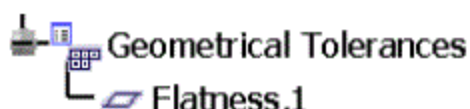


The options will be discussed as they are used throughout the exercises. Only the pertinent ones will be discussed in each exercise.

Change the *Numerical* value to 0.03 and select *OK*. The tolerance appears. Notice that the tolerancing advisor stays active and that the flatness specification is highlighted in the window.



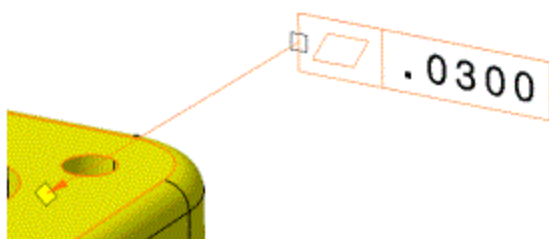
The tolerance will appear in the tree as shown below.



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

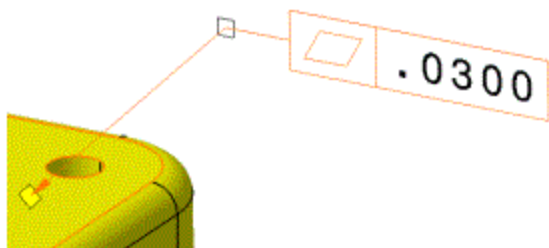
Press and hold the first mouse button while on the tolerance and drag it to the right.

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.



Press and hold the first mouse button while on the white square and drag it to the left.

The tolerance should now appear as shown below.



You will probably want to move the tolerances to better locations throughout the exercises. Since it is the same procedure every time it will not be discussed every time.

The tolerancing advisor should still be active. If it is not, select the Tolerancing Advisor icon again.



The *Semantic Tolerancing Advisor* window appears.

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.

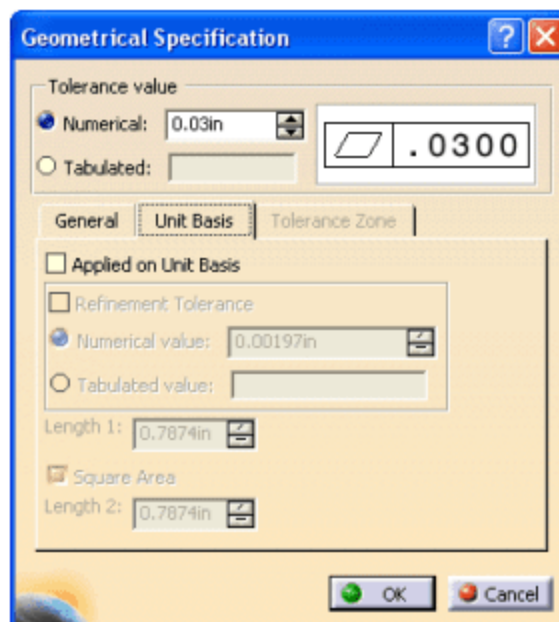
Select the Flatness Specification.



The *Geometrical Specification* window appears.

Change the Numerical value to 0.03. This time you are going to specify a refinement on a unit basis.

Select the **Unit Basis** tab. The options change in the window.

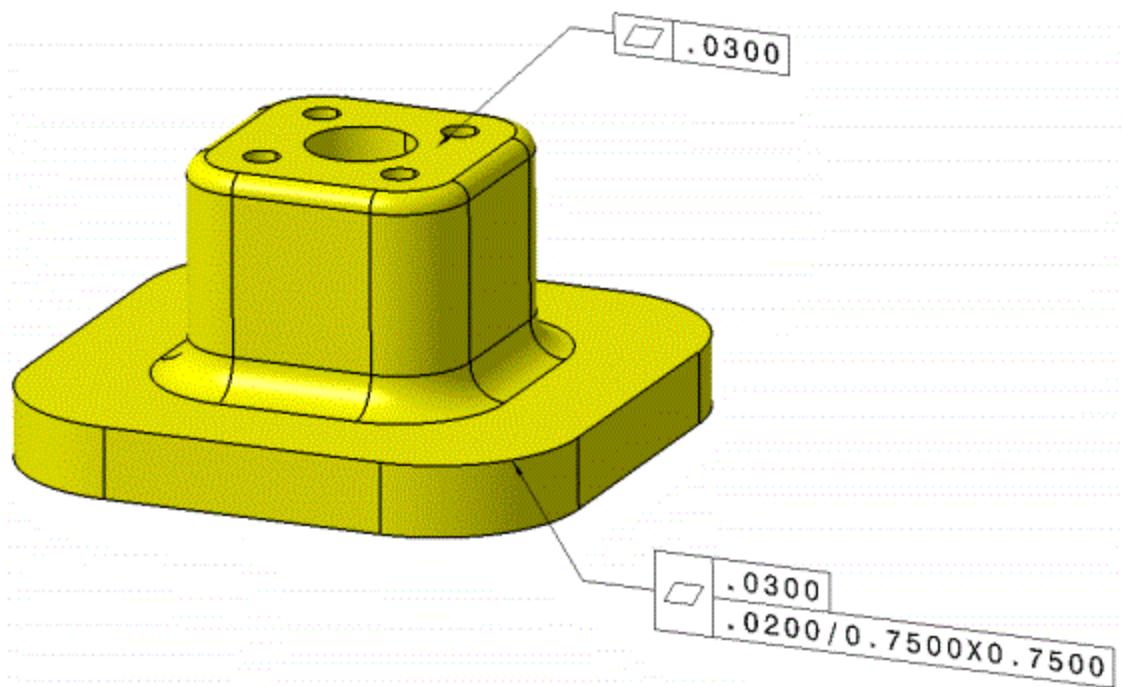


The **Unit Basis** tab allows you to define a refinement tolerance for a smaller area of the surface along with a total variation or you can use it by itself. In this case you are going to define a refinement stating that for a 0.75 by 0.75 square area the maximum variation can only be 0.02.

Select the **Applied on Unit Basis** option and the **Refinement Tolerance** option.

Change the **Numerical value** of the refinement to be 0.02 and **Length 1** to be 0.75 and select **OK**. The tolerance appears.

Position the tolerance as shown below. That tolerance is stating that the maximum variation across the entire surface can only be 0.03 inches but you can only have a maximum variation of 0.02 inches within a 0.75 inch square area.



Caution should be given if you decide to use 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 give you a huge variation across the entire surface.

Save and close your document.

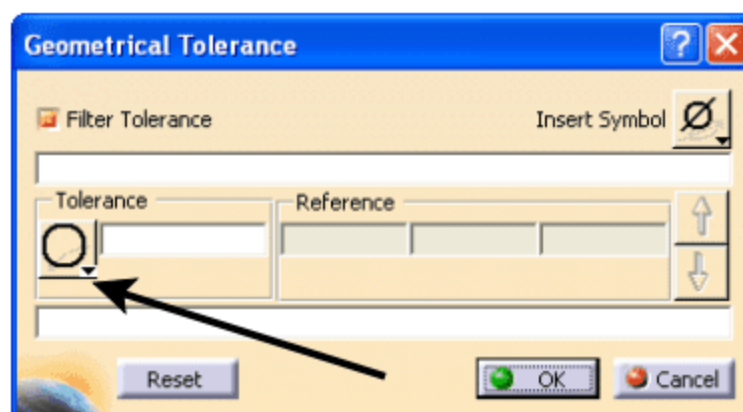
Manually

You are going to create the same geometrical tolerances but you are going to do it manually.

Open the Flatness document again. Make sure you open the original document and not the one you just saved.

Select the Geometrical Tolerance icon.  Nothing will happen until you select an element.

Select the top face of the part. The *Geometrical Tolerance* window appears. This window will allow you key text above and below the feature control frame as well as define the geometrical tolerance. In addition a *Tools Palette* toolbar will appear with propagation options. These were discussed in an earlier exercise.



Select the black arrow under the circle as shown above. Notice a lot more tolerancing options appear because it is not as knowledgeable as the advisor.

Select Flatness.




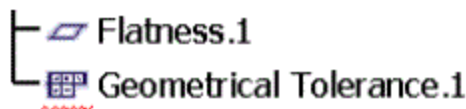
Key in 0.03 in the box next to the flatness and select OK. The tolerance appears and it is basically identical to the one you created with the tolerancing advisor. The difference is that some red ~ symbols appeared on the tolerance until you selected OK. This is the symbol it uses for non-semantic annotations, meaning that it considers them invalid either due to syntax or due to associativity. Once you finalized the creation of the tolerance it removed those markings because it realized that it is a valid tolerance. Those markings can be set to not appear using *Tools, Options*.

Select the Geometrical Tolerance icon again.



Select the bottom face of the part. The *Geometrical Tolerance* window appears.

Change the symbol to Flatness with a value of 0.03.  Before you select *OK*, look at the tolerance in the graphic area, you should see a red ~. Look in the specification tree as well. Notice that the tolerance is referred to as a *Geometrical Tolerance* instead of a *Flatness* at the moment. Also notice the red underline signifying that it is non-semantic.



Select *OK*. The tolerance is now referred to as a Flatness and is almost identical to the one defined with the tolerancing advisor except for the unit basis.

Double select on the last tolerance using the first mouse button. The *Geometrical Specification* window appears. This is the same window that appears when you use the tolerancing advisor.

Select the *Unit Basis* tab, turn on the *Applied on Unit Basis* and the *Refinement Tolerance* options, change the *Numerical value* to 0.02 and *Length 1* to 0.75 and select *OK*. It is now identical to when you used the tolerancing advisor.

Close your document.

Straightness

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

If applied to a surface, it controls how straight a line element of the surface has to 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 separated by the tolerance value.

If applied to an axis (center line) of a cylindrical feature of size then it controls the straightness of the axis. The axis has to exist within the tolerance zone specified by a cylinder whose diameter is equal to the tolerance value.

If applied to a center plane then it is controlled similar to a surface where every line element of the plane has to exist within the tolerance zone specified by two parallel planes separated by the tolerance value.

Open the Straightness document. Notice some views and some dimensions already exist in this part. Once again you either use the tolerancing advisor or you can use the geometrical tolerance option.

Tolerancing Advisor

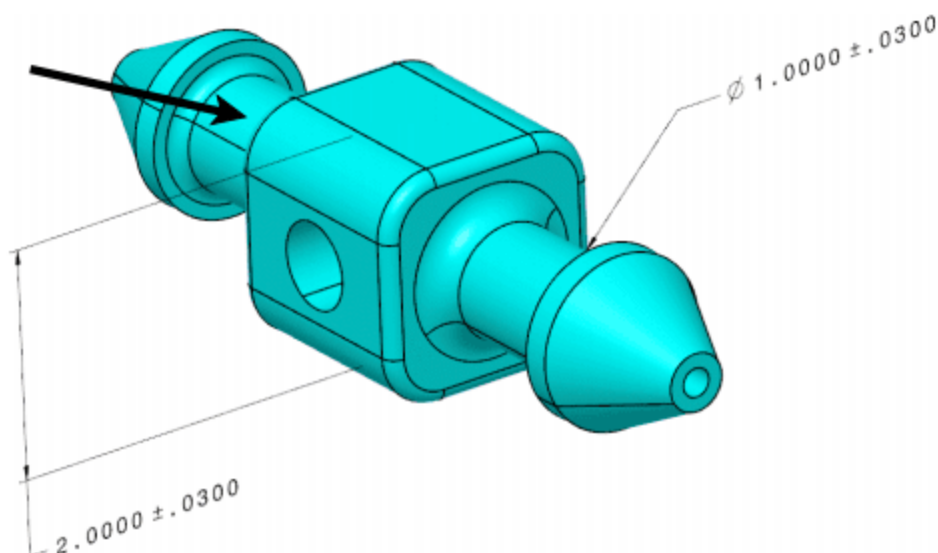
As stated before the tolerancing advisor will help guide you to make sure you create valid geometrical tolerances.

Select the Tolerancing Advisor icon.

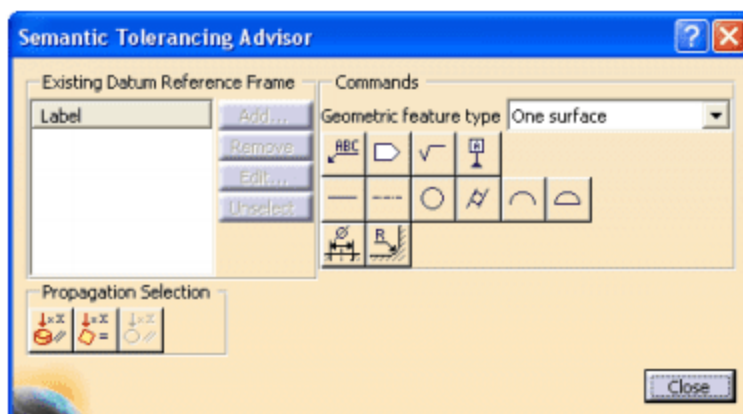


The *Semantic Tolerancing Advisor* window appears.


Select the cylindrical surface as shown below. You are going to create a straightness tolerance for this surface. This is not a feature of size therefore it will apply to the line elements of the surface and not the axis (center line) of the surface.

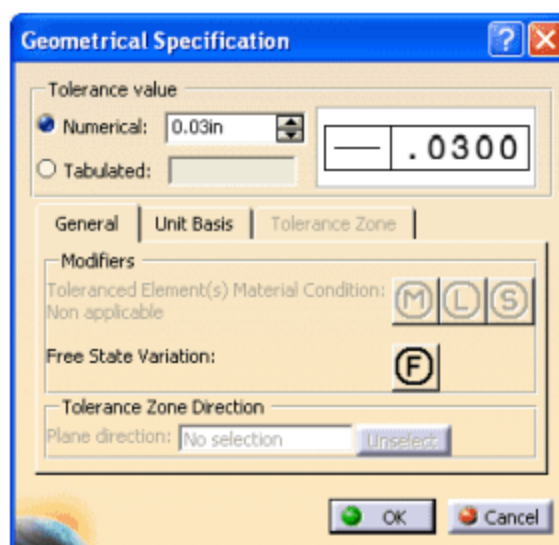


Notice that the *Semantic Tolerancing Advisor* window changes. Based on your selection of a cylindrical surface the options available are shown. The advisor filters out the options that are not valid for this surface.



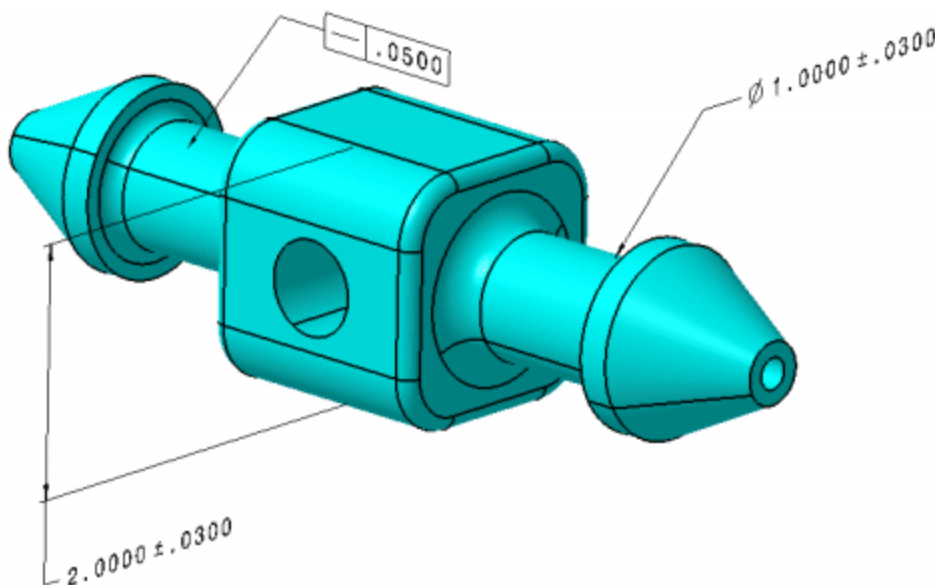
原创力文档
max.book118.com
预览与源文档一致, 下载高清无水印

Select the Straightness Specification.  The Geometrical Specification window appears.

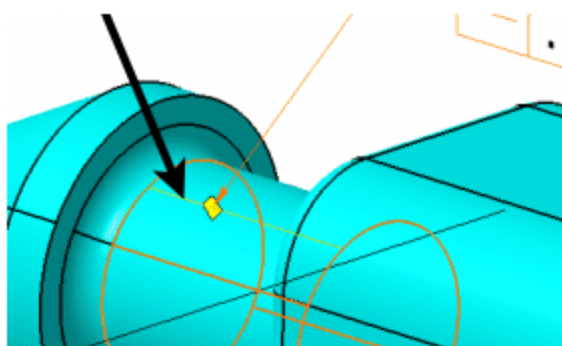



Change the *Numerical* value to 0.05 and select **OK**. The tolerance appears. Notice that the tolerancing advisor stays active and that the straightness specification is highlighted in the window.

Select *Close* and position the tolerance as shown below.




Select on the tolerance and press and hold the first mouse button on the yellow diamond. You should see a yellow line appear showing you how the arrow head can be dragged along for this specification. The line is hard to see since it is yellow so you may have to zoom in on the area.

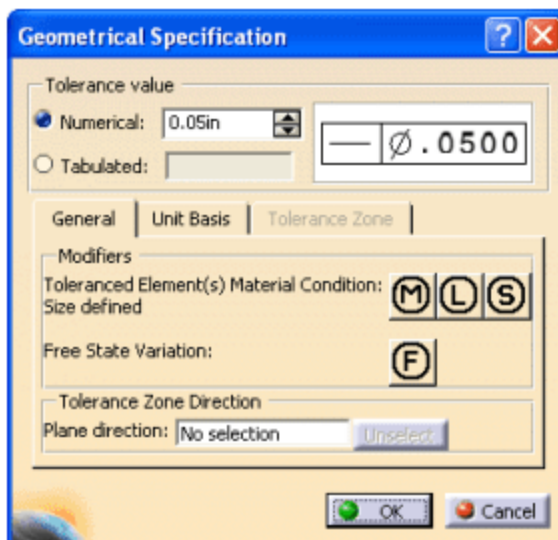


Select the Tolerancing Advisor icon again.  The *Semantic Tolerancing Advisor* window appears. This time you are going to select an existing dimension in order to add the straightness specification to the feature of size.

Select the 1.0000 dimension. The same options appear in the window with the diameter option highlighted.

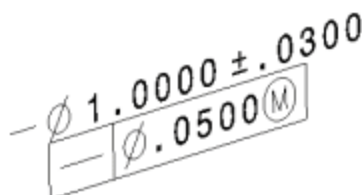
Select the Axis Straightness Specification.  The *Geometrical Specification* window appears.


Change the Numerical value to 0.05. Notice that the diameter symbol automatically appears in the feature control frame since it knows that it is a cylindrical tolerance zone. Also notice that the material conditions are available now.



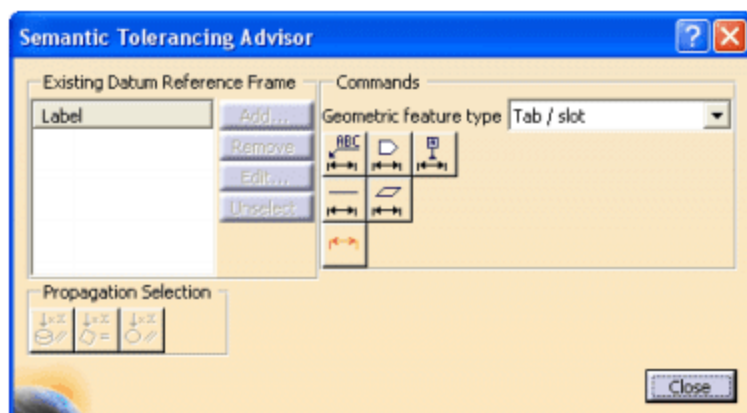
Select the Maximum Material Condition.  This allows for extra tolerance while still insuring the function of assembly.

Select OK and select Close. Notice that the straightness tolerance appears under the dimensional tolerance and it is linked to it so if you move the dimension the straightness tolerance will move with it. It is also in the view that contains the dimension.




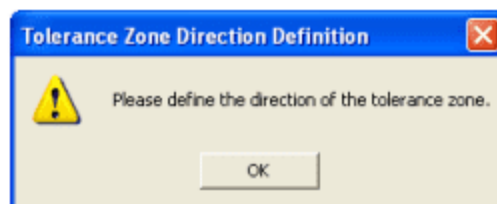
Select the Tolerancing Advisor icon again.  The *Semantic Tolerancing Advisor* window appears.

Select the 2.0000 dimension. The *Semantic Tolerancing Advisor* window looks different since you have basically selected two parallel faces by selecting the dimension. Therefore, it refers to the feature type as a *Tab/slot*.



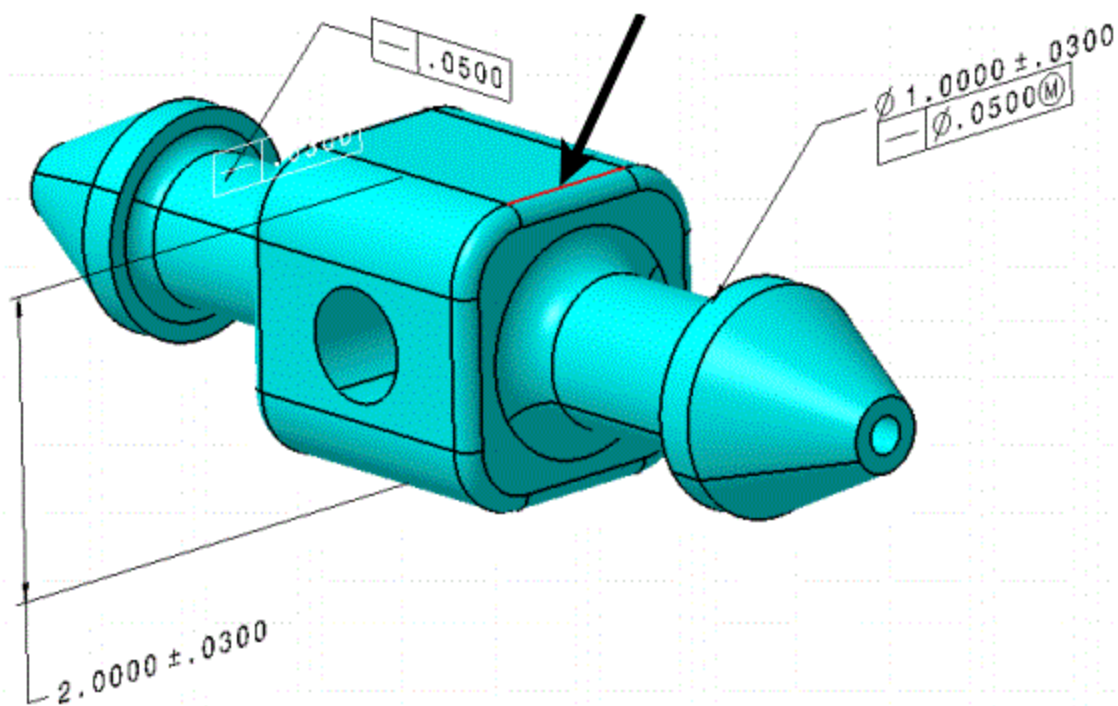
Select the Straightness Specification.  The *Geometrical Specification* window appears.

Change the Numerical value to 0.03 and choose the Maximum Material Condition and select OK.  A *Tolerance Zone Direction Definition* window appears.



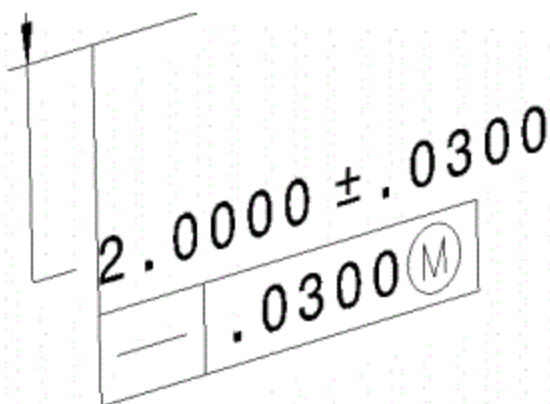
Select **OK**. A direction for the tolerance zone has to be defined because it is being applied to a plane.

Select in the *Plane direction* box and select the edge shown below.



Select **OK** and select **Close**. You will notice that the tolerance appears up above the top extension line. Normally you would want it to be located with the dimension.

Drag the value down to be located under the dimension value as shown below.



Select on the tolerance and press the third mouse button while on the yellow diamond at the end of the leader. Many options appear but they will be discussed later in the course.

Select the *Remove Leader / Extremity* option. The leader is removed.

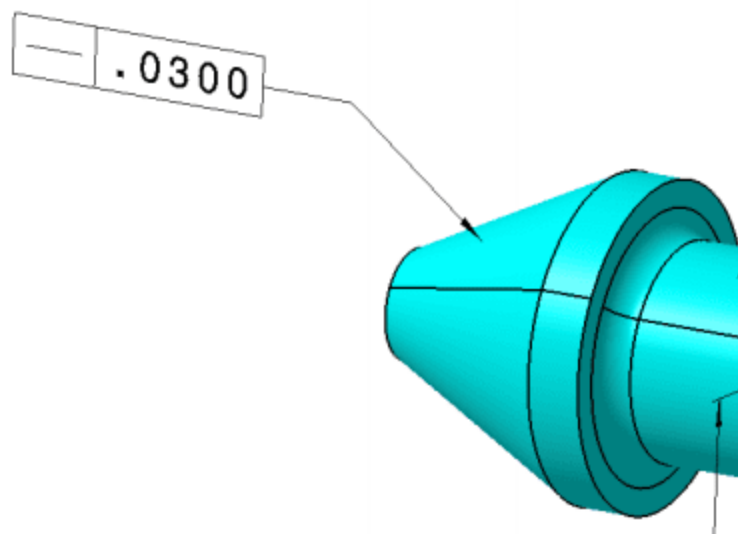
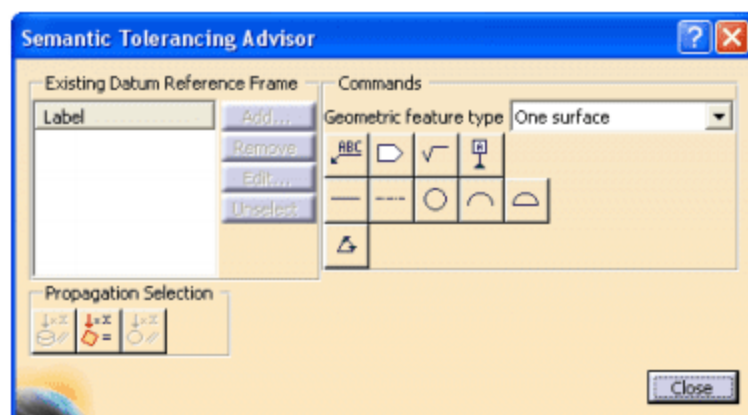
Move the dimension around. Notice that the tolerance follows the value correctly. Make sure you locate the dimension back to the original location.

You can also apply a straightness tolerance on a conical shape as well.

Using the tolerancing advisor create a straightness tolerance as shown below.




You should take note that instead of a diameter and radius option a cone angle option appears in the *Semantic Tolerancing Advisor* window. You may have a hard time getting the leader to attach to the right side of the feature control frame but do not worry about that now.



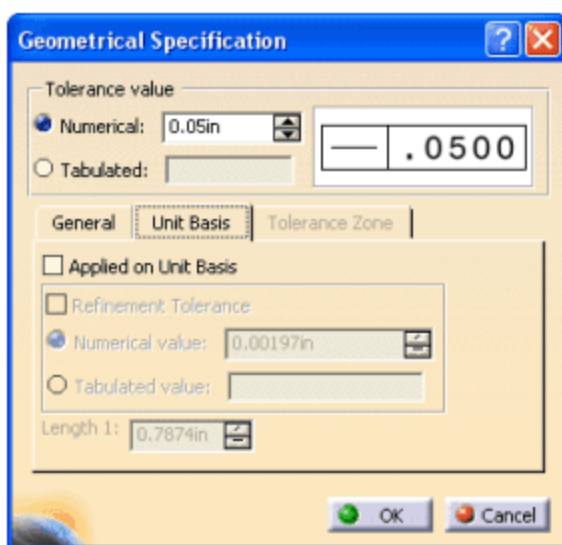
Save and close your document.

Open the Straightness - Unit Basis document. You are going to define some straightness tolerances and refine them with unit basis tolerances.

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

Select the top face of the part and choose the Straightness Specification.  The *Geometrical Specification* window appears. Notice that a view was automatically created since one did not already exist.

Change the Numerical value to 0.05 and select the Unit Basis tab. The options change in the window. These are very similar to the flatness option except you only have one length definition available.



The *Unit Basis* tab allows you to define a refinement tolerance for a smaller length of the plane along with a total variation or you can use it by itself. In this case you are going to define a refinement stating that for a 0.75 length the maximum variation can only be 0.02.

Select the *Applied on Unit Basis* option and the *Refinement Tolerance* option.

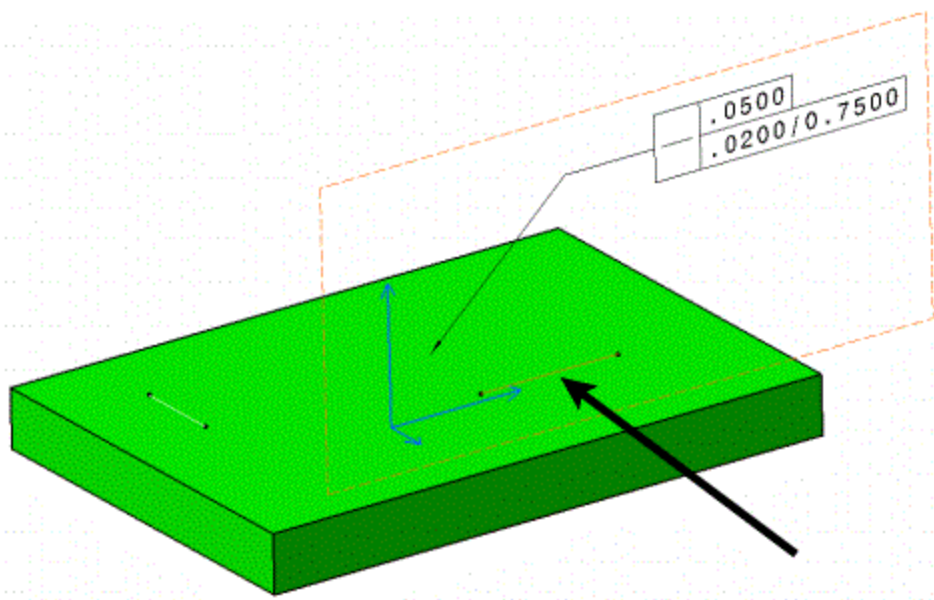
Change the Numerical value of the refinement to be 0.02 and Length 1 to be 0.75 and select OK. A *Tolerance Zone Direction Definition* window appears. Remember for a plane you must specify a tolerance direction for the straightness.

Select OK.

Select on the General tab.

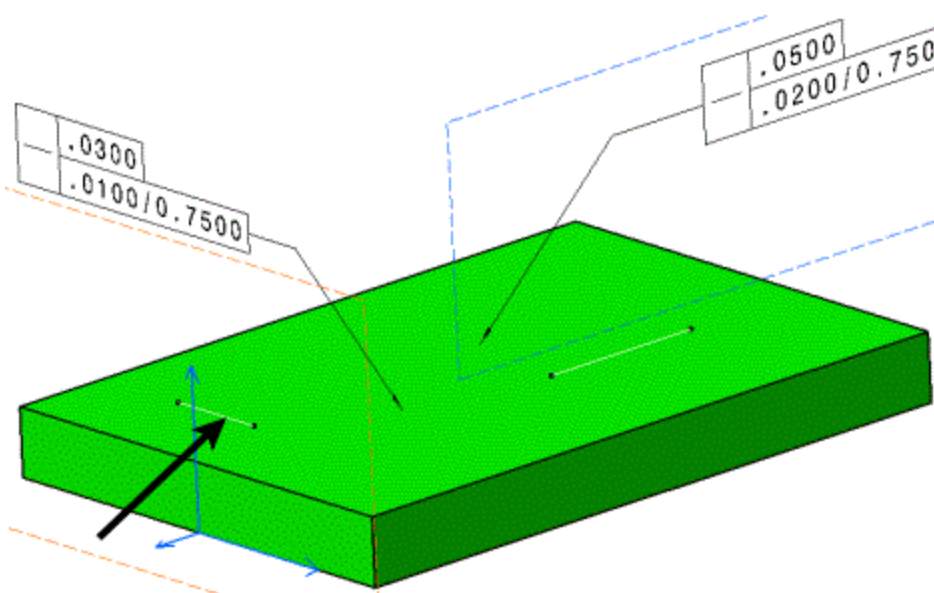
Select in the *Plane direction* box and select the line as shown below and select *OK*.

Position the tolerance as shown below.



Create a view that is normal to the existing view and make it current.

Using the same method, create another straightness tolerance as shown below using the other line as the direction.



Save and close your document.

Manually

You are going to create the same geometrical tolerances but you are going to do it manually.

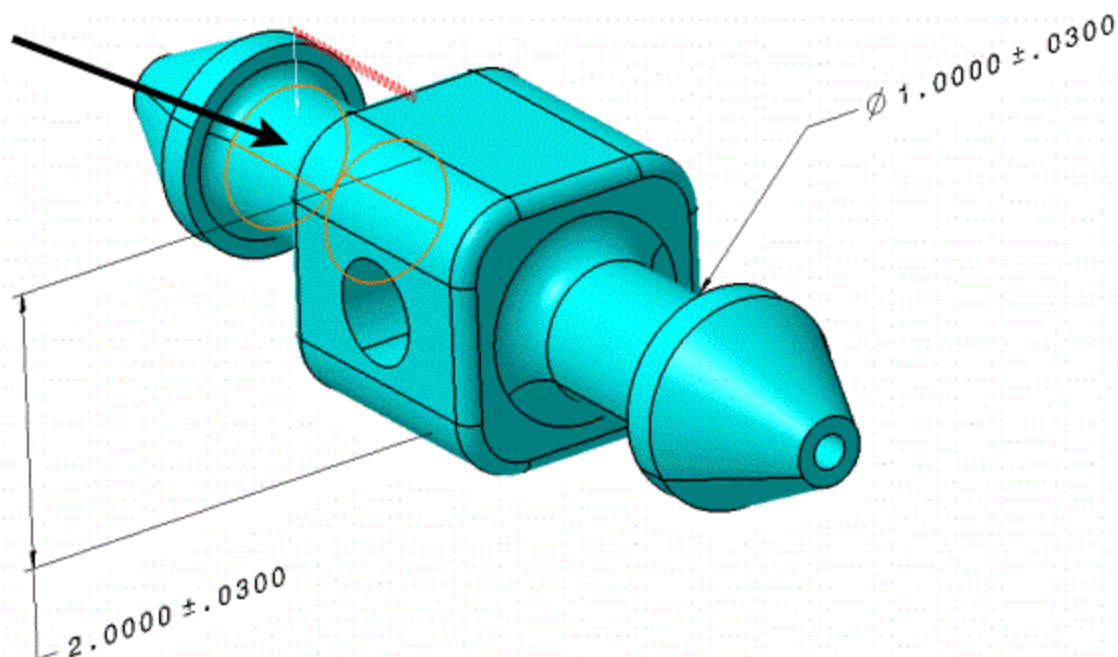
Open the Straightness document again. Make sure you open the original document and not the one you saved.

Select the Geometrical Tolerance icon.



Nothing will happen until you select an element.


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

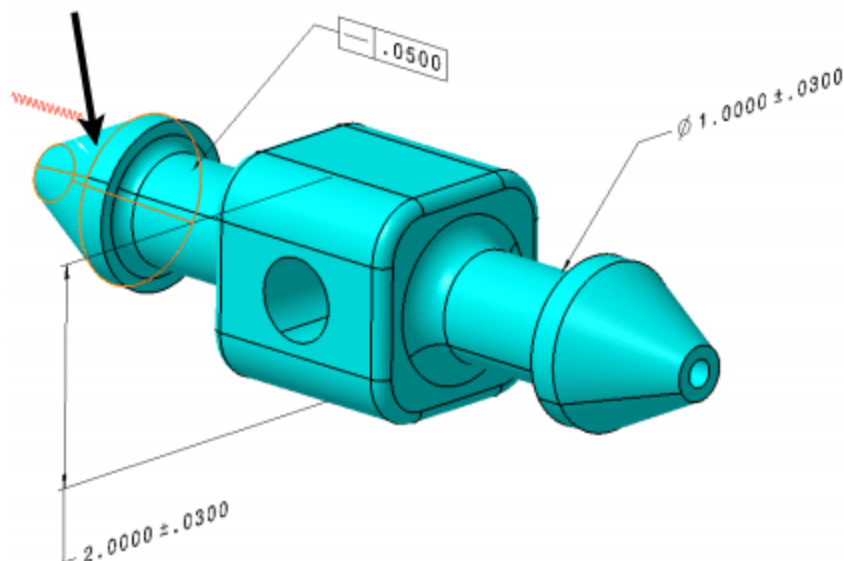



Change the specification to Straightness.





Key in 0.05 in the box next to the straightness specification and select OK. The tolerance appears and it is basically identical to the one you created with the tolerancing advisor.

Select the Geometrical Tolerance icon again and select the conical surface as shown below.  The Geometrical Tolerance window appears.

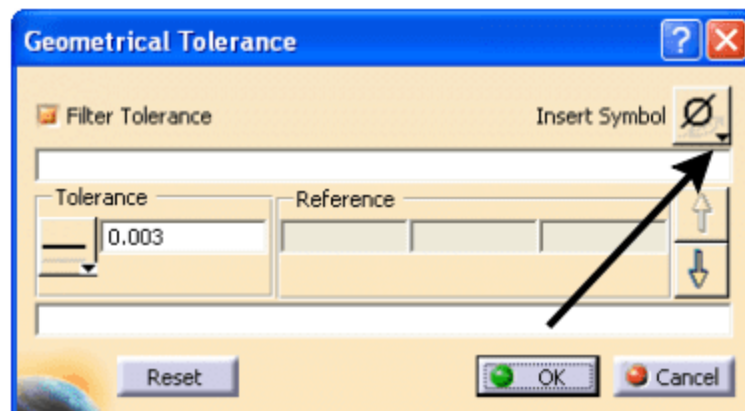



Change the symbol to Straightness with a value of 0.03 and select OK.  The tolerance appears.

Select the Geometrical Tolerance icon again and select the 1.0000 dimension.  The Geometrical Tolerance window appears.


Change the symbol to Straightness.  This time you will need to put the diameter symbol in with the value since it will not automatically do that for you like the tolerancing advisor did.

In the box next to the straightness specification, go to the left of the value and select the black arrow on the Insert Symbol icon as shown below. A bunch of symbols appear.






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

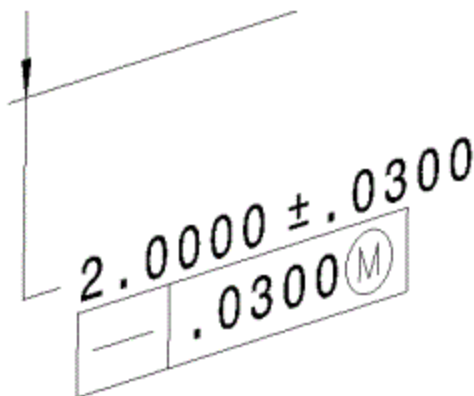
Change the value to be 0.05.

With the cursor at the right of the value 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*. Notice that the tolerance appears but it is dimmed out because it is missing the diameter symbol and it appears with an exclamation point in the specification tree. The geometrical tolerance option does not currently allow you to define an axis straightness tolerance.

Select the Geometrical Tolerance icon again and select the 2.0000 dimension.  The *Geometrical Tolerance* window appears.

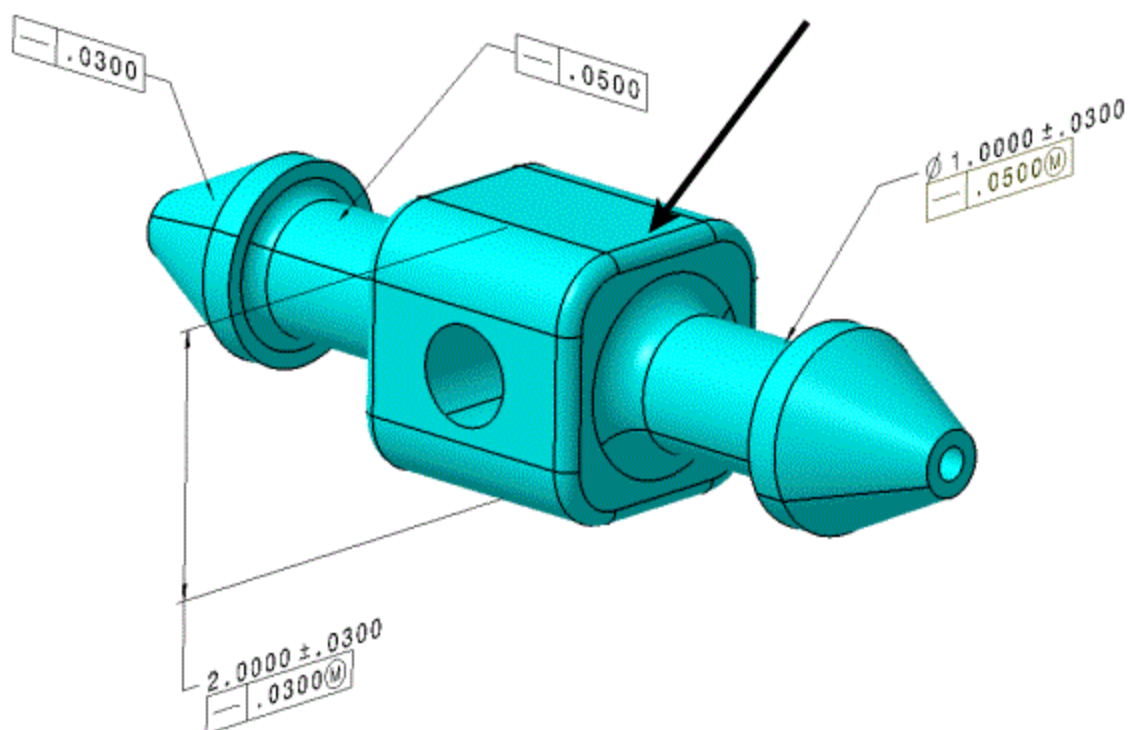
Change the symbol to Straightness with a value of 0.03 with Maximum Material Condition and select *OK*.   The tolerance appears.



A couple of things you should note are that you could not do an axis straightness tolerance and that the straightness on the center plane did not have you define a tolerance direction.

Double select the last straightness tolerance with the first mouse button. The *Geometrical Specification* window appears.

Select in the *Plane direction* box and select the edge shown below and select *OK*. In order to define the tolerance direction you have to edit the tolerance after you create it.



Close your document.