Stacked Blocks Tutorial

For ME 577

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Purpose:
In this tutorial, you will learn how to analyze the Stacked Blocks model using CE/Tol. You will analyze the model in both parametric and overlay modes.

1. Analysis Preparation

Start Up Programs:

1. On any CAEDM UNIX workstation, type "proe" in Dterm widow. Load version 2000i^2.
2. Type "cetol" in Dterm window.
3. Link CE/Tol with Pro/E (blue button in upper toolbar)
4. Select *Edit – Preferences – Analysis* from the CE/Tol pulldown menu. Verify that *Include 2nd Order Effects* is unchecked and that the *Analysis Preferences* are set to *Statistical Mode*
5. Select *Edit – Preferences – Save* to save all the set preferences

Open Stacked Blocks Assembly:

1. Copy the following files to your own directory from *cats/me577/cetol/stackblocks*:
   - **base_park.prt**
   - **cyl.prt**
   - **mblock.prt**
   - **stack.asm**
2. In Pro/E, change your working directory to the directory where you just saved the files
3. Open **stack.asm**

2. Parametric Tolerance Analysis

Define Design Specification:

The first step in defining a tolerance model is to select what needs to be studied. In the stacked blocks model the gap between the base and the top of the cylinder is the dimension of interest.

1. Select the *Add Measurement* button (the green button) in the *Assembly Network Diagram*
2. In the *Measurement Create/Edit* window assure that the spec type selected is *Gap*.
3. Select the button with a computer mouse on it (the *Pro/E select* button) under *references*

Network Diagram Summary:

In CE/Tol, network diagrams are used to see the tolerance setup in a graphical form. Joints, parts, datums etc. are all shown as icons and are interconnected to show their relationships.

The icons in the diagrams often have to be rearranged to better see them. This is done by clicking on the icon and moving it to a new location.

Icon Summary:

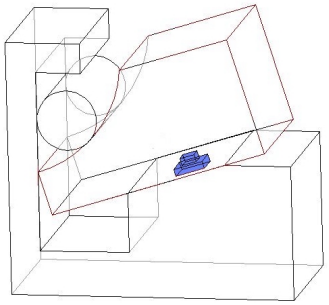
Assembly Network Diagram		Part Network Diagram	
Icon	Description	Icon	Description
	Part		Default Part DRF
	Measurement Spec		DRF
	Parallel Cylinders		Datum
	Cylinder Slider		
	Planar		
	Edge		

4. In Pro/E, select the under side of the overhang on the **base_park** part.
5. Select the curved face of the **cyl** part.
    - Note: the program will take a few seconds to identify the surfaces
6. After the CE/Tol has finished processing, verify that the nominal value is 5.945699
7. Set limit type to relative, and set limits at ± 1.5
8. Under name, type “GAP”

Define Kinematic Joints and Set Degrees of Freedom:

Now we must define how the parts fit together and which contacting surfaces control the position and orientation of the parts relative to one another.

Define the planar joint:

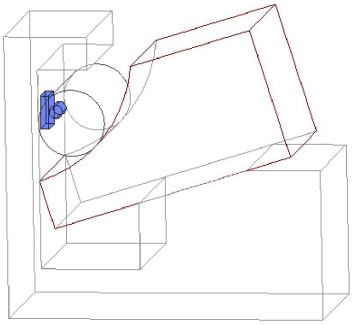


1. Select the *Add Joint* button (the yellow button) in the *Network Diagram* Window.
2. Select the *Pro/E select* button in the *Joint Create/Edit* window.
3. In Pro/E, select the slanted face of the **base_park** part.
4. Select the bottom face of the **mblock** part
    - CE/Tol will select the planar joint type
5. In the name space, type “Base_Block1”
6. Under degrees of freedom, set all the joints as fixed except in the translational z-direction (see Table 2.1)

Table 2.1

	Translation	Rotational
X	Fixed	Fixed
Y	Fixed	Fixed
Z	Kinematic	Fixed

Define the Cylinder Slider Joint:



1. Select the *Add Joint* button.
2. Select the *Pro/E select* button.
3. In Pro/E, select the curved surface of the **cyl** part
4. Select the vertical inside face of **base_park** part.
5. In the name space, type “Base_Cyl”.
6. Enter degrees of freedom according to Table 2.2

Note: If the icons are over each other in the part network diagram, click on the parts and rearrange them

Table 2.2

	Translation	Rotational
X	Kinematic	Fixed
Y	Fixed	Fixed

Z	Fixed	Fixed

Define the Edge Slider Joint:

1. Select the *Add Joint* button.
2. Select the *Pro/E select* button.
3. In Pro/E, select the top left edge of the **mblock** part (after the curved surface).
4. Select the vertical inside face of the **base_park** part.
5. In the name space, type "Base_Block2".
6. Enter the degrees of freedom according to Table 2.3

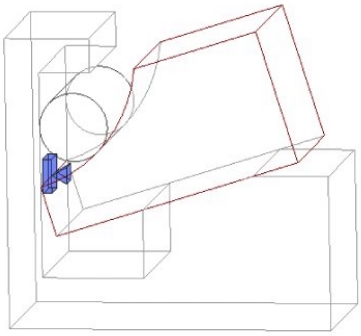


Table 2.3

	Translation	Rotational
X	Kinematic	Fixed
Y	Fixed	Fixed
Z	Fixed	Kinematic

Define the Parallel Cylinders Joint:

1. Select the *Add Joint* button.
2. Select the *Pro/E select* button.
3. In Pro/E, select the curved surface of the **mblock** part.
4. Select the curved surface of the **cyl** part.
5. In the name space, type "Block_Cyl".
6. Enter the degrees of freedom according to Table 2.4

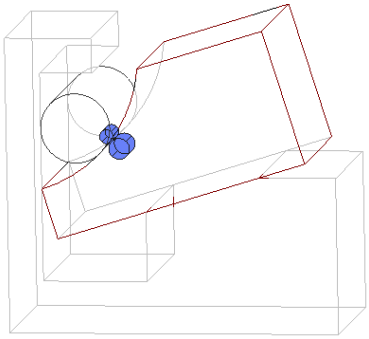
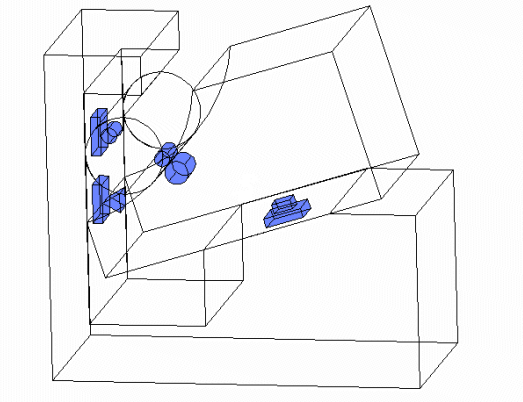


Table 2.4

	Translation	Rotational
X	Kinematic	Fixed
Y	Fixed	Fixed
Z	Fixed	Fixed

Note: When you have all of your joints constrained, your model should be over constrained by 6 degrees of freedom (it is a 2-D problem).



Once all your joints are defined, your model in Pro/E should look like Figure 2.1.

Part Modeling – Choose the Part Dimensions for the Analysis

The purpose of this exercise is to select the part dimensions on the part in Pro/E that will be used in our analysis. We will basically selecting which dimensions will be involved in the vector loops for the analysis. Choosing too few dimensions will keep you from get good results. Choosing too many dimensions doesn't hurt the analysis, but can make it take a lot longer.

Choose the Dimension for each part

1. Double-click on the **base_park** icon in the *Assembly Network Diagram* to bring up the *Part/Create Edit* window
2. Make sure that *Parametric* is selected under variables. Click on the *Get Variables* button.
3. Select *All Dimensions* from the *Get Variables* Options dialog and select *OK*
4. It will take a little time for the computer to get the full list of variables. Once the list is complete, check the boxes next to the following dimensions:
   - **d4**
   - **d6**
   - **d7**
   - **d9**
   - **d11**
5. Repeat steps 1-4 for the **mblock** part and the **cyl** part and select the following dimensions for each part:

mblock	**cyl**
d3	**d2**
d4	

Analysis – Calculate Sensitivities and Review Results

The purpose of this exercise is to calculate the sensitivities of each dimension on the Gap dimension. CE/Tol will calculate these sensitivities, and knowing them will allow us to know how much effect each dimension has on the Gap.

1. Select the *Calculate Sensitivities* button (the calculator) on the tool bar in the main menu.
2. A warning will appear on the screen that will ask if you wish to save your tolerance model before you calculate sensitivities, push *No.*
3. The analysis will take some time, depending on the speed of your computer this can be anywhere from two minutes to seven or eight minutes.
4. After the sensitivities have been calculated you will see a *Gap1 specification distribution* at the bottom of the page.
5. Select *Sensitivity Plot* from the *Analysis* menu of the main window.
6. The plot should match Figure 2.2 below:

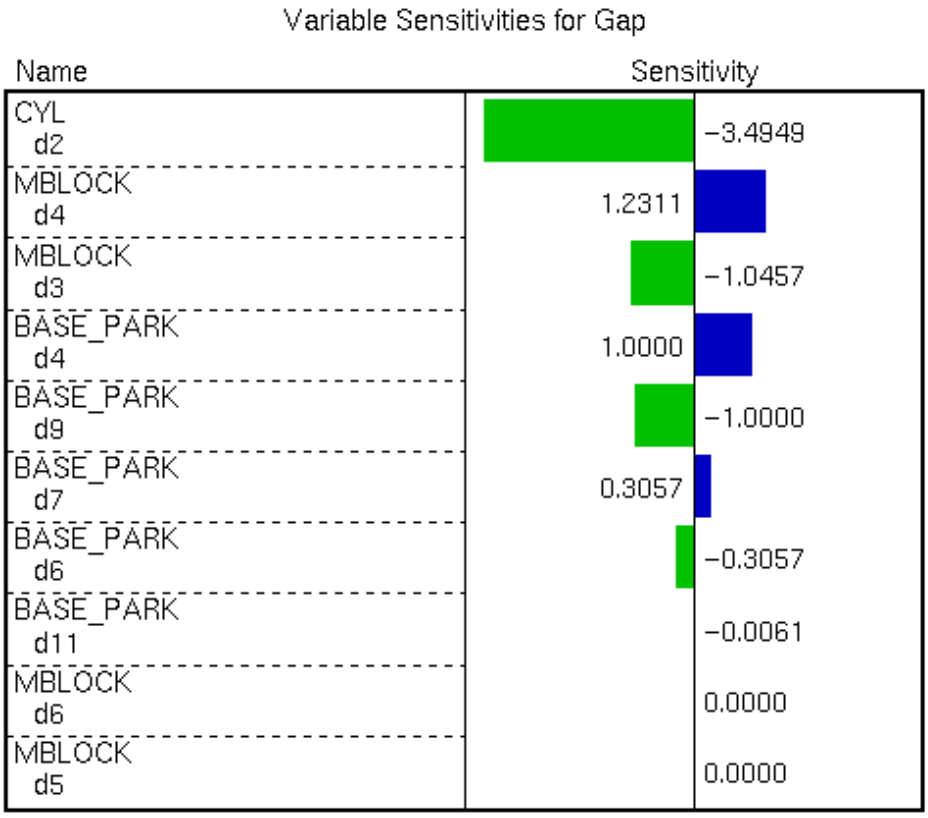


Figure 2.2

3. Overlay Tolerance Analysis

Now that you have run the analysis of the **Stacked Blocks Assembly** in parametric mode, we will learn how to do it in Overlay Mode. While Parametric Mode uses the dimensioning scheme that was developed in Pro/E, Overlay Mode alloys you to specify your own dimensioning scheme. Overlay Mode allows more freedom and does the analysis faster, but it also is more complicated.

Before we will go through the step by step process of using overlay mode, we will talk through the general process. In order to define the dimensioning scheme that you wish to use, you must first create DRFs (Datum Reference Frames) and datums that are related to the DRF. Just as in GD&T, a DRF is defined by up to three geometric elements and is an anchor for the rest of the system. In order to define dimensions we will choose datums, that when related to a DRF, give the dimension we need.

We will start with the model and CE/Tol analysis that we developed for Parametric Mode.

Changing Parts to Overlay Mode:

1. In the *Assembly Network Diagram*, double click on the **base_park** part icon

2. In the *Part Create/Edit* window, in the (the yellow button *Variables* section, check the *Overlay* box. All the variables should disappear
3. Repeat steps 1 and 2 for the **mblock** part and the **cyl** part. When Overlay is clicked for these parts, all the dimensions will disappear except one (the radius)

Establishing Dimensioning Schemes:

Establishing the Dimensioning scheme for the Base_park Part:

Creating the DRF's

1. Right click on the **base_park** part icon in the *Assembly Network Diagram* and choose *View Part Diagram*
2. In the *Part Network Diagram* rearrange the icons so that they are organized and can be easily seen. Throughout this section it will be necessary to arrange the icons
3. Press the *Add DRF* button (the white button)
4. In the *DRF Create/Edit* window, type "abc" in the *Name* box
5. Press the *Pro/E select* button
6. In Pro/E, on the **base_park** part, choose the left side, the front side, and the bottom side to define your DRF. This DRF will be you primary reference as you create datums to define your dimensioning scheme
7. In the three blanks in the References section, type "left", "front", and "bottom", one in each blank in the order that you chose them
8. In the *Part Network Diagram* press the *Add DRF* button again
9. In the *DRF Create/Edit* window, type "def" in the *Name* box
10. Press the *Pro/E select* button
11. In Pro/E, on the **base_park** part again, choose the front face, the slanted face, and the edge in between the slanted face and the vertical face. This DRF will be your secondary reference.
12. In the three blanks in the References section, type "front", "slant", and "edge", one in each blank in the order that you chose them

Creating the Datums

1. In the *Part Network Diagram*, select the *Add Datum* button (the purple button)
2. In the *Datum Create/Edit* window, press the *Pro/E select* button
3. In Pro/E, select the inside left face of the **base_park** part
4. In the name blank type "inside_left"
5. In the references section, change the DRF from *Default* to *abc*
6. Repeat steps 1 – 6 for the "inside_top", and the "inside_bottom" faces as seen in figure 3.1
7. In the *Part Network Diagram*, double click on the *edge* icon
8. In the references section, change the *DRF* from *Default* to *abc*
9. In the *Part Network Diagram*, double click on the *slant* icon
10. In the references section, change the *DRF* from *Default* to *abc*
11. Your final Part Network Diagram should look like Figure 3.2

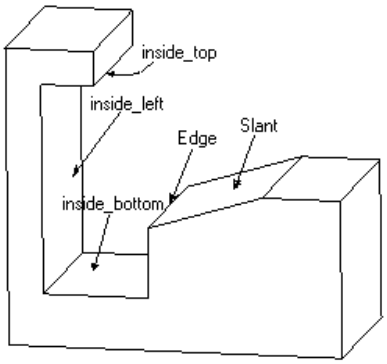


Figure 3.1

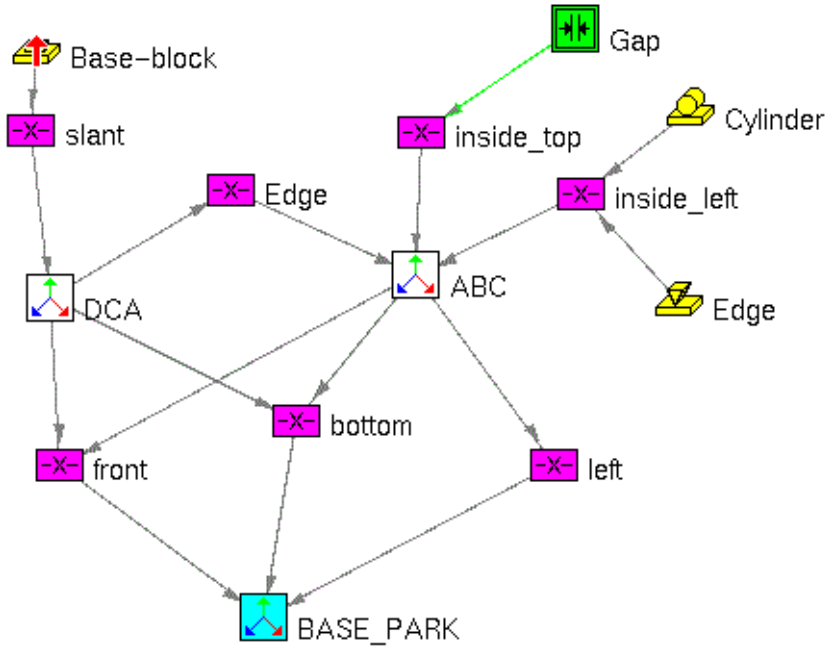


Figure 3.2

Establishing the Dimensioning scheme for the Mblock Part:

Creating the DRF's

1. Right click on the **mblock** part icon in the *Assembly Network Diagram* and choose *View Part Diagram*
2. In the *Part Network Diagram* press the *Add DRF* button (the white button)
3. In the *DRF Create/Edit* window, type "abc" in the *Name* box
4. Press the *Pro/E select* button
5. In Pro/E, on the **mblock** part, choose the left side, the front side, and the bottom side to define your DRF. This DRF will be you primary reference as you create datums to define your dimensioning scheme
6. In the three blanks in the References section, type "left", "front", and "bottom", one in each blank in the order that you chose them

Creating the Datums

All your datum's are created, you just need to modify some of the names and references (see Figure 3.3).

1. In the *Part Network Diagram*, double click on the datum that connects to the *edge* joint icon
2. In the Datum Create/Edit window, change the name from "dDatum" to "edge"
3. In the references section, change the *DRF* from *Default* to *abc*
4. In the *Part Network Diagram*, double click on the datum that connects to the *parallel cylinders* joint.
5. Change the name from "dDatum" to "Curve"

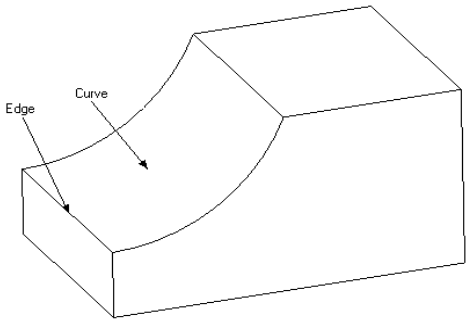


Figure 3.3

6. In the references section, change the *DRF* from *Default* to *abc*
7. Your final Part Network Diagram should look like figure 3.4

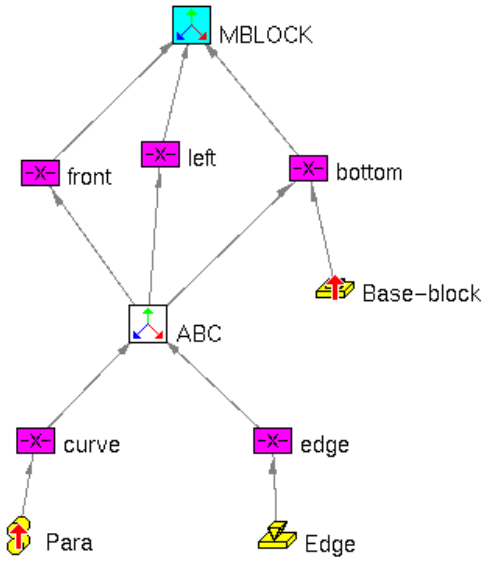


Figure 3.4

Establishing the Dimensioning Scheme for the Cyl Part:

Because of the simplicity of the part, the **cyl** part is already complete.

Part Modeling – Choose and Name the Part Dimensions for the Analysis

Naming Dimensions for the Base_Park Part:

1. In the *Assembly Network Diagram*, shift to the *Part Network Diagram* for the **Base_Park** part
2. Double click on the *Edge* datum icon
3. In the *Datum Create/Edit* window in the dimension section, expand the 40 dimension line by pressing the plus on the far left side. **Note:** The dimension numbers can be either positive or negative and it doesn't matter
4. Double click on the dimension name *TX* to bring up the *Variable Edit* window.
5. Change the name from TX to d7.
6. In Datum Create/Edit window, expand the 35 dimension
7. Double click on TY
8. In the Variable Edit window, change the name from TY to d9
9. Repeat steps 1 – 5 for all the variables according to Table 3.1

Table 3.1

Value	Old Name	New Name
Slant		
17	RX	d11
Inside_top		
75	TY	d4
Inside_left		
10	TX	d6

Naming Dimension for the Mblock Part

1. Repeat steps 1 – 5 of the last section for the Mblock part. Change your Variable names according to Table 3.2

Table 3.2

Value	Old Name	New Name
Curve		
55	TY	d3
Edge		
15	TY	d11

Choosing Variables For Analysis

1. Go back to the Assembly Network Diagram and double click on each part, putting a check mark in the square next to each important dimension for each part. The Dimensions are listed below:

Base_park	Mblock	Cyl
d4	d3	d2
d6	d4	
d7	d11	
d9		
d11		

Analysis – Calculate Sensitivities and Review Results

The purpose of this exercise is to calculate the sensitivities of each dimension on the Gap dimension. CE/Tol will calculate these sensitivities, and knowing them will allow us to know how much effect each dimension has on the Gap.

1. Select the *Calculate Sensitivities* button (the calculator) on the tool bar in the main menu.
2. A warning will appear on the screen that will ask if you wish to save your tolerance model before you calculate sensitivities, push *No.*
3. The analysis will take some time, but it will take less then for Parametric Mode.
4. After the sensitivities have been calculated you will see a *Gap1 specification distribution* at the bottom of the page.
5. Select *Sensitivity Plot* from the *Analysis* menu of the main window.
6. The plot should match Figure 3.5 below:

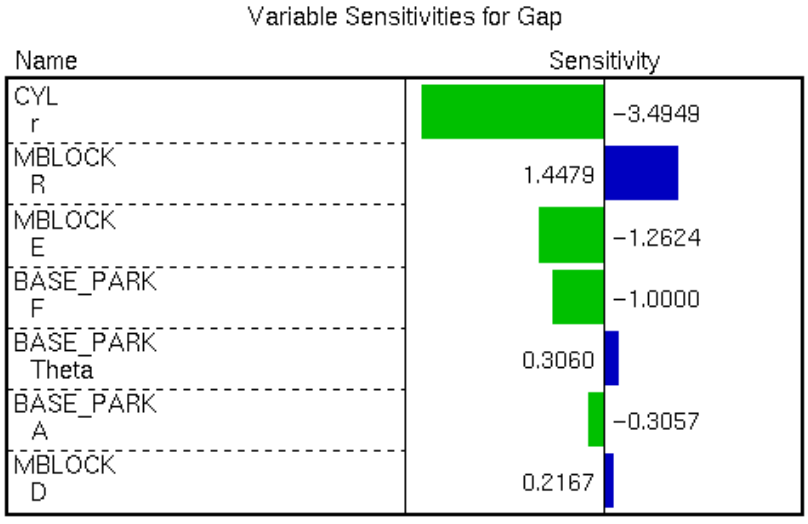


Figure 3.5

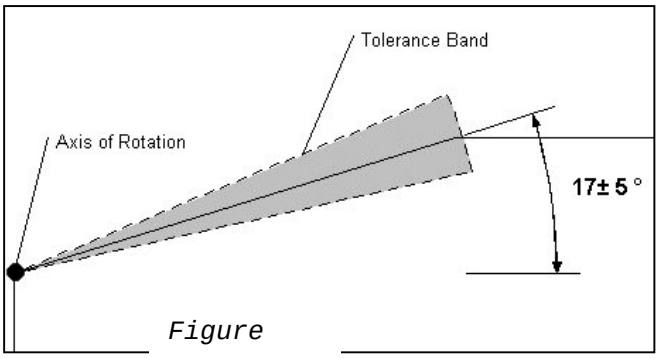
As you will notice, the sensitivities in overlay mode are not the same as the sensitivities in parametric mode. Because of the method with which CE/Tol chooses the axis of rotation of the angle, the program is solving a slightly different problem in each method. This causes the change in the sensitivities.

4. Discussion of Results

If you compare the sensitivities CE/Tol calculated in *Parametric Mode* and in *Overlay Mode*, you will notice they are not the same. When CE/Tol solves a tolerance problem, it has built in tolerance models. In the Stacked Blocks model, it makes different assumptions in *Parametric Mode* then it does in *Overlay Mode.* Because of the different assumptions, the program solved two similar, but different problems. In this section of the tutorial you will learn some assumptions CE/Tol makes and the different problems it was solving.

Background:

There are two ways to call out a tolerance on an angle. You can apply a tolerance on the angle itself, or, using GD&T you can call out an angularity tolerance zone.



Figure

When you call out a tolerance on the angle itself, you simply give a range of angles the actual angle must fall within. This creates a wedge-shaped tolerance band that radiates from the axis of rotation (see Figure 4.1).

When you use an angularity tolerance call out, you define a set of parallel lines falling on either side of the desired line. The actual line must fall somewhere in that band. This allows the line to rotate around an axis at the center of the line (see Figure 4.2).

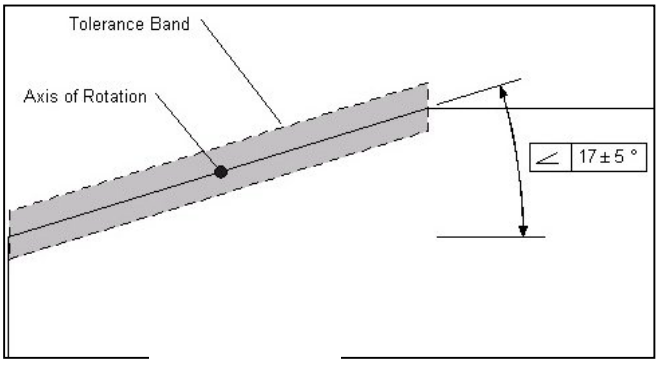


Figure 4.2

CE/Tol Assumptions:

In the analysis of the Stacked Blocks model in *Parametric Mode*, CE/Tol placed the axis of rotation of the angle at the corner. For this reason, the analysis in *Parametric Mode* modeled the angular variation as a wedge shaped tolerance zone.

In *Overlay Mode*, CE/Tol will always place the axis of rotation of the angle at the midpoint of the angled side. The analysis in *Overlay Mode* models angular variation as a rectangular tolerance zone, similar to a GD&T angularity call out.

Conclusion:

Before you start an analysis in CE/Tol, it is important to decide how you wish to tolerance the part. This decision will influence the way in which you set up the tolerance analysis in CE/Tol.

Summary Table:

Tolerance on the Angle	Angularity Tolerance
Parametric Mode	
Specify tolerance on the angle	Specify zero tolerance on the angle and add GD&T tolerance zone
Overlay Mode	
Can not do at this time	Specify tolerance on the angle

In this tutorial, we have had the computer solve the same problem using two different methods, *Parametric Mode* and *Overlay Mode*. The results differed because the representation for angular variation was not the same. Whenever an analysis is done in CE/Tol, it is important to know the method with which the problem will be solved. Your choice of mode will depend on how you wish to represent the angular variation.