

# Finding the right fit with Autodesk Inventor Tolerance Analysis Lab Exercises

Paul Munford, Autodesk

## Learning Objectives

- Why does Tolerance cause problems when assembling components?
- How can Autodesk Inventor Tolerance Analysis help?
- How to analysis native and non-native assemblies for tolerance stack up problems with Inventor Tolerance Analysis.
- How to interpret and report on Tolerance Analysis results to benefit manufacturing processes.

## Description

*Manufacturing tolerances have a significant impact on cost and quality. Tradeoffs can be made to ensure proper fit; however, calculating stack-ups manually introduces the potential for errors.*

*Autodesk® Inventor® Tolerance Analysis® is designed to help Inventor users make informed decisions while specifying manufacturing tolerances.*

*Inventor® Tolerance Analysis® is a CAD-embedded, 1D solution that helps understand the mechanical fit and performance of your design based on dimensional tolerances.*

*Stack-ups automatically capture the relationships defined in the assembly model and customers using Model-Based Definition capabilities can edit tolerance values directly inside Tolerance Analysis®, further streamlining the process between design and manufacturing*

## Speaker

**Paul Munford** is a UK based Technical Marketing Manager for the Autodesk product Design and Manufacturing Collection (PDMC).

Paul had 8 years of experience “on the tools” before joining the CAD department in 2004. Paul was a specialist joinery draughtsman (a “Setter out”) and CAD/CAM manager for a U.K. based custom furniture contractor, before joining the Autodesk Channel as an Application Engineer for Graitex UK.

Today, Paul creates awesome marketing content to show off just how cool Inventor and the PDMC really is!

Paul is the Author of the [CAD Setter Out](#) Blog and Mastering Autodesk Inventor 2016. Paul has been an attendee at AU Las Vegas since 2004 and a speaker since 2012.

[@PaulCADmunford](#)



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## How do I get Autodesk Inventor Tolerance Analysis?

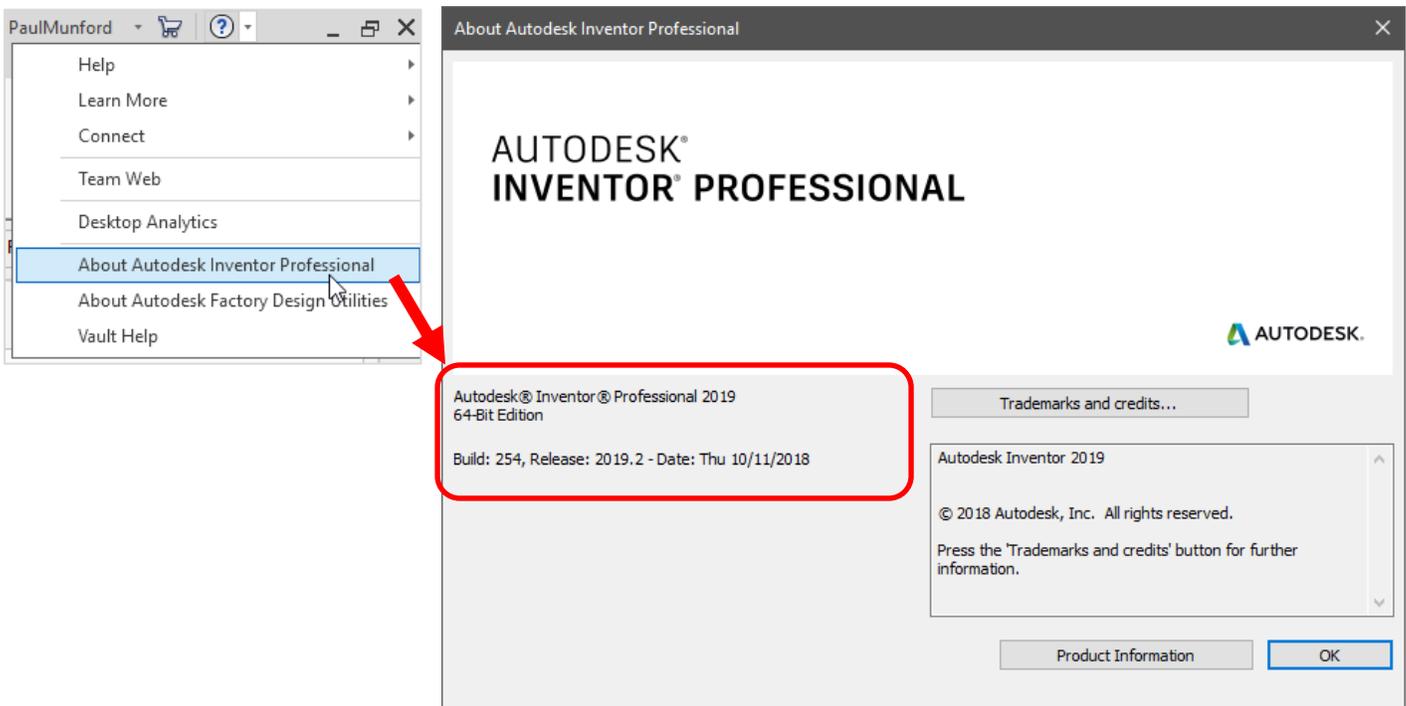
*Autodesk Inventor Tolerance analysis is exclusively available to subscribers to the Autodesk Product Design and Manufacturing Collection (PDMC).*

Autodesk Inventor Tolerance Analysis is **FREE** to subscribers.

### Prerequisites

Autodesk Inventor Tolerance Analysis Addin requires Inventor **2019.1** or later.

You can check this by clicking on the arrow next to the 'Help' icon (The question mark on the top right of the Inventor User Interface) and click on 'About Autodesk Inventor Professional'.



Online help and Release Notes:

<http://help.autodesk.com/view/INVTOL/2020/ENU/>

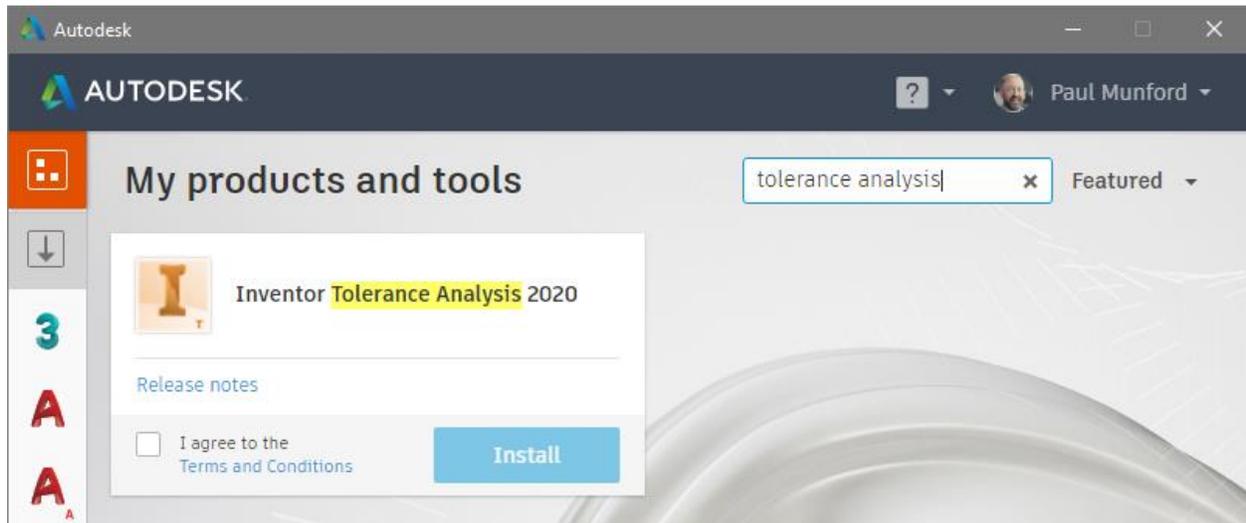
### Downloading a Trial

You can download a trial of Autodesk inventor Tolerance Analysis here:

<https://www.autodesk.com/products/inventor-tolerance-analysis/overview>

## Installing with Autodesk Desktop App

Start the Autodesk Desktop App. Scroll or search for Inventor Tolerance Analysis. Tick 'I agree to the terms and conditions', click the 'Install' Button.

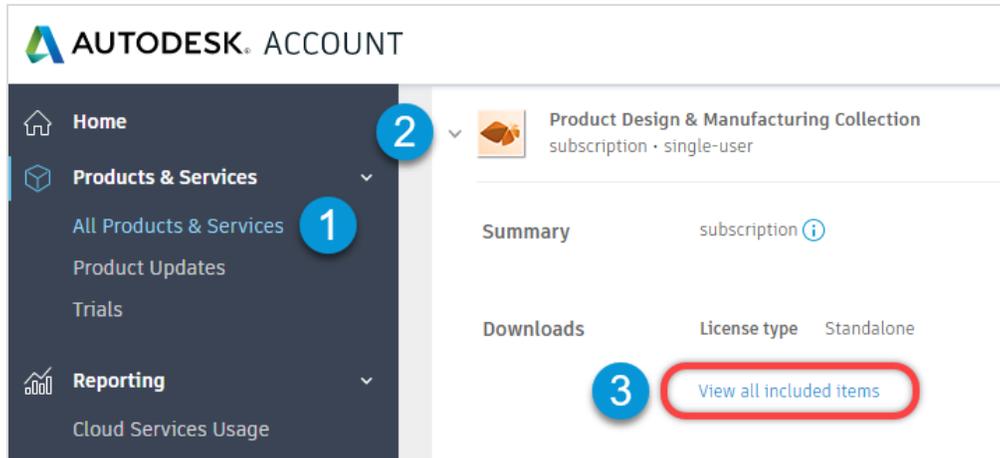


## Downloading from your account

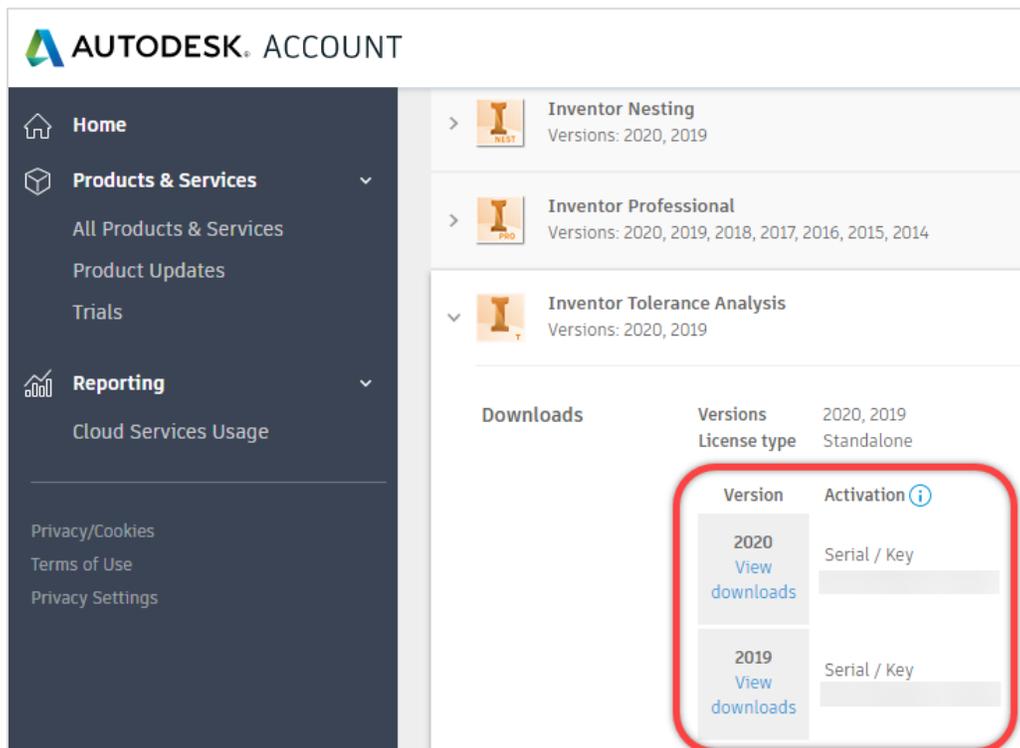
Navigate to your Autodesk Account:  
<https://manage.autodesk.com/home/>

**Note:** If you are not your companies Autodesk Account Administrator, you may need to ask your account Admin to grant you access first.

1. Click on 'All Products and Services'.
2. Search for 'Product Design & Manufacturing Collection', Click on the Chevron to expand the entry,
3. Click on 'View all Included items'



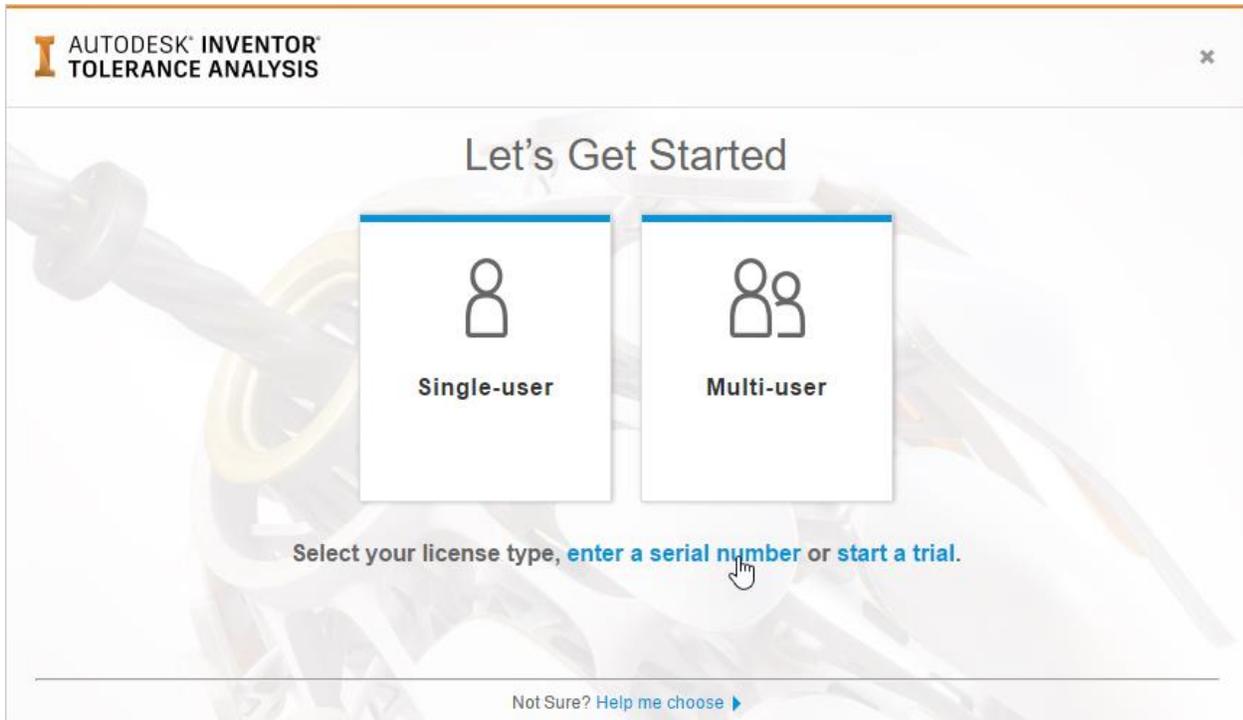
Click on the links to download the appropriate version. Don't forget to take note of your serial number and Product Key!



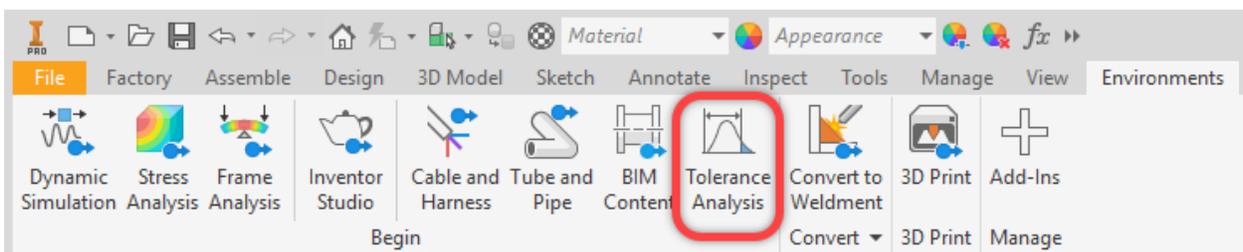
## Activation

When prompted, start a trial or Click single user. You may be prompted to enter your serial number.

If your company has a server-based licence, click 'Multi-User'.



You will find the Tolerance analysis tools under the 'Environments' tab:



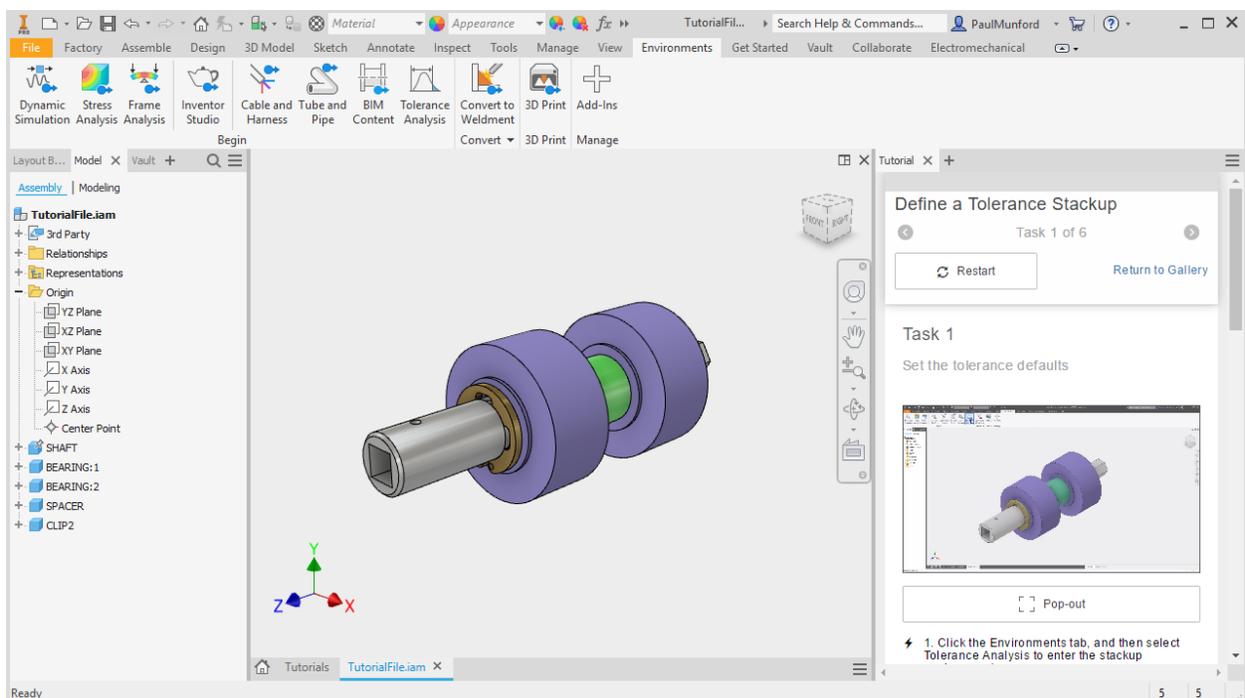
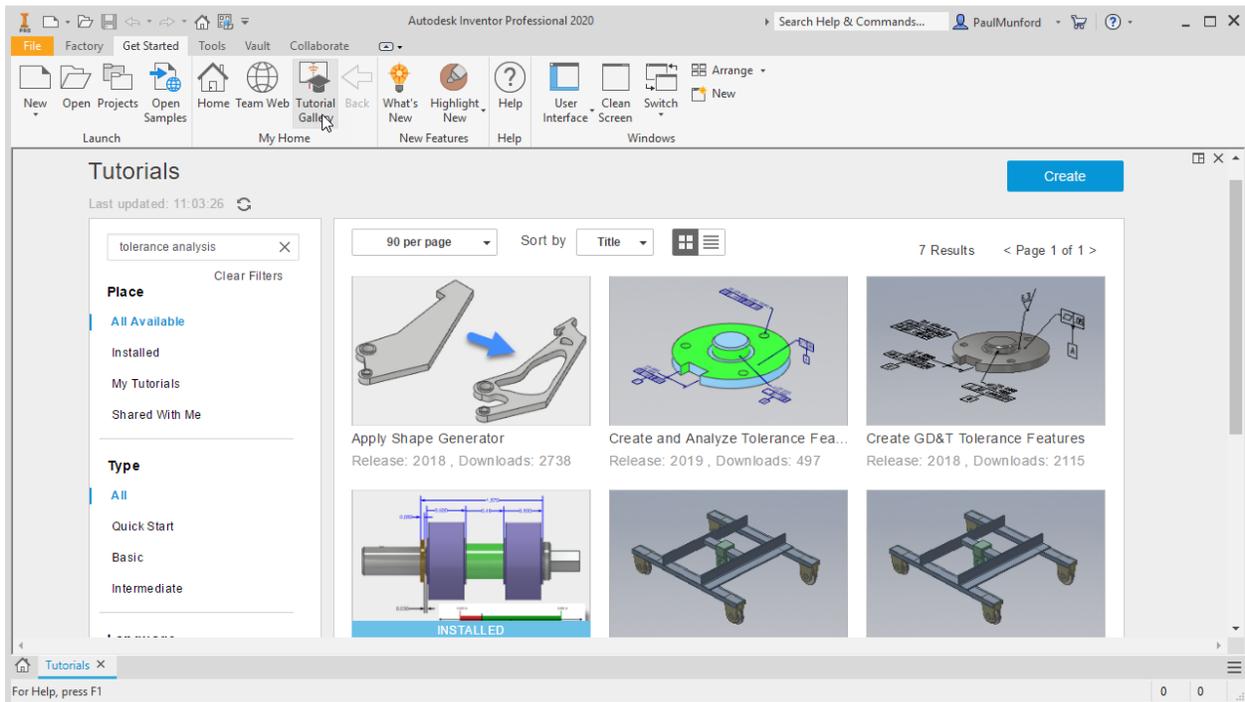
## Learning Materials

Try the guided tutorial - step by step instructions with Videos and staged datasets, built into Inventor.

Click on 'Get Started' Tab > 'My Home' Panel > 'Tutorial Galley' Button.

Search for 'Tolerance Analysis'.

Click on the Image to download and 'Play' the Tutorial.



# Lab Exercises

## Data set

The data set for the lab can be found on the computer you are sitting at:

C:\Datasets\MFG318994-L Finding the right fit with Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis

**Note:** Please feel free to take a copy of this dataset. You will be able to download the Dataset, the Exercises handout and the Presentation from the Autodesk University website after the class has finished.

[Autodesk University Online – Search ‘Tolerance Analysis’ – Product ‘Inventor’](#)

## Watch the Screencasts

each of these exercises has been recorded as an Autodesk Screencast.

You can find the Screencasts here:

<https://knowledge.autodesk.com/community/collection/autodesk-inventor-tolerance-analysis>

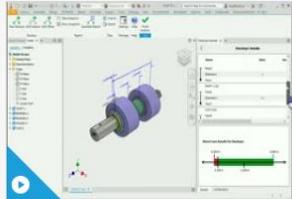
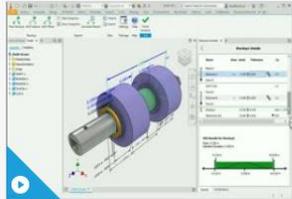
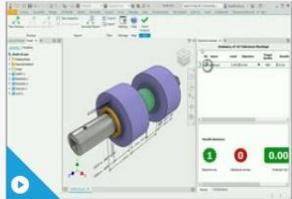
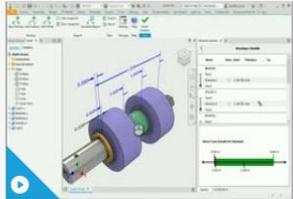
## Autodesk Inventor Tolerance Analysis

 My Collection  
Products and versions covered ▾

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Autodesk Inventor Tolerance Analysis Screencast Videos created for the Autodesk University lab: Finding the right fit with Autodesk Inventor Tolerance Analysis; 01 - Create a stackup 02 - Analyze the stackup 03 - Generate a Report 04 - Create a Manual Stackup 04 - Extra Credit Exercise, Infer a loop via constraints 05 - Challenge 01, Create a stackup from a dataset with relationships and GD&T 3DA 05 - Challenge 02, Create a stackup from an AnyCAD referenced Dataset



 <p><b>Autodesk Inventor Tolerance Analysis 01</b></p> <p>Jun 5, 2019</p> <p>This Screen cast has been recorded as part of an Autodesk University Lab...</p>	 <p><b>Autodesk Inventor Tolerance Analysis 02</b></p> <p>Jun 5, 2019</p> <p>This Screen cast has been recorded as part of an Autodesk University Lab...</p>	 <p><b>Autodesk Inventor Tolerance Analysis 03</b></p> <p>Jun 5, 2019</p> <p>This Screen cast has been recorded as part of an Autodesk University Lab...</p>	 <p><b>Autodesk Inventor Tolerance Analysis 04</b></p> <p>Jun 6, 2019</p> <p>This Screencast has been recorded as part of an Autodesk University Lab...</p>
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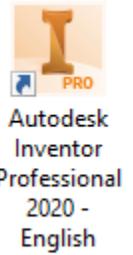
## Before we begin...

Before we test out Tolerance Analysis – let's get ready. First, we need to:

- Start Inventor
- Set the Project file

### Start Inventor

- Double click with your left mouse button (LMB) on the Inventor 2020 icon on your desktop to start the application.



### Set the Project File

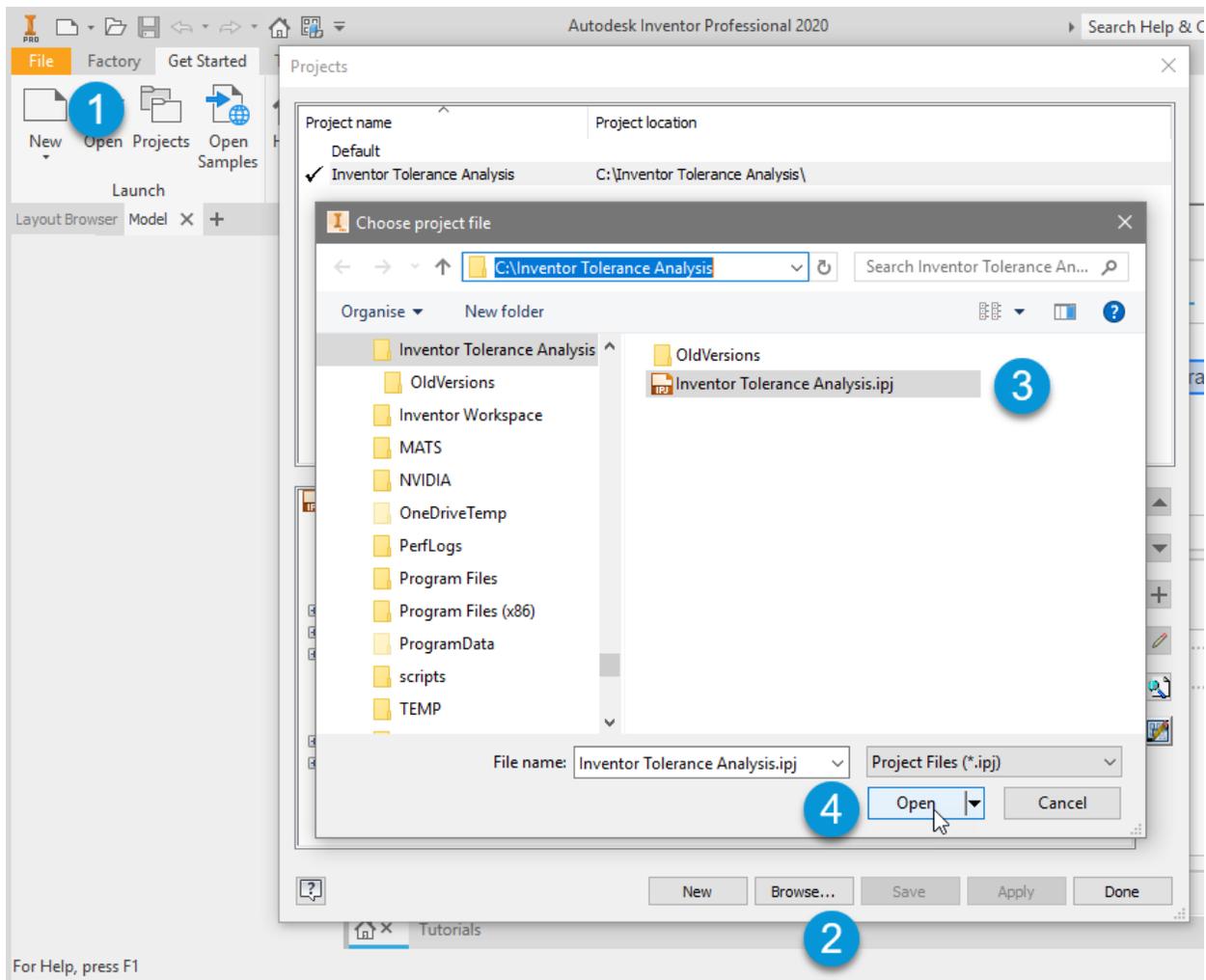
Navigate to:

*'Get started' Tab > 'Launch' Panel > 'Projects' Button*

C:\Datasets\MFG318994-L Finding the right fit with Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis

LMB to select the **'Inventor Tolerance Analysis.ipj'** file.

Click on 'Open' to set this Inventor project file to be the current project.



*Tip: On the Inventor Home screen > Projects Panel – you will see a Green Tick next to ‘Inventor Tolerance Analysis’.*

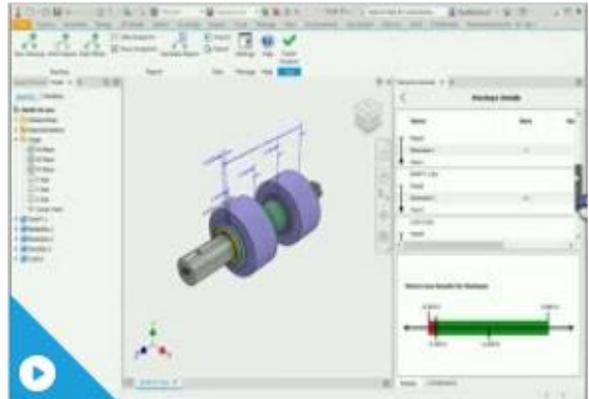
## Exercise 01 – Create a Stack up Analysis

In this Exercise we will:

- Open the data set for this exercise
- Navigate to the Tolerance Analysis Environment
- Look for the Gap
- Understand the Loop
- Create a stack up

*Watch the Screencast:*

[Finding the Right Fit with Autodesk Inventor Tolerance Analysis Exercise 01](#)



### Open the data set for this exercise

Navigate to:

*‘Get Started’ Tab > ‘Launch’ Panel > ‘Open’ Button*

Navigate to:

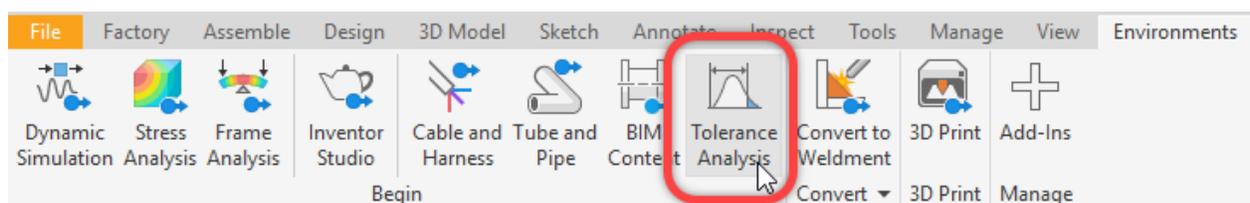
C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 01

LMB select **‘Shaft-01.iam’** (Inventor assembly file) and click on the ‘Open’ button.

### Navigate to the Tolerance Analysis Environment

Navigate to:

*‘Environments’ Tab > ‘Begin’ Panel > ‘Tolerance Analysis’ Button*



### Look for the Gap

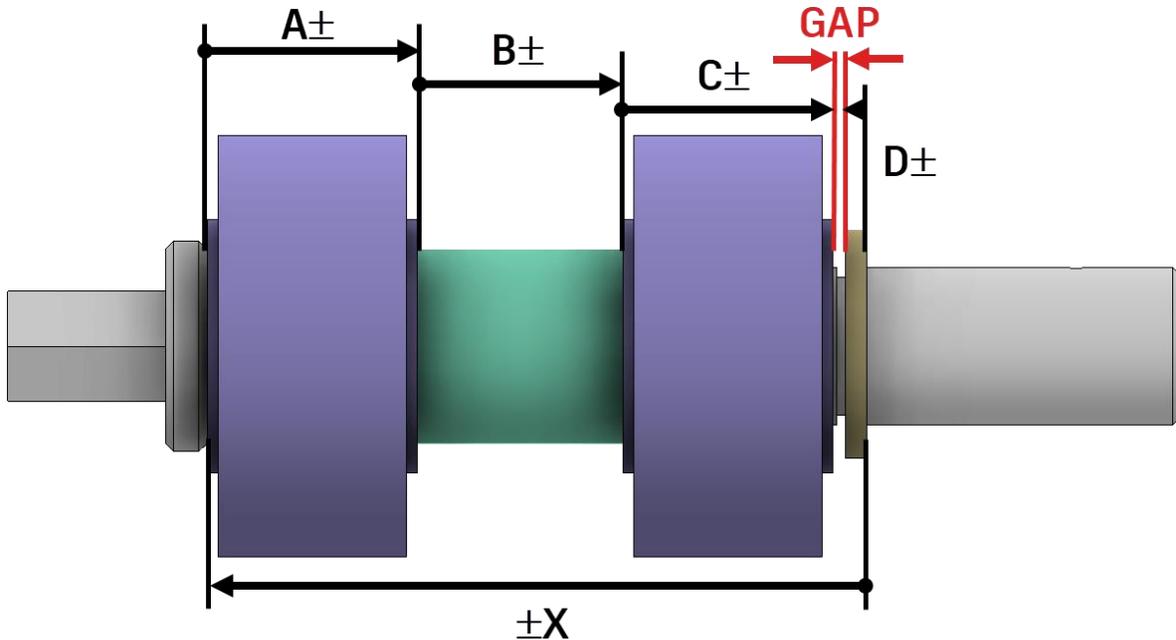
The first step in the analysis is to decide which gap we are going to analyse. For this exercise we will analyse the gap between the Clip and the Bearing

**Note:** When you try this with your own dataset, the ‘Gap’ you want to examine could be modelled at a nominal dimension of zero.

## Understand the Loop

When analysing a 1D tolerance, we are comparing the total tolerances applied to one set of dimensions, with the total tolerances applied to a second set of dimensions.

We can express this relationship with a diagram, showing the dimensions as a 'Loop'.



For this exercise the dimensions we will compare are the distance on the shaft between the shoulders, with the thickness of the two bearings, the shaft and the clip.

It's worth understanding the minimum and maximum material condition of the components.

The worst-case conditions for our assembly will be:

The Shaft manufactured at the smallest possible size, and the two bearings, shaft and clip manufactured at the largest possible size

*This combination will result in the smallest Gap.*

At the other extreme the Shaft is manufactured at the largest possible size, and the two bearings, shaft and clip manufactured at the smallest possible size

*This combination will result in the largest Gap.*

## Create a stack up

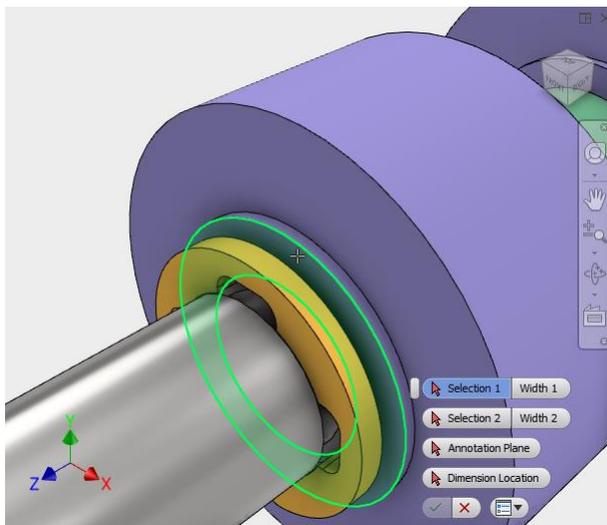
Navigate to:

'Tolerance Analysis' Tab > 'Stackup' Panel > 'New Stackup' button.

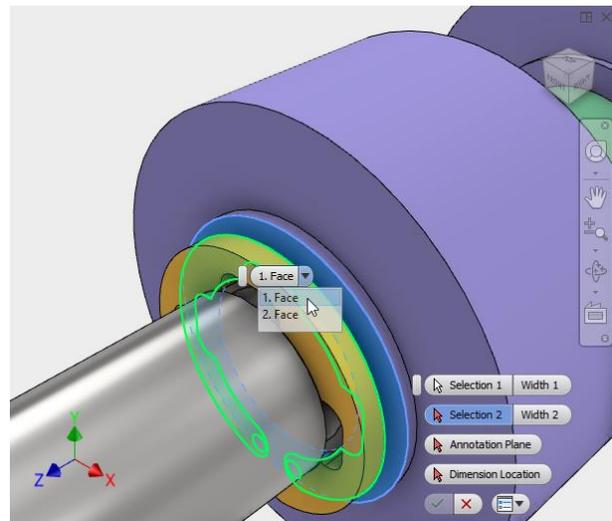
'Selection 1' is the front face of Bearing:2

'Selection 2' is the back face of Clip:5

**Tip:** Hover your mouse over the edge of the clip. After a pause the 'Select Other' drop down will appear. Click on the black drop-down arrow at the end of the selector to pick from the faces that were under your mouse when you paused.



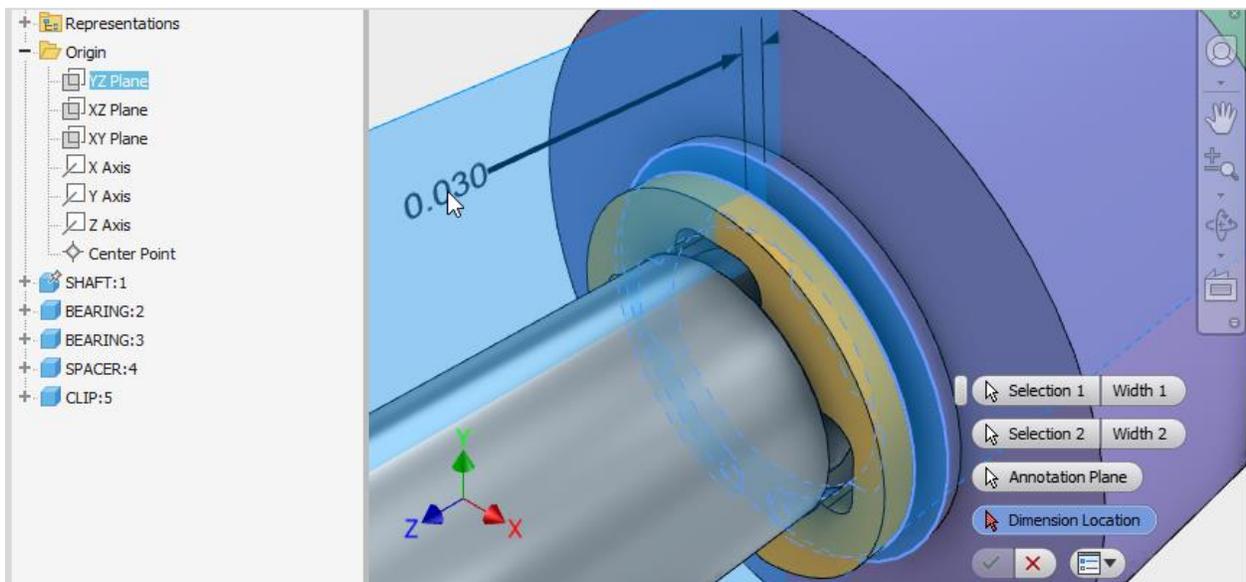
First Selection



Second Selection

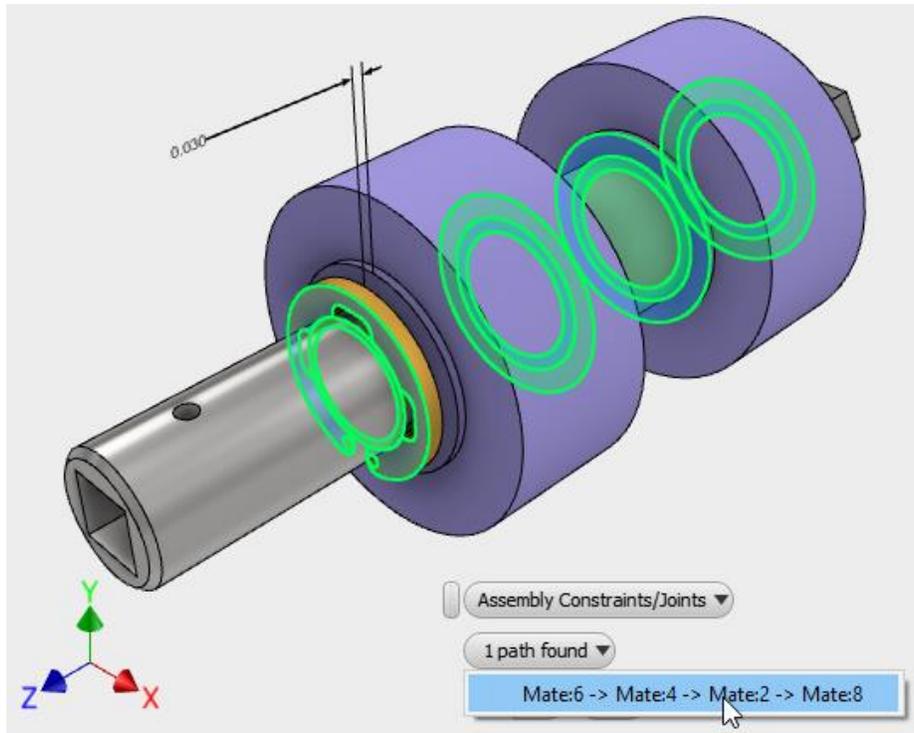
For 'Annotation Plane', navigate to the browser. Next to the 'Origin' Folder, click on the '+' symbol to expand the Origin Folder. Select the YZ Plane for your Gap Annotation.

Back in the graphics windows, move your cursor and LMB click to place the Gap Dimension Annotation.



Inventor Tolerance Analysis will now analyse the assembly relationships (Joints and Constraints) to see if it can find a loop.

Navigate to the in-canvas menu Path Selector – the button has the text '1 Path found'. Click on the black triangular drop-down selector at the end of the button.



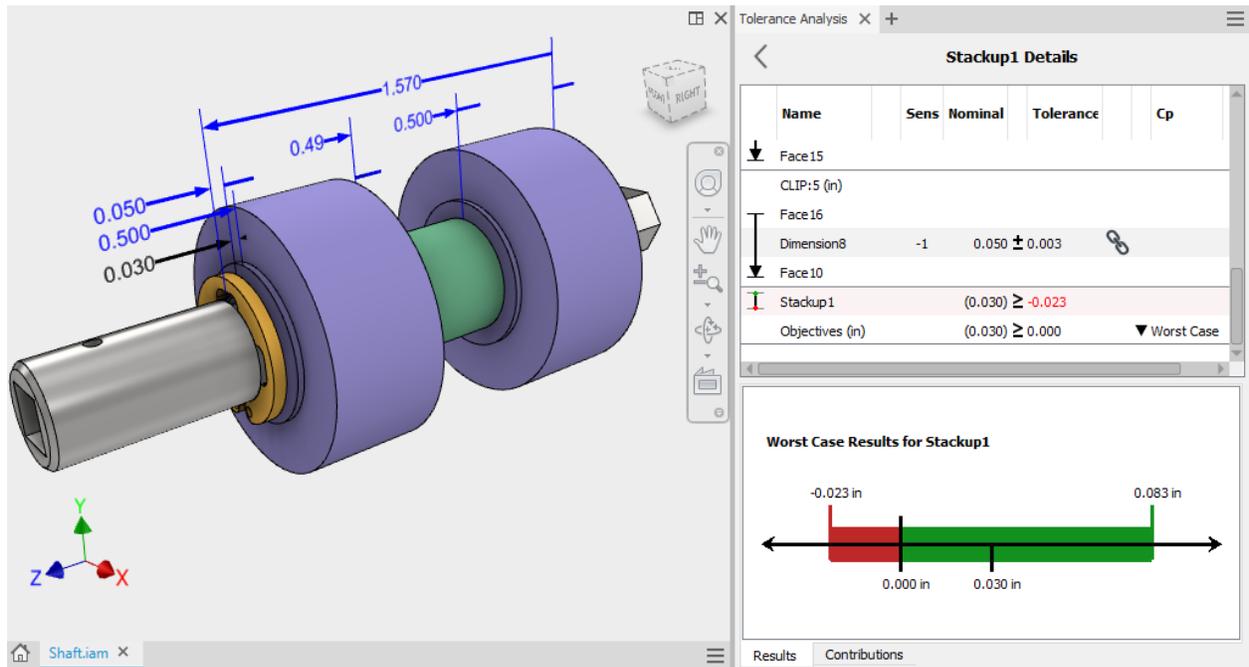
In this case, only one loop has been detected:

'Mate:6 -> Mate:4 -> Mate:2 -> Mate:8'

Select the loop and click the button with the Green 'Tick' ['Check'] ✓ button to complete the stack up.

*Save your work if you want – we will use the pre-prepared dataset for the next exercise.*

*Close the file before moving on to the next exercise.*



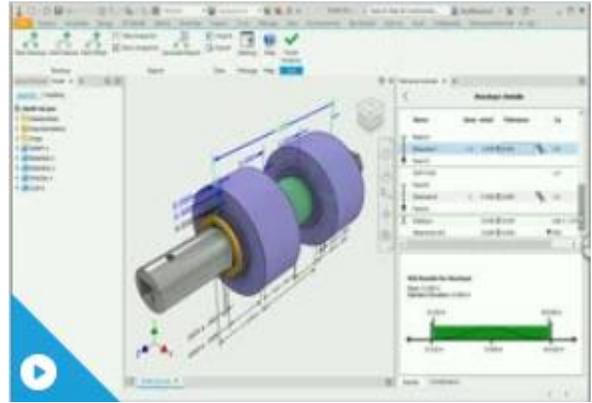
### Exercise 01 Extra Credit

- Adjust the positions of the Gap Annotation and the stack up annotations. Hover over the annotation until the extension lines highlight, then LMB click+drag your cursor to place the annotations.
- Create a saved view representation.
- Save your work if you want – we will use the pre-prepared dataset for the next exercise.
- Close the file before moving on to the next exercise.

## Exercise 02 – Analysing the Stackup

- Understand the Objective
- Change the Objective
- Find out which dimensions contributes most to the worst-case analysis.
- Adjust the tolerances to adjust the results.
- Worst Case vs RSS

Watch the Screencast:  
[Finding the right fit with Autodesk Inventor Tolerance Analysis Exercise 02](#)



### Open the data set for this exercise

Close the dataset you had open for the previous exercise.

Navigate to:

'Get Started' Tab > 'Launch' Panel > 'Open' Button

Navigate to:

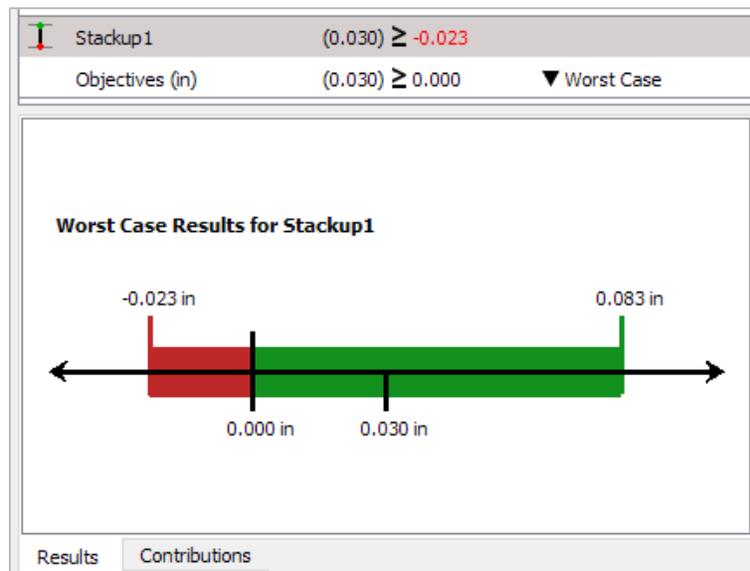
C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 02

LMB select '**Shaft-02.iam**' (Inventor assembly file) and click on the 'Open' button.

Enter the Tolerance analysis environment.

### Understand the Objective

The 'Worst Case' results graph shows us the minimum and maximum sizes of the Gap.



If the Bearing, Spacer and Clip are manufactured to the smallest size allowed by their individual tolerances, and the Shaft is manufactured to the largest size allowed, the Gap will be **0.083 in**.

If the Bearing, Spacer and Clip are manufactured to the largest size allowed by their individual tolerances, and the Shaft is manufactured to the smallest size allowed, the Gap will be **-0.023 in** (a clash!).

The nominal size of the Gap (as modelled) is **0.030 in**.

The current objective assumes that the Gap must be a minimum of **0.000** - with no maximum limit.

### Change the Objective

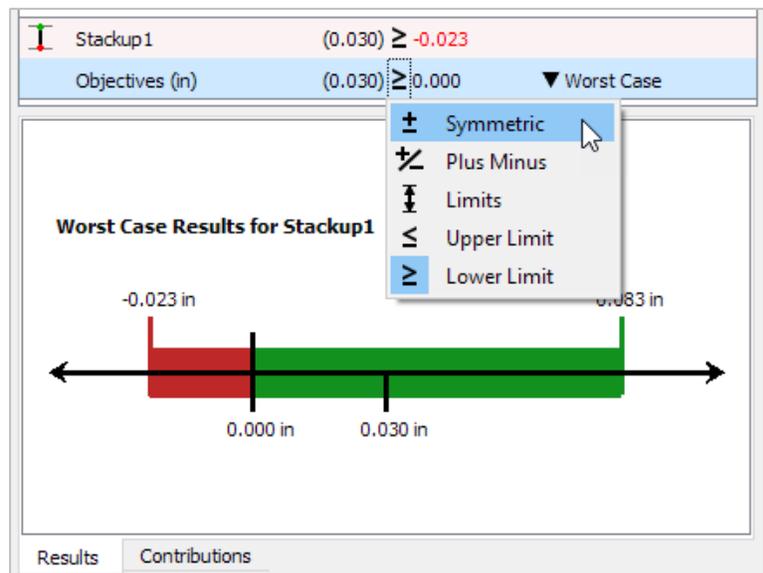
We would like the dimension of the Gap to be **0.03 in** - within a tolerance of **0.01**

Navigate to:

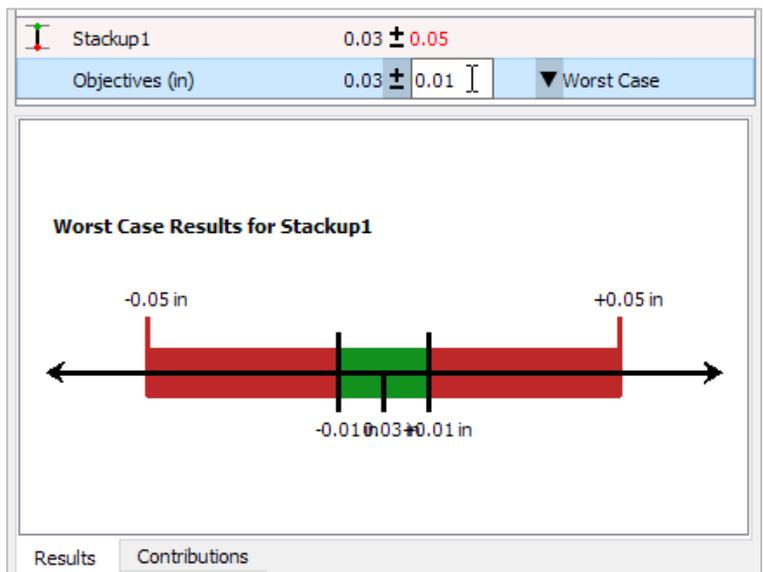
*Tolerance Analysis Panel > 'Objectives (in)' row.*

Click on the 'Equal to or Less than'  $\geq$  **Lower Limit** symbol to choose an objective.

Pick  $\pm$  **Symmetric** from the drop-down menu.



Change the value to give a Tolerance of **0.01**



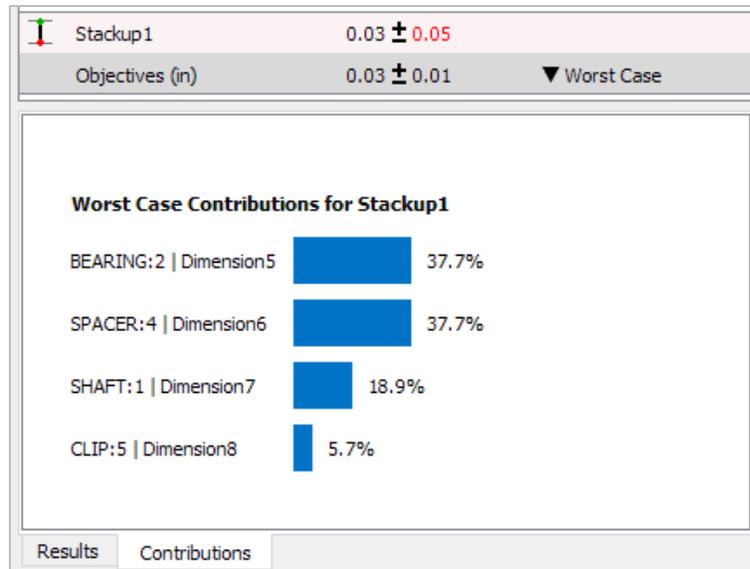
## Which dimensions contribute most to the worst-case analysis?

We can see that the tolerancing applied to our component will cause the components manufactured at the minimum and maximum extremes of the tolerances, to fall outside our desired Gap.

*But which tolerances should we change to meet our desired result?*

Navigate to:

*Tolerance Analysis Panel > 'Contributions' tab.*



We can see that the Bearing and the Spacer are the biggest contributors.

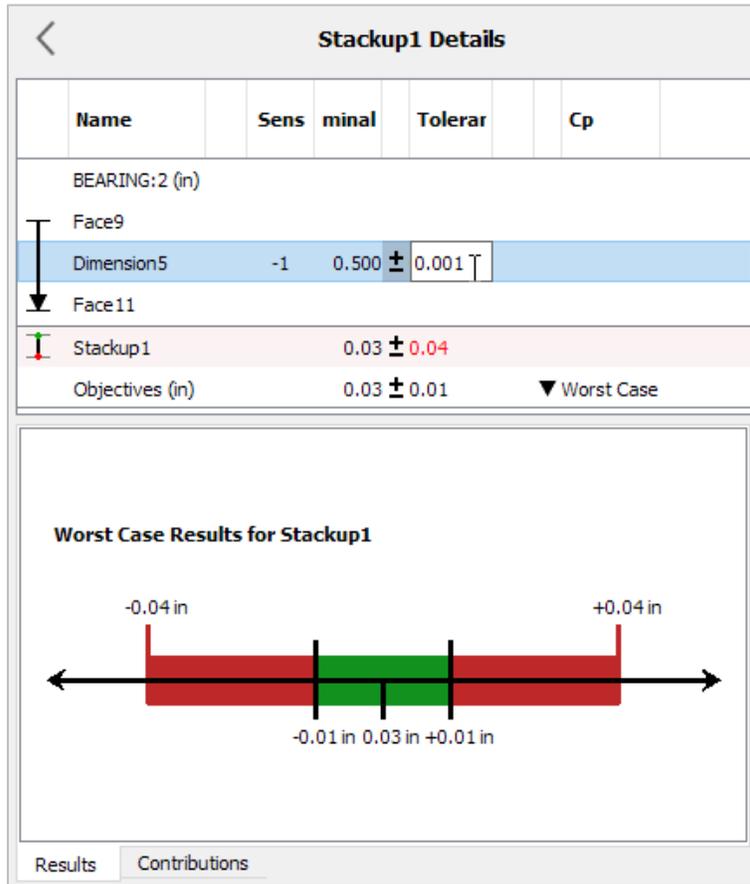
There are two bearings, so let's tweak the tolerance on these first.

Switch back the Results tab, then scroll up and down the Tolerance Analysis Panel until you find the entry for **Bearing:2**.

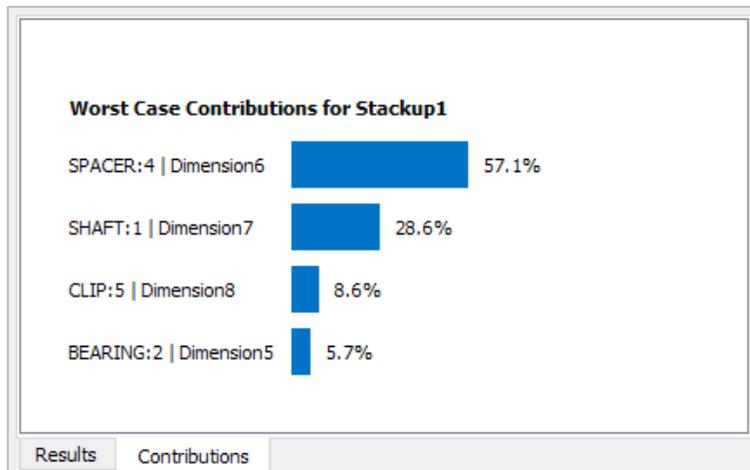
**Tip:** *Bearing:2 and Bearing:3 both reference the same part. Changing one entry will change the tolerance on both parts.*

Change the tolerance on the Bearing:2 (in) > Dimension5 from **0.010** to **0.001**

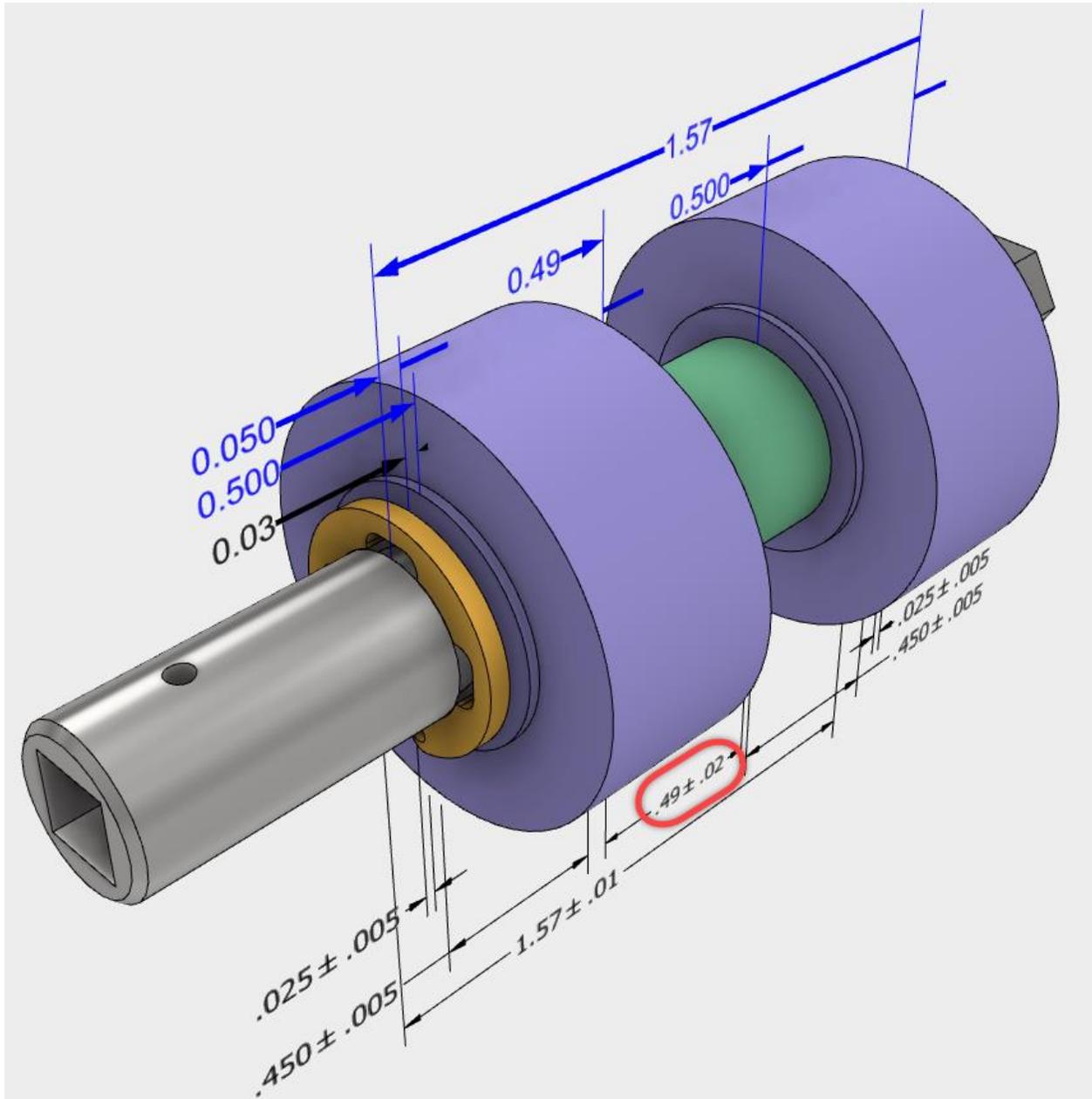
*Notice the Results Graph update as you make the change.*



Switch back to the 'Contributors' tab. Spacer:4 is now the main contributor.



Look at the Graphics window and find the 3D Annotation for the spacer.

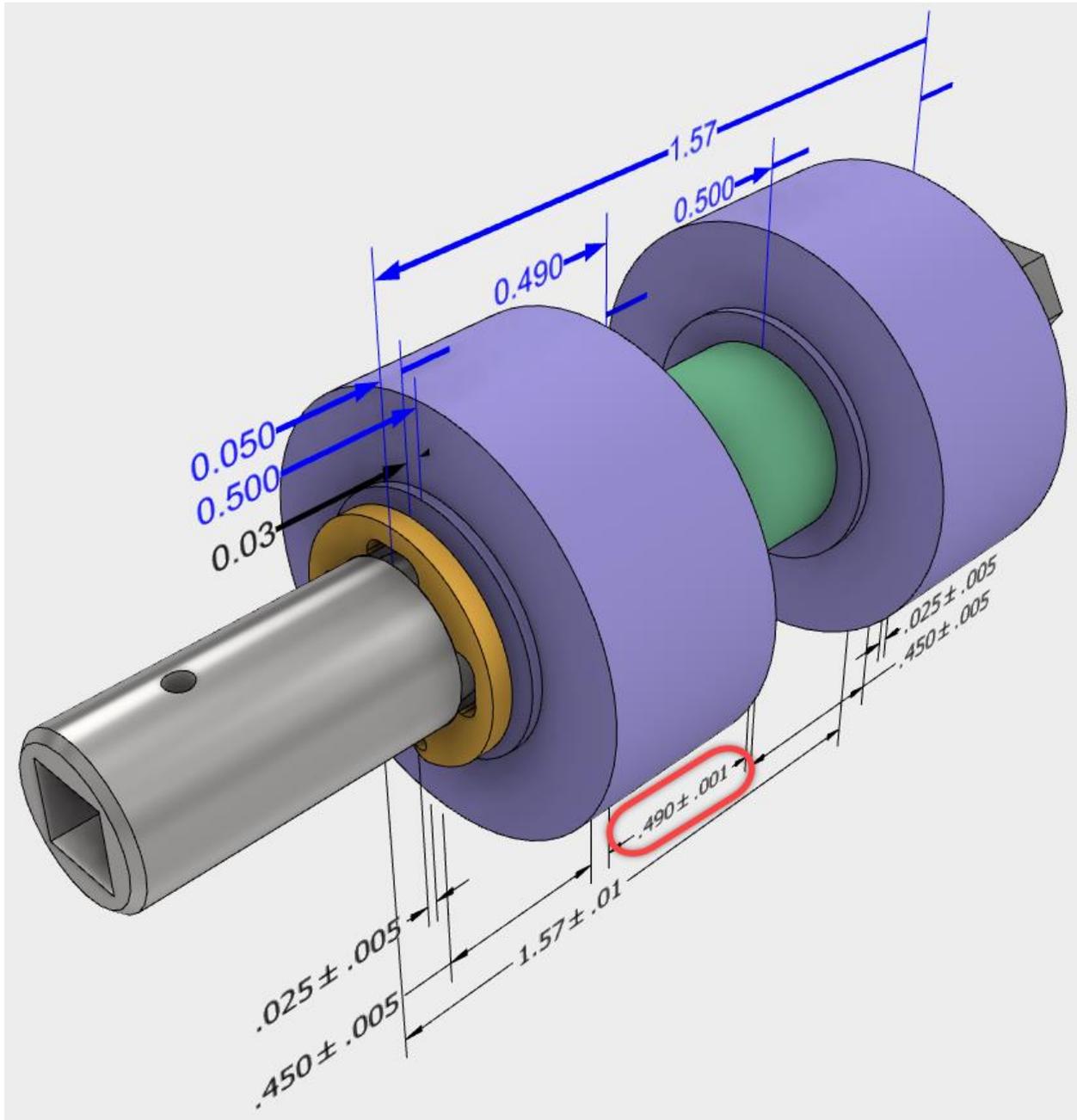


Switch back to the Results tab, scroll to Spacer:4 > Dimension6 and change the tolerance from **0.020** to **0.001**

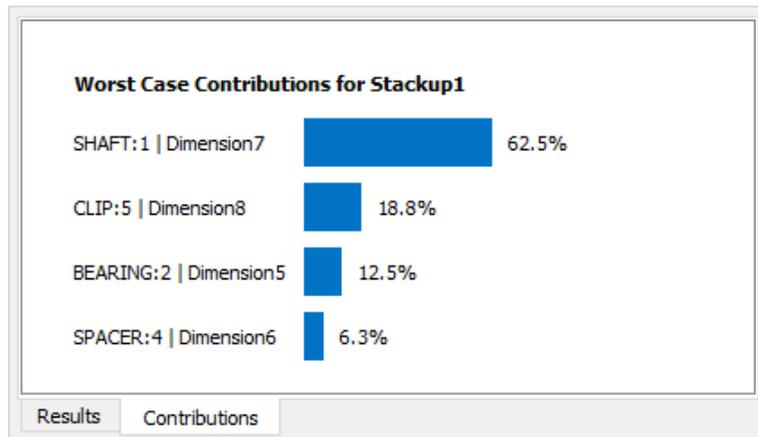
**Note:** Notice the 'Chain' icon next to Spacer:4 > Dimension6. This means that the value in the Tolerance Analysis panel is linked to the tolerance on the 3D Annotation in the part. When you change the tolerance in the Tolerance analysis panel, the 3D annotation in the part file is also updated.

SPACER:4 (in)			
Face12			
Dimension6	-1	$0.490 \pm 0.001$	
Face13			

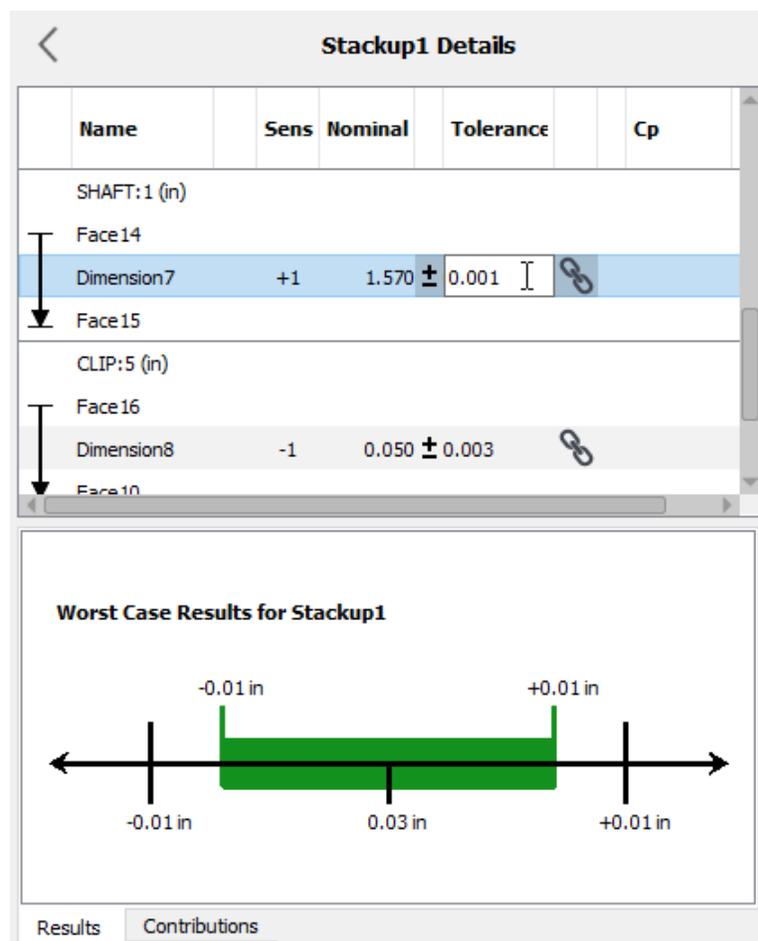
Notice that the 3D Annotation for the spacer is in sync with the adjustment made in the Tolerance Panel.



Switch back to the Contributions tab. The main contributor is now the shaft.



Switch back to the Results tab, scroll to Shaft:1 > Dimension7 and change the tolerance from **0.010** to **0.001**



The worst-case results graph shows that the tolerances applied to the individual parts now fall between the tolerances we specified for the Gap.

This means that, if we batch manufacture many of these components to the tolerances specified, no matter if the components are all manufactured to the maximum material condition, or the minimum material condition, they will always fit together with a gap of  $0.3 \text{ in} \pm 0.01$

## Worst Case vs RSS

A 'Worst case Stack'-up is the easiest Tolerance Analysis to calculate, and the one that we are most familiar with.

You may already have a spreadsheet for calculating stack ups, or you may calculate them by hand (or in your head!).

However, worst case tolerance analysis can lead us to over specify the tolerance on our components. We could save a great deal of money if we loosen up those tolerances, so it's worth taking an additional step.

Navigate to:

*Tolerance Analysis Panel > 'Objects (in)' Row > 'Worst Case' Drop down.*

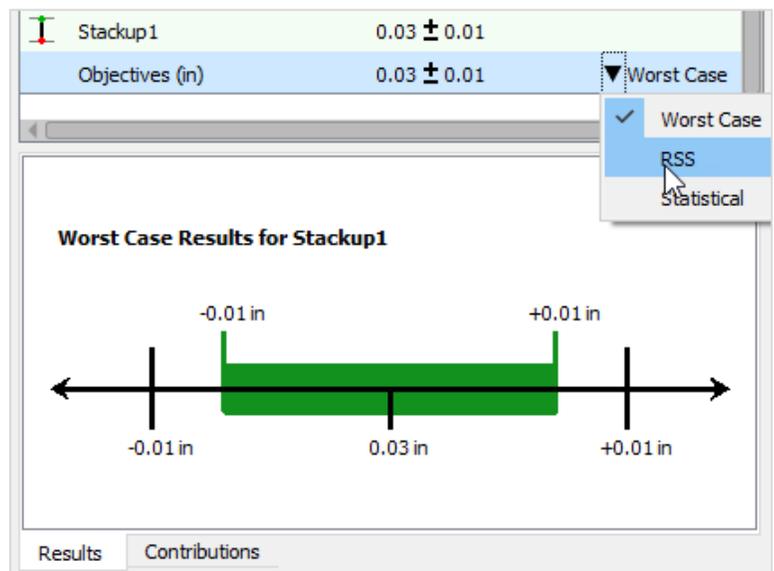
LMB click on the black drop-down arrow and click 'RSS'.

'Worst case' analysis assumes that ALL the components manufactured will ALL be at the maximum or minimum tolerance.

In real life, this rarely happens. In fact, components coming off the machine slowly 'creep' from the nominal size to the extreme of tolerance allowed.

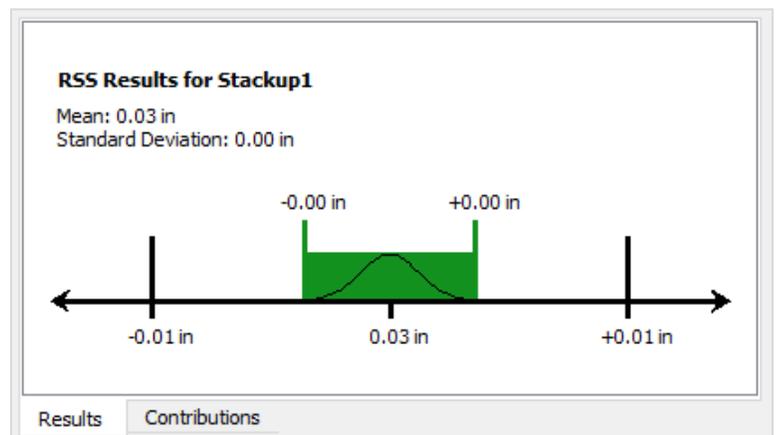
The person picking the components for assembly, will be picking from a bin of components – each a slightly different size.

*This will happen for every component.*



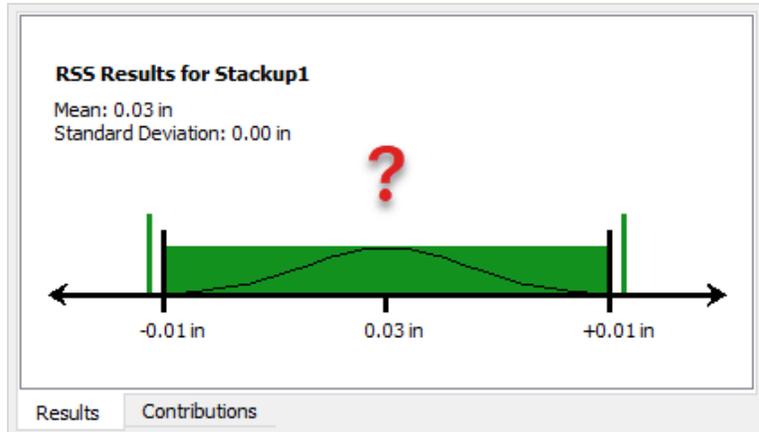
Some components will be at their minimum tolerance. Some will be at their maximum, most will be somewhere in between.

The RSS Results graph shows us the *probability* of each component being manufactured at the extreme tolerance and coming together with other components which are also at the extreme.



### Exercise 02 Extra Credit

- Why doesn't Bearing:2 Dimension5 have a 'link' symbol next to it?
- Using the worst-case results graph, what is the largest Plus Minus tolerance you can allow on SHAFT:1 > Dimension7 and still fall within the tolerance zones specified?
- Using the RSS results graph, what is the largest Plus Minus tolerance you can allow on SHAFT:1 > Dimension7 and still fall within the tolerance zones specified?
- Using the RSS results Graph, experiment with the tolerances on the Bearing, Shaft and Spacer until you are happy with the result.
- Switch back to the Worst-case results graph and compare the results.



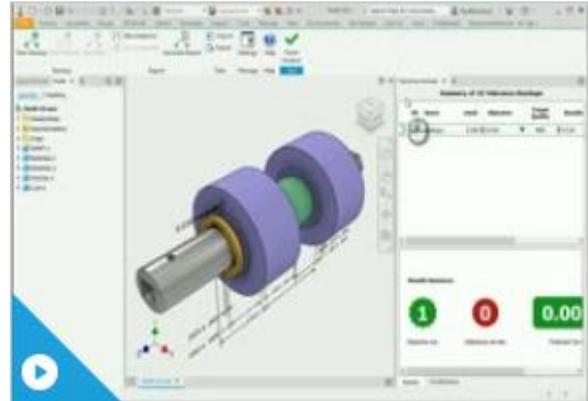
*Save your work if you want – we will use the pre-prepared dataset for the next exercise.  
Close the file before moving on to the next exercise.*

## Exercise 03 – Create a Report

- Rename your stackup
- Rename Components, Faces and Dimensions.
- Take a Snap Shot.
- Generate a report.

Watch the Screencast:

[Finding the Right fit with Autodesk Inventor Tolerance Analysis Exercise 03](#)



### Open the data set for this exercise

Close the dataset you had open for the previous exercise.

Navigate to:

'Get Started' Tab > 'Launch' Panel > 'Open' Button

Navigate to:

C:\Datasets\MFG318994-L Finding the right fit with Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 03

LMB select '**Shaft-03.iam**' (Inventor assembly file) and click on the 'Open' button.

Now we have worked out the optimal tolerances for our parts, we want to share this information with our colleagues. Let's create a report to share.

### Rename your stackup

Before we create our report, we should take the time to rename the important information so that it makes sense (It might be YOU who must read this report in three months - will you remember what each dimension is referring to!).

Navigate to:

Tolerance Analysis Panel > Chevron (<) next to 'Stackup1 Details'.

Clicking on the Chevron will minimise the Stack up.

**Tip:** Notice that you can have multiple stackups defined in one assembly file. With the stackup closed, you can see a results summary of all the stackups defined.

This can be helpful when you need to create multiple stack ups that must all be satisfied.

LMB on the Stack up name. Rename the stackup 'Shaft Stackup'

Name	Sens	Nominal	Tolerance	Cp
------	------	---------	-----------	----

OK	Name	Nominal	Objective	Target Quality	Results
✓	Shaft Stackup	03	±0.01	RSS	±0.01

## Rename Components, Faces and Dimensions.

Click on the Chevron next to your stackup to open the stackup again.

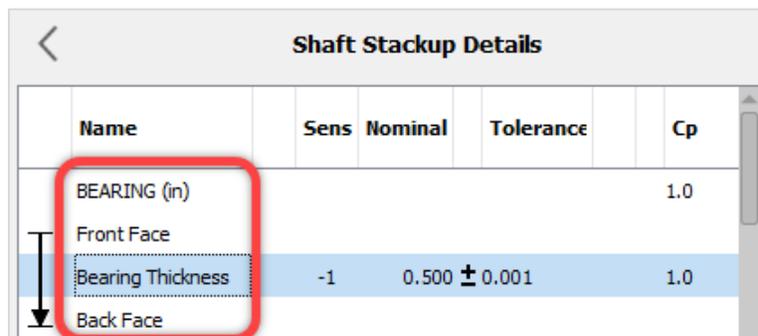
Find the entry for **Bearing:2**

Rename **Bearing:2** to '**Bearing**'.

Rename **Face9** to '**Front Face**'

Rename **Face11** to '**Back Face**'

Rename **Dimension5** to '**Bearing Thickness**'



Name	Sens	Nominal	Tolerance	Cp
BEARING (in)				1.0
Front Face				
Bearing Thickness	-1	0.500	±0.001	1.0
Back Face				

Take a few minutes to re-name the rest of the components, faces and dimensions

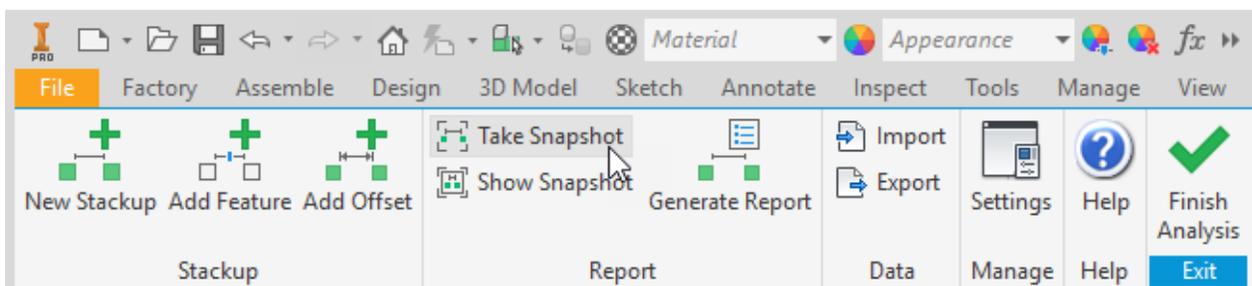
## Take a Snap Shot.

To illustrate the stackup we have analysed in our report we will include a screenshot of our design.

Navigate to:

'Tolerance Analysis' Tab > 'Report' Panel > 'Take Snapshot' Button.

Line up your assembly so that it looks good on screen and click the '**Take Snapshot**' button.



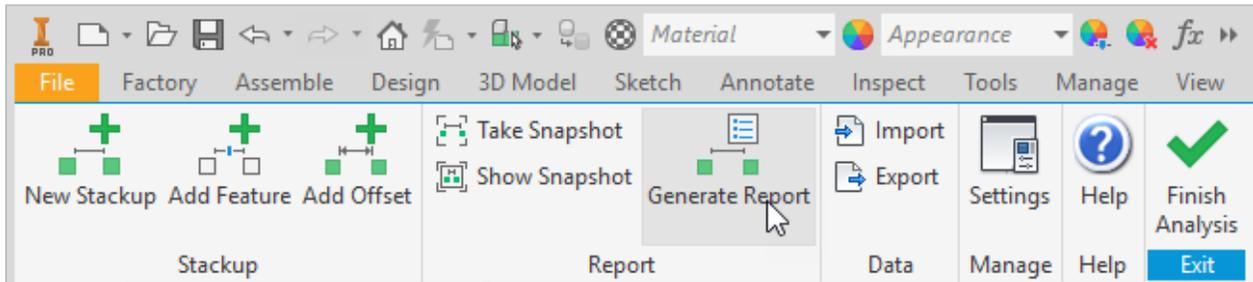
**Note:** You won't see an indication that a snapshot has been taken. To check, alter your view of the model and then click on the 'Show Snapshot' button, to return the view back to the orientation shown in the snapshot.

## Generate a report.

Now we are prepared to generate our report.

Navigate to:

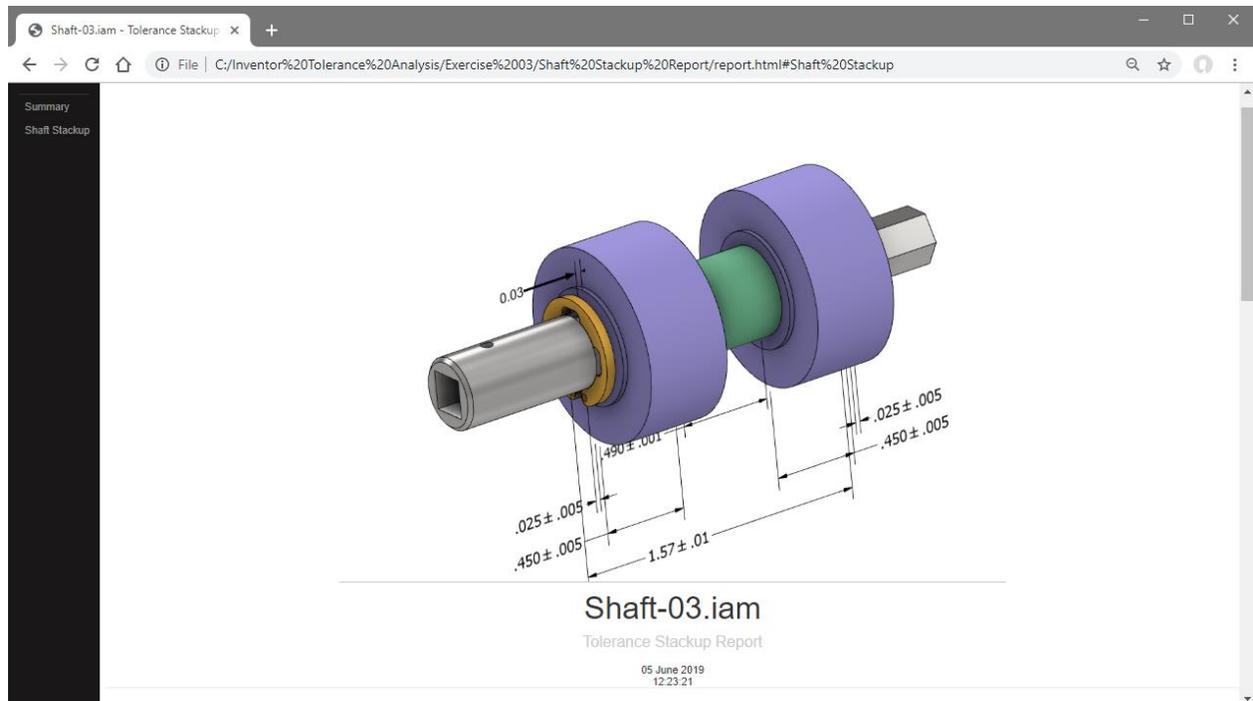
'Tolerance Analysis' Tab > 'Report' Panel > 'Generate Report' Button.



The 'Save Report' dialog will open.

Name your report 'Shaft Stackup Report' and save it in the Exercise 03 Folder.

Click the 'Save' Button, and the report should immediately open in your default Web browser:



### Exercise 03 Extra Credit

- How could you 'Link' the Tolerance Analysis Stack up report to the Assembly file?  
(This would automatically bring the stack up report into Vault).



*Save your work if you want – we will use the pre-prepared dataset for the next exercise.*

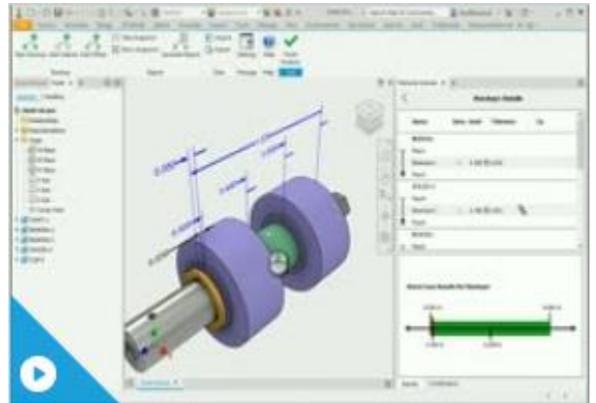
***Close the file before moving on to the next exercise.***

## Exercise 04 – Create the Loop manually

- Look for the Gap
- Understand the Loop
- Select the components
- Select the faces
- Create a stack up

Watch the Screencast:

[Finding the right fit with Autodesk Inventor Tolerance Analysis Exercise 04](#)



### Open the data set for this exercise

Close the dataset you had open for the previous exercise.

Navigate to:

'Get Started' Tab > 'Launch' Panel > 'Open' Button

Navigate to:

C:\Datasets\MFG318994-L Finding the right fit with Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 04

LMB select '**Shaft-04.iam**' (Inventor assembly file) and click on the 'Open' button.

### How to Create the Loop Manually

Autodesk Inventor Tolerance Analysis can automatically infer a loop from the Assembly relationships (Joints and Constraints) applied to your model.

But what if Tolerance analysis can't find the Loop you need? What if you are working with an imported model that doesn't have any constraints applied?

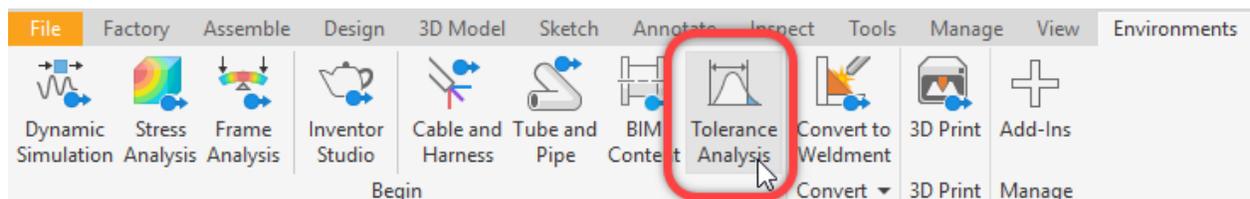
In this exercise we will learn how to create the loop **manually**.

### Look for the Gap

We will analyse the same gap as Exercise 01 – between the front face of Bearing:2 and the back face of Clip:5.

To enter the Tolerance Analysis environment, navigate to:

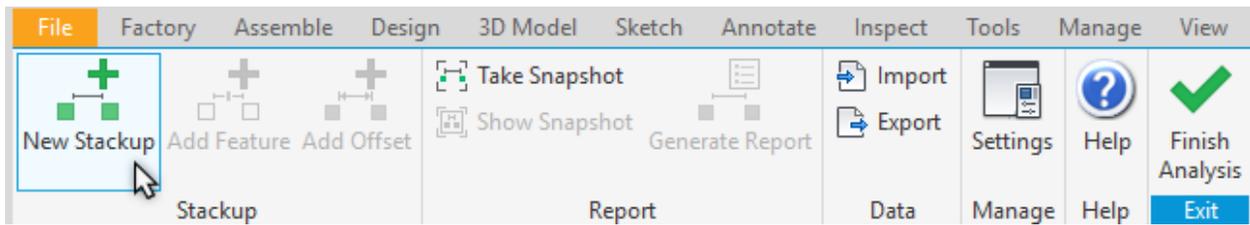
'Environments' Tab > 'Begin' Panel > 'Tolerance Analysis' Button.



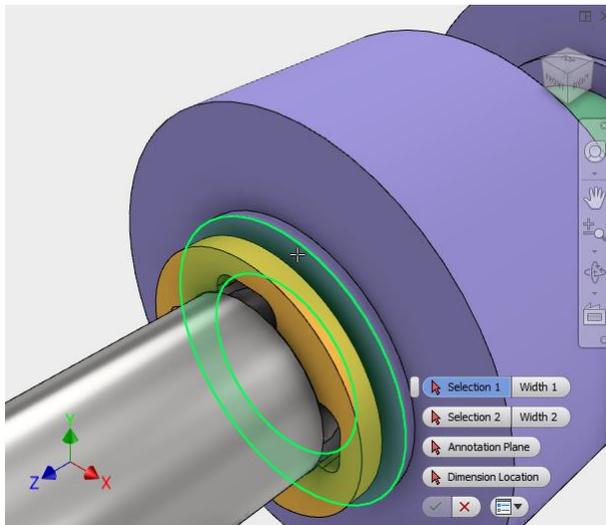
To create a stack up, navigate to:

'Tolerance Analysis' Tab > 'Stackup' Panel > 'New Stackup' Button.

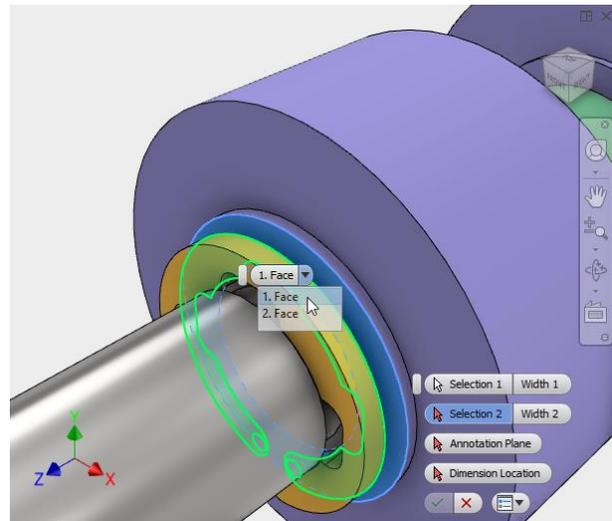
Click on the 'New Stackup' button to start the stack up.



In the Graphics window, select the front face of Bearing:2, then the back face of Clip:5.



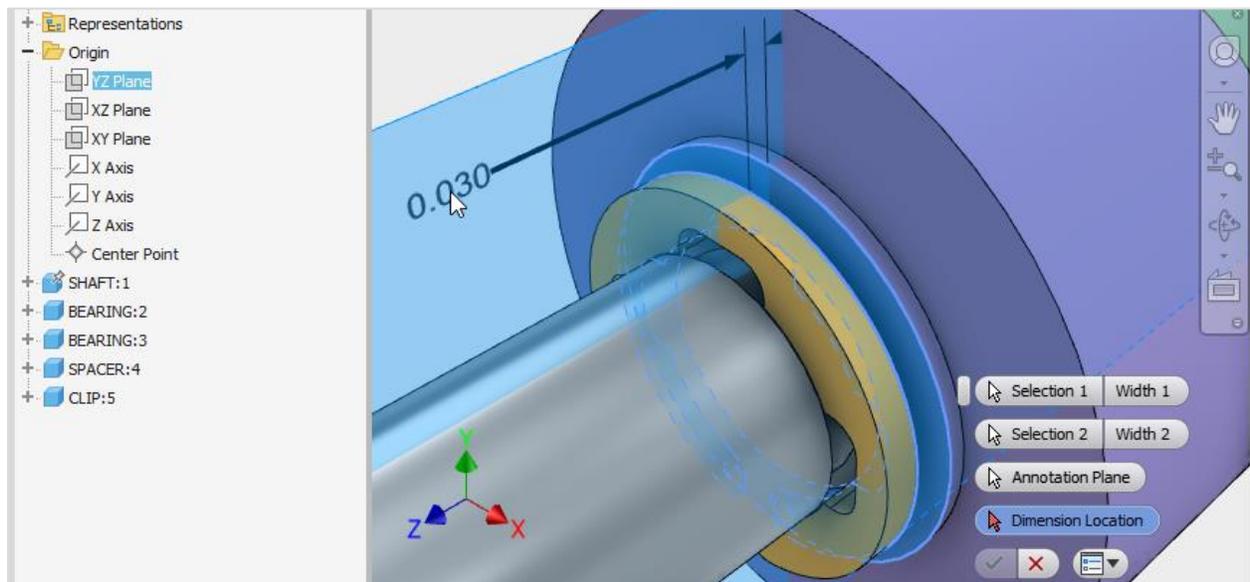
First Selection



Second Selection

From the model browser, Select the YZ Plane for your Gap Annotation.

LMB click in the Graphic window to place your annotation.



This first part is the same as in Exercise 01 – from here on the process is different!

## Understand the Loop

In Exercise 01, Tolerance Analysis worked out the loop for us from the constraints applied to the Assembly. This time we are going to do the process manually.

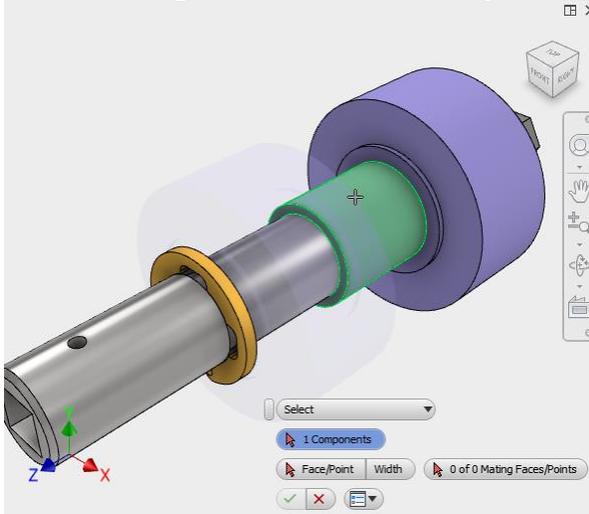
The first step is to tell Tolerance Analysis which components will be included in the Loop.

### Select the components

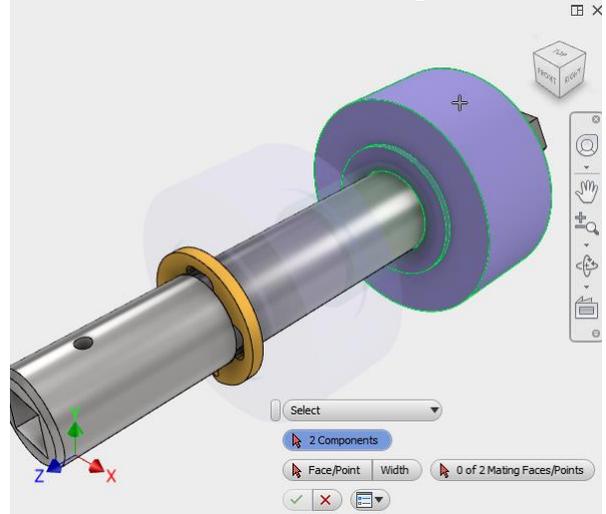
Tolerance Analysis knows that Bearing:2 will be included in the Stackup (because we picked a face on it) so it is selected and becomes 'Transparent' and disabled (You can't select it again) in the Graphics window.

Tolerance Analysis would now like you to pick the rest of the components for the Loop - *in the order that you would like them to be included in the Loop*. Picking Clip:5 will complete the loop.

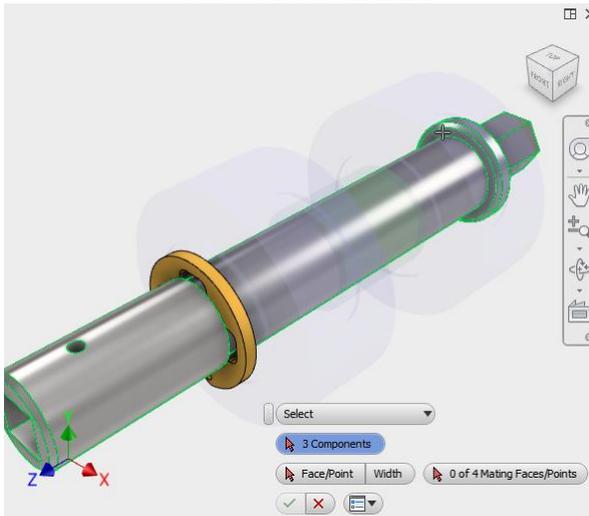
**01 > Bearing:2 Selected - Select Spacer:4**



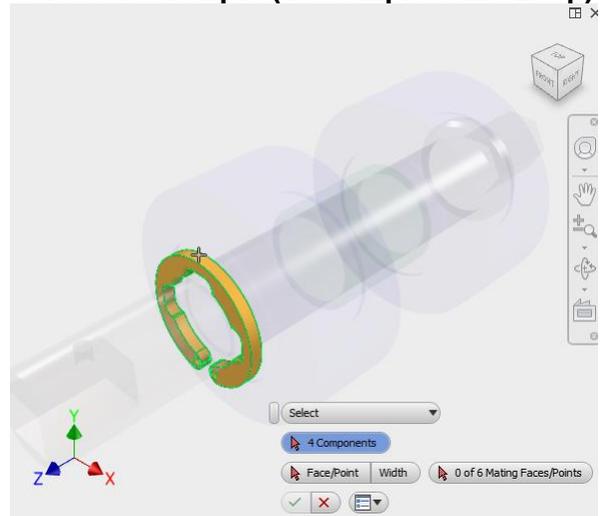
**02 > Select Bearing:3**



**03 > Select Shaft:1**



**04 > Select Clip:5 (To complete the Loop)**



## Select the faces

When we have selected all the components that will complete the Loop, Tolerance Analysis will now ask us to identify which faces on each component to analyze.

All components will disappear, apart from Bearing:2 – which is opaque, and Spacer:4 – which is transparent.

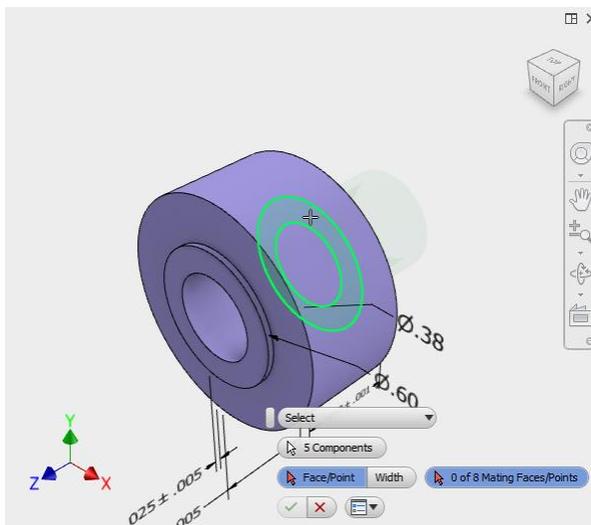
Tolerance Analysis would like you to select which face on Bearing:2 to match with which face on Spacer:4.

Pick the BACK face on Bearing:2, where it meets Spacer:4.

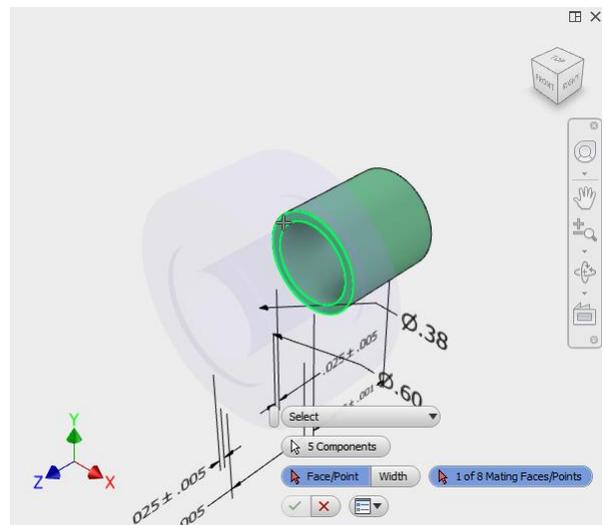
Immediately, Bearing:2 becomes translucent, and Spacer:4 becomes opaque. We can now pick a face on Spacer:4

Continuing picking the mating faces to complete the Loop.

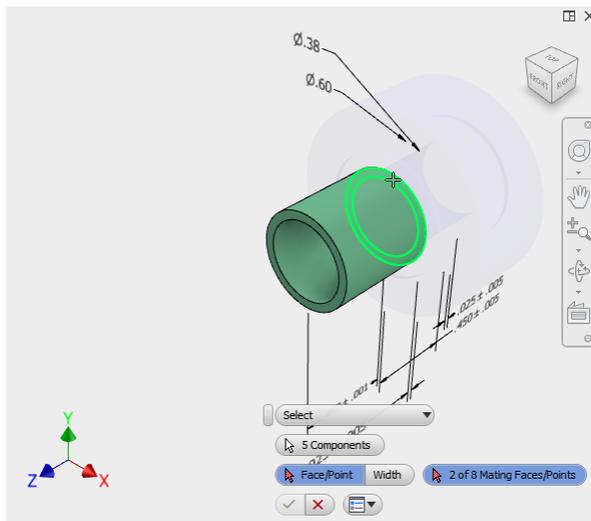
### 01 > Pick the BACK face on Bearing:2



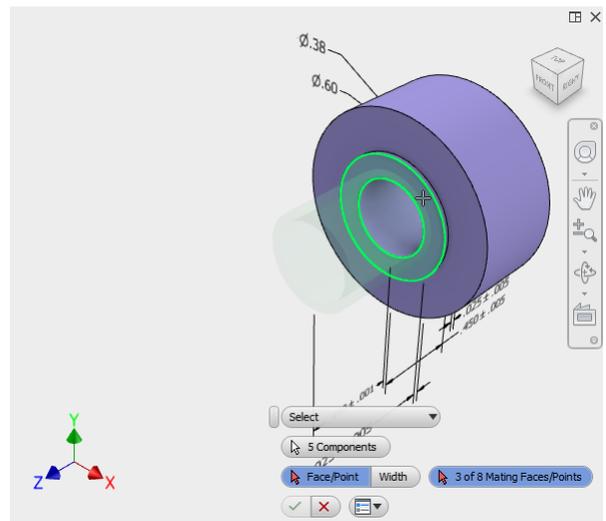
### 02 > Pick the front face of Spacer:4



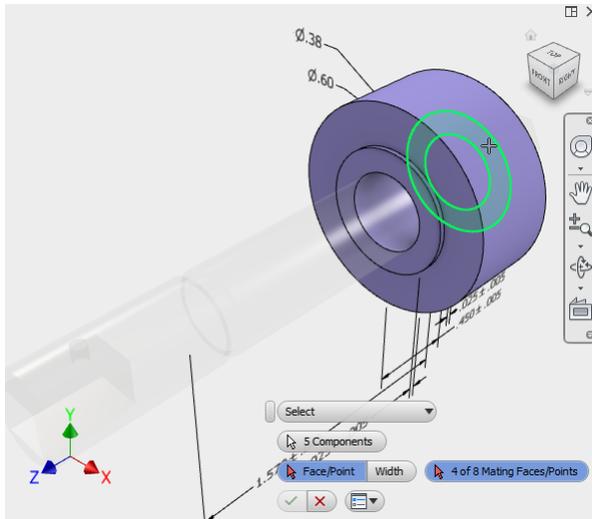
### 03 > Back face of Spacer:4



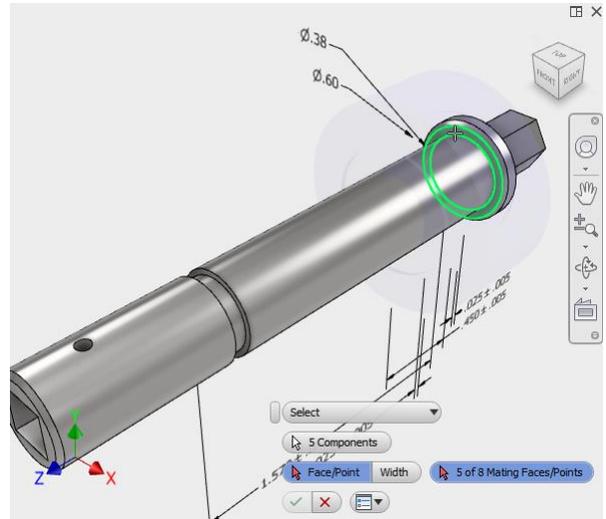
### 04 > Front face of Bearing:3



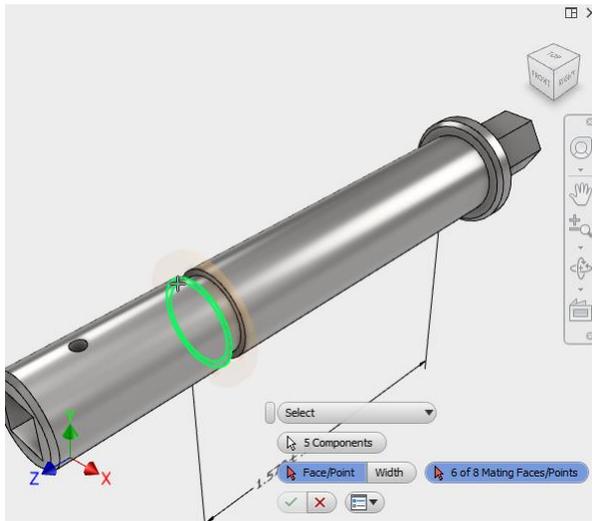
**05 > Back face of Bearing:3**



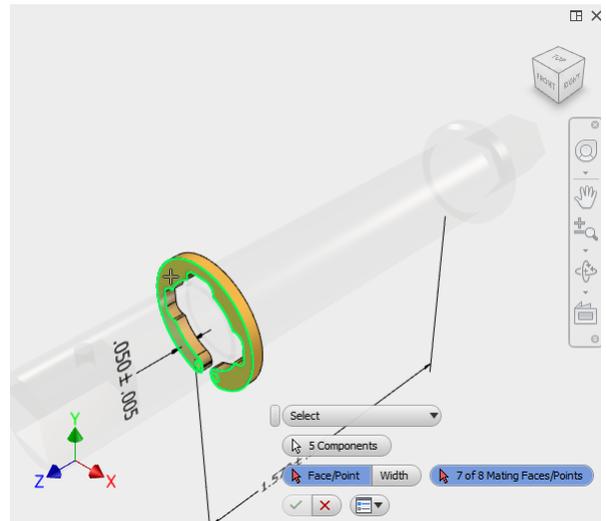
**06 > Rear shoulder on Shaft:1**



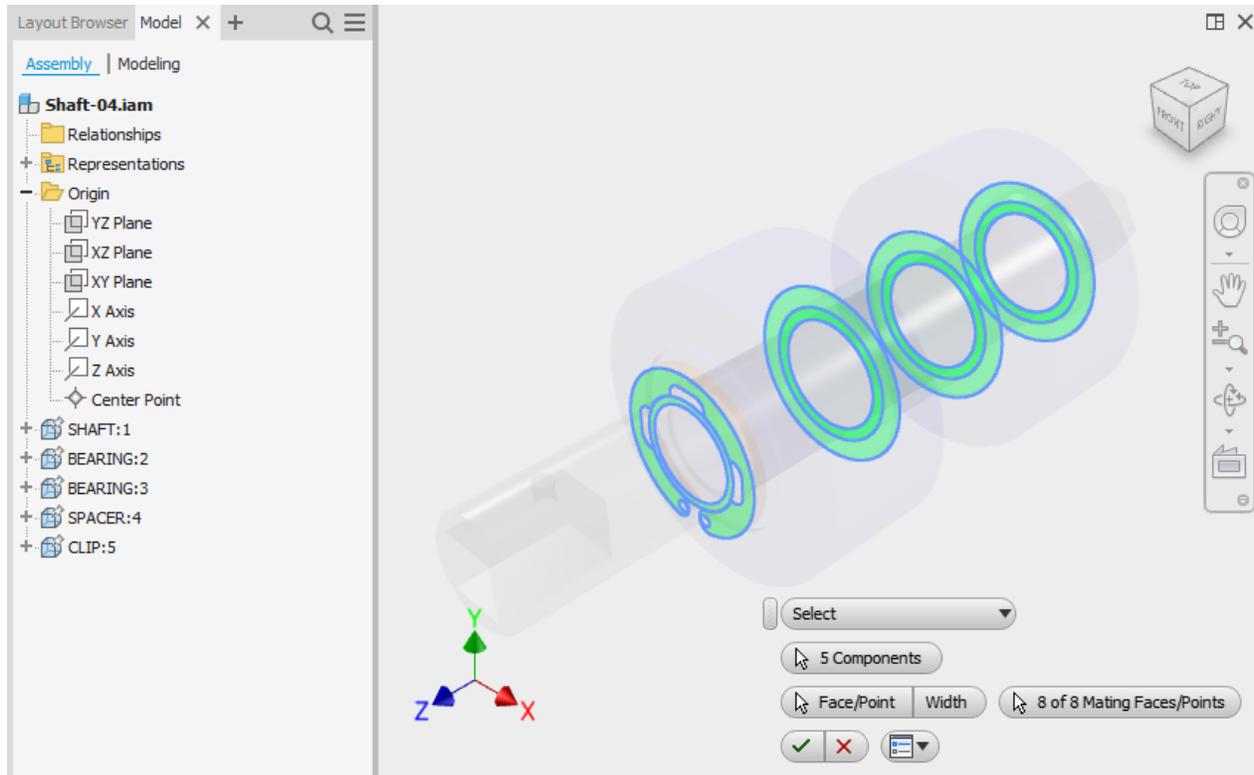
**07 > Front face of Groove on Shaft:1**



**08 > Front face of Glip:5  
...to complete the Loop**

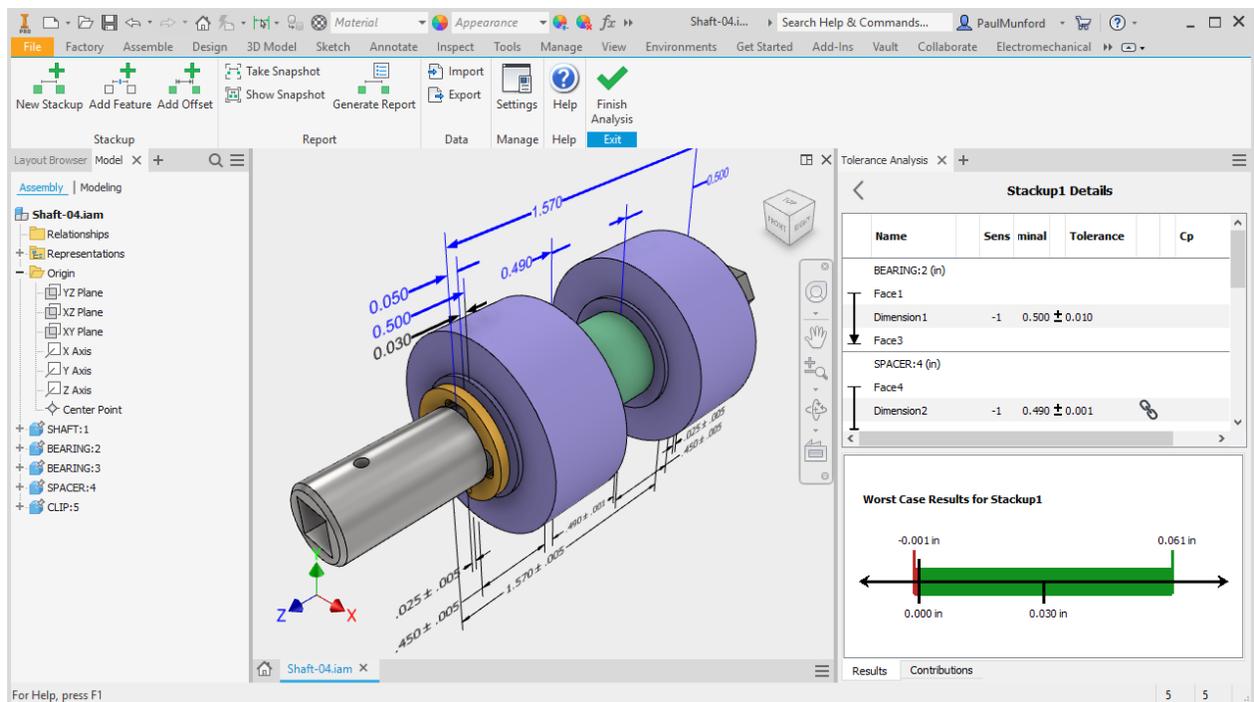


All the relationships will highlight. Click on the green 'Tick' [Check] ✓ button to complete the command.



## Create a stack up

Your stack up is created.



## Exercise 04 Extra Credit

Tolerance analysis V.1 will only infer a loop if each component is mated face to face, similarly to how we manually selected the Loop in Exercise 04.

*Insert constraints are not analyzed in older versions. This has been implemented in V2020.1*

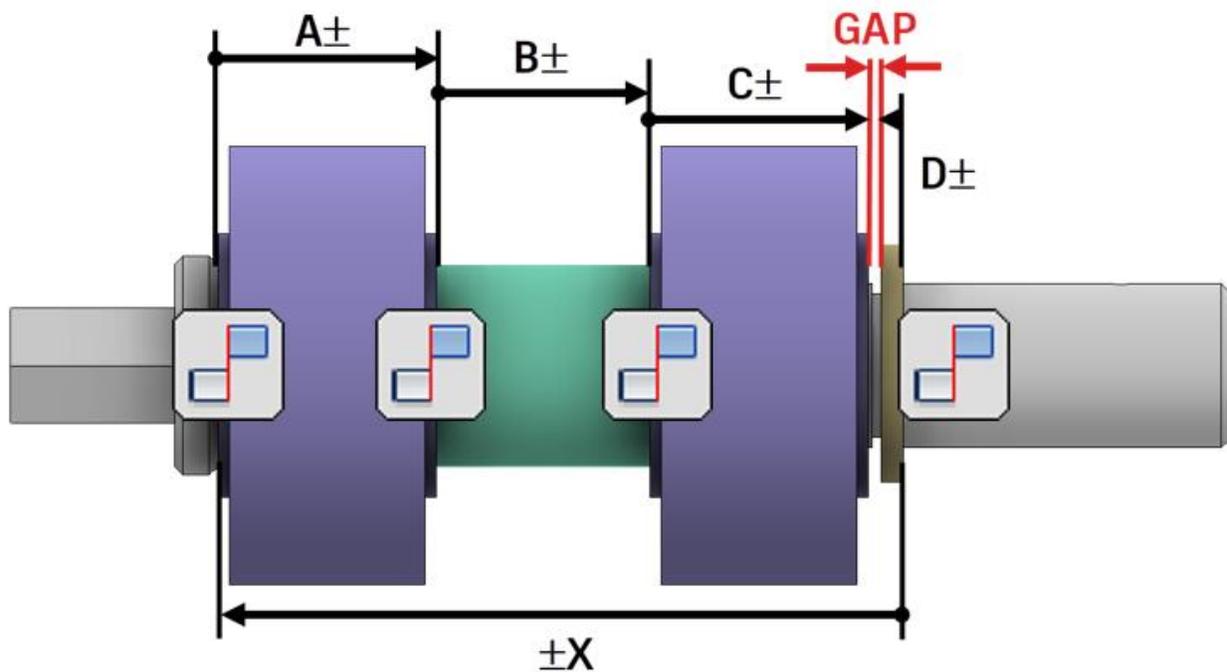
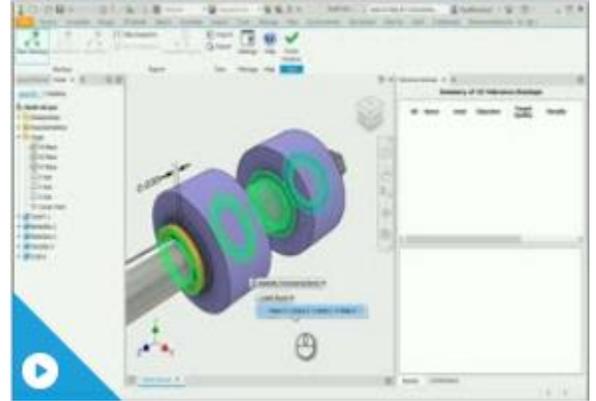
Watch out for the point releases as the year goes on for new functionality - or join the Beta to learn what's coming next for Inventor Tolerance Analysis

<https://Beta.Autodesk.com>

*Can you constrain the components in such a way that Tolerance analysis can Automatically detect a loop?*

Watch the Screencast:

[Finding the right fit with Autodesk Inventor Tolerance Analysis Exercise 04 \(Extra Credit\)](#)

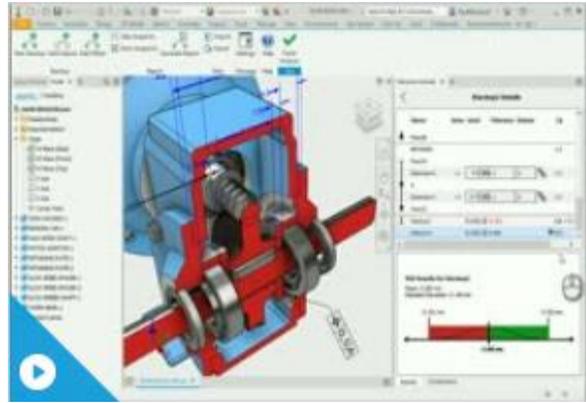


## Exercise 05 – Your choice!

- Choose a challenge
- Open the dataset
- Enter the Tolerance Analysis Environment
- Apply what you've learned

### Challenge 01 – Gear Reducer

- Practice creating a stackup
- Practice automatic Loop detection
- Explore a dataset that uses GD&T 3D annotation with Tolerance Analysis.



Watch the Screencast:

[Finding the right fit with Autodesk Inventor Tolerance Analysis Exercise 05 – Automatic Loop Stackup Creation Practise](#)

### Open the data set for this exercise

Navigate to:

'Get Started' Tab > 'Launch' Panel > 'Open' Button

Navigate to:

C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 05\Gear Reducer\

LMB select '**GEAR REDUCER.iam**' (Inventor assembly file) and click on the 'Open' button.

### Change the view

Navigate to the Model Browser. Next to the 'Representations' folder, click on the + symbol to expand the folder.

Next to 'View', click on the + symbol to Expand the saved Views node.

Double LMB click on '**Slow Speed Shaft**' to activate this saved view.

### Enter the Tolerance Analysis Environment

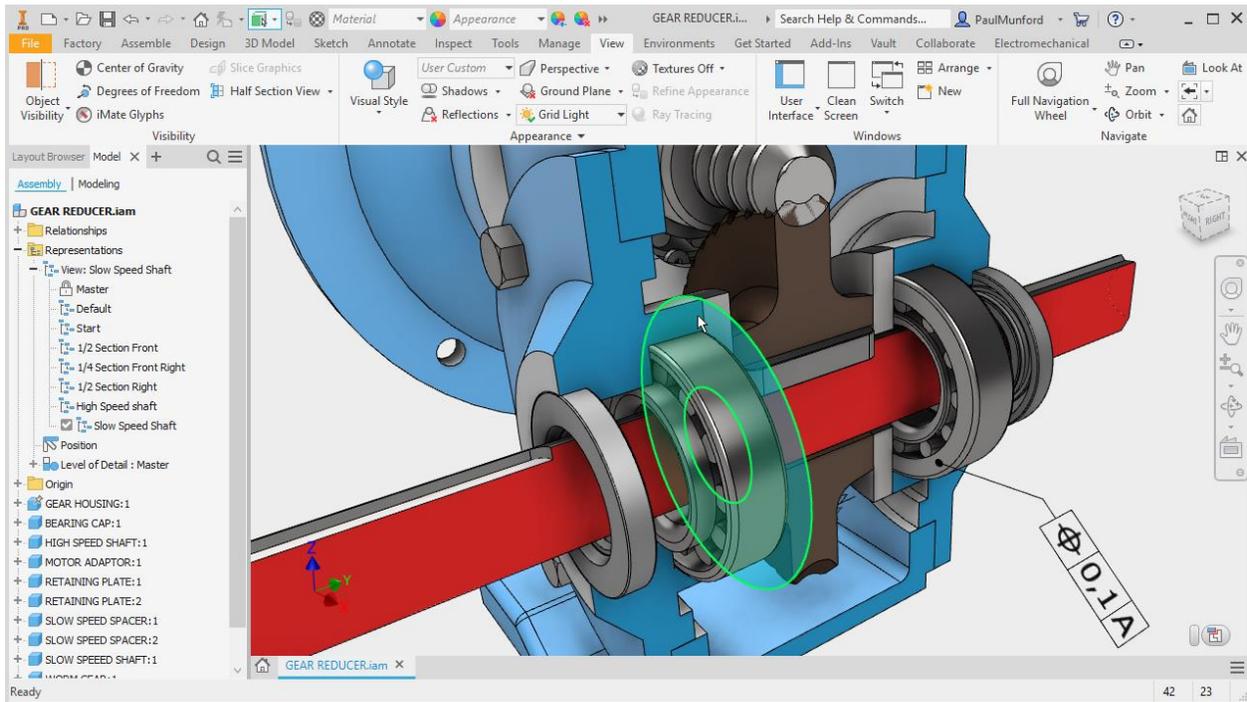
Start Tolerance Analysis – notice that there is one stackup saved in this file.

**Optional:** If you'd like to review this stackup, set the saved view to '1/2 Section Front' and set the Projection to 'Perspective with Orthographic Faces'.

Carry on with the next step.

## Find the Gap

In this exercise we will analyse the gap between the Slow Speed Spacer and the Retaining Plate.

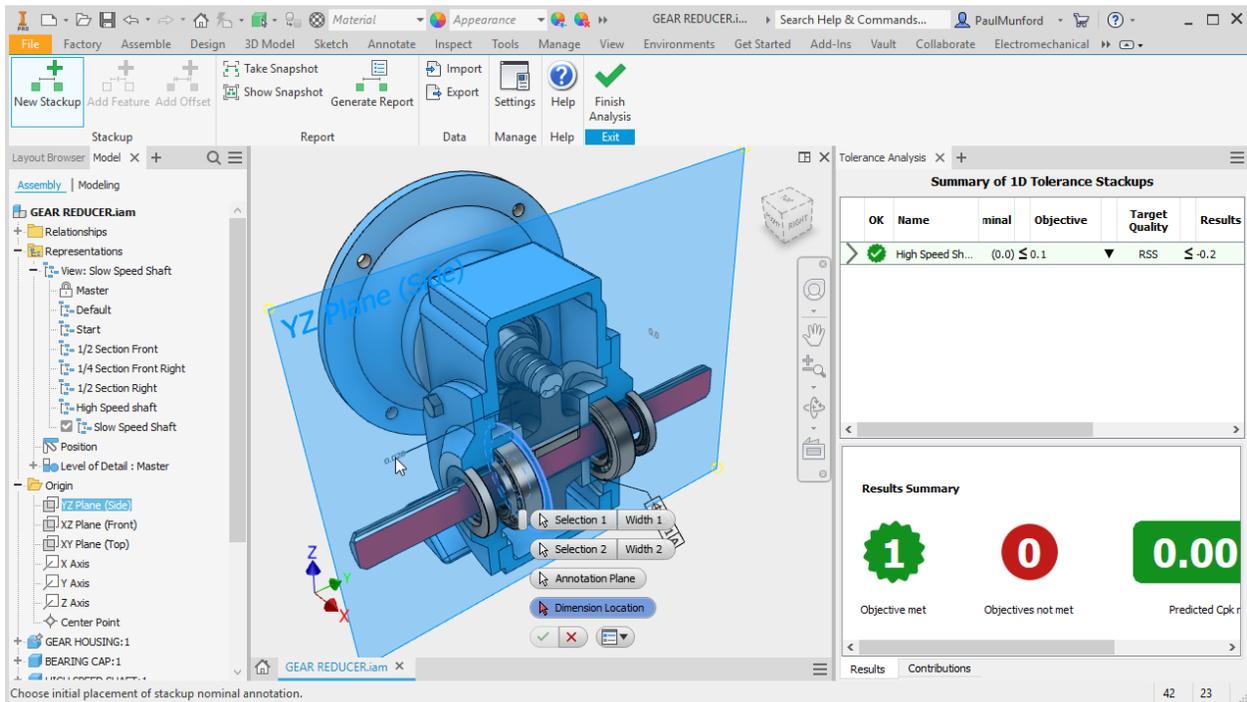


## Create a Stackup

Create a stackup analysis between the Slow Speed Spacer:2 and Retaining Plate:1

Use the YZ plane for your Gap Annotation.

The Loop path will be detected automatically.



## Establish the Objective

Let's imagine that the Slow Speed Spacer is made of a material that can compress slightly.

We would like the Gap to be 0.0 mm. We can accept a Gap that is slightly less (The material will compress), but no more (We don't want a physical gap).

Set the objective to 'Limits' with an upper value of 0.00 and a lower value of 0.25

## Compare results

Compare the 'Worst Case' results Graph with the RSS Graph.

## Compare contributors

Which component tolerances are contributing most to the stackup?

## Adjust your tolerances.

Adjust the tolerances on the components until the results are Green on the RSS Graph.

## Compare results

Compare the 'Worst Case' results Graph with the RSS Graph. Can you optimise your tolerances any further?

The screenshot displays the SolidWorks Tolerance Analysis interface. The main window shows a 3D model of a gear reducer assembly with a gap dimensioned as 0.00 mm. The 'Stackup2 Details' table is visible, showing the following data:

Name	Sens	ninal	Toleranc	Datum	Cp
Dimension3	+1	15.88	0.01	A	
Face22					
Stackup2	(0.00)		0.03		
Objective...	(0.00)		0.00		▼ Worst Ca

The 'Worst Case Results for Stackup2' graph shows a range from -0.25 mm to 0.03 mm, with a target of 0.00 mm. The graph is currently in red, indicating a failure to meet the objective.

## Create a report

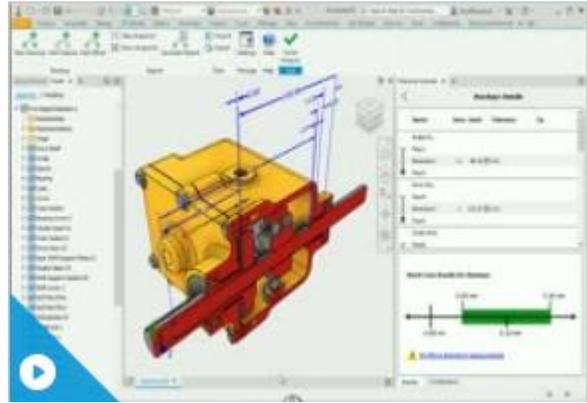
- Name your Stackup
- Create a snapshot
- Create a report

## Challenge 02 – 2 Speed Gear Box

- Practice opening a non-native CAD file in Inventor using AnyCAD.
- Practice Creating a Manual stack up

Watch the Screencast:

[Finding the right fit with Autodesk Inventor](#)  
[Tolerance Analysis Exercise 05 – Manual Loop](#)  
[Stackup Creation Practise](#)



### Open the data set for this exercise

Navigate to:

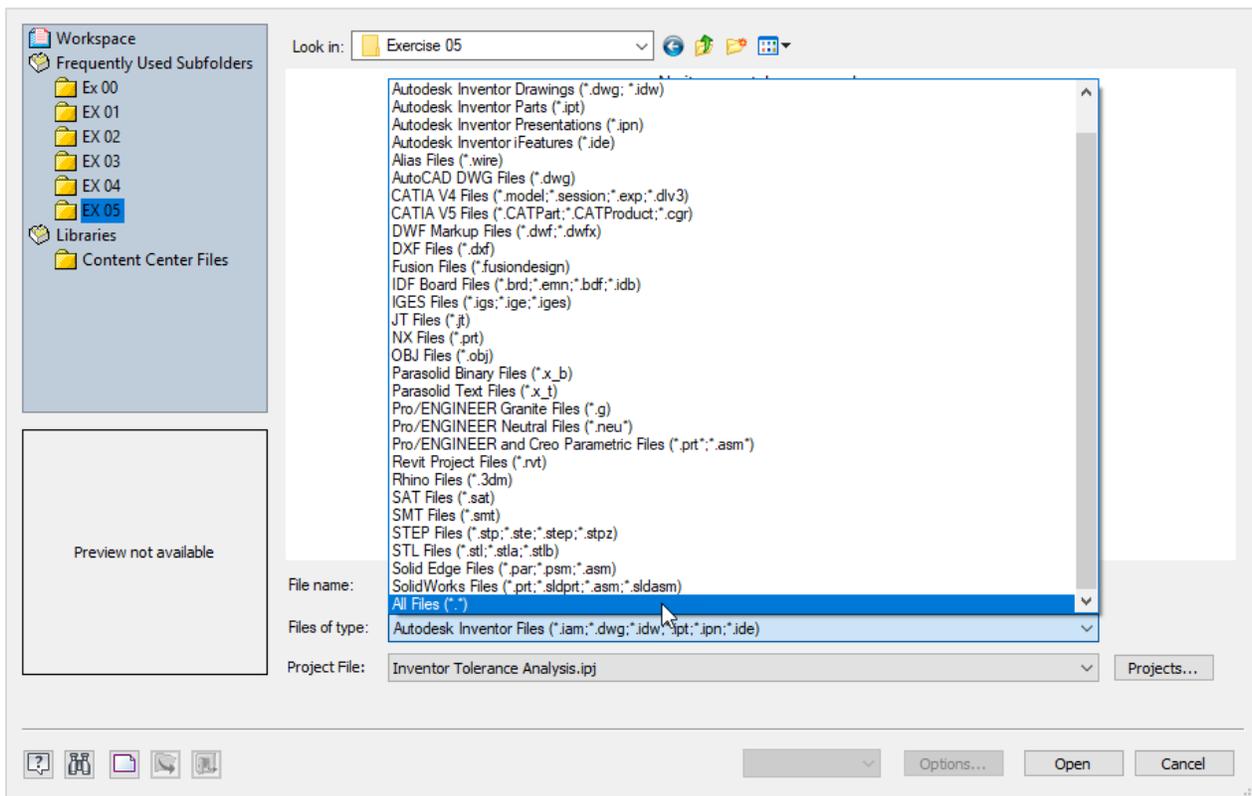
'Get Started' Tab > 'Launch' Panel > 'Open' Button

Switch 'Files of type:' to 'All Files'

Navigate to:

C:\DATASETS\MFG318994-L Autodesk Inventor Tolerance Analysis\Tolerance Analysis\Exercise 05

LMB select '**Two-Speed Gearbox.stp** (STEP file) and click on the 'Open' button.



## Reference a file into Inventor using AnyCAD

'AnyCAD' is a technology which allows Inventor to reference CAD data from other CAD applications into Inventor.

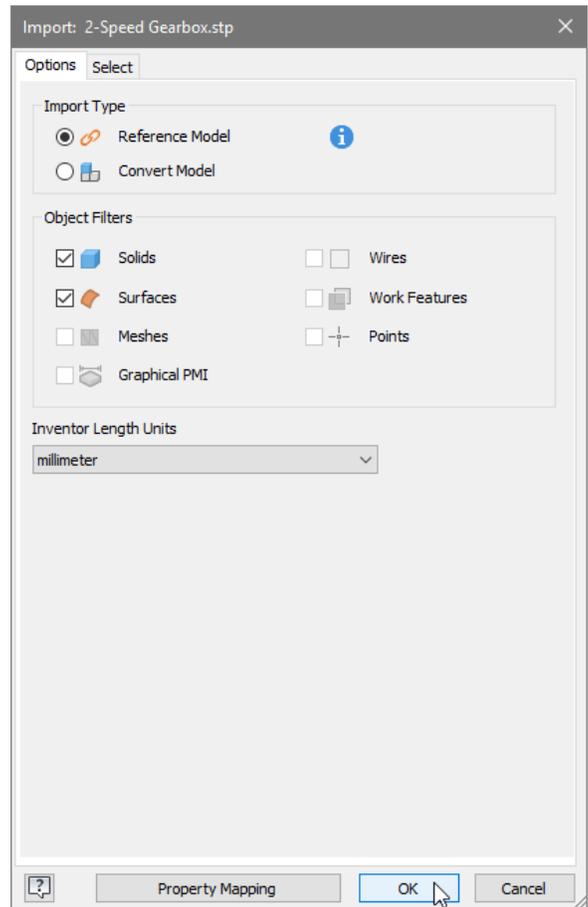
The original files can be edited using the authoring programme that created it. When the file is saved, Inventor will prompt you to update the file – and you will see the updates inside Inventor.

This can be used on Solidworks, Pro-E and Catia files, along with many others.

When the 'Import' Dialogue opens:

- Pick 'Reference Model'
- Select 'Inventor Length Units' > Millimetre

Click the 'OK' Button to open the file.



## Create a Section View

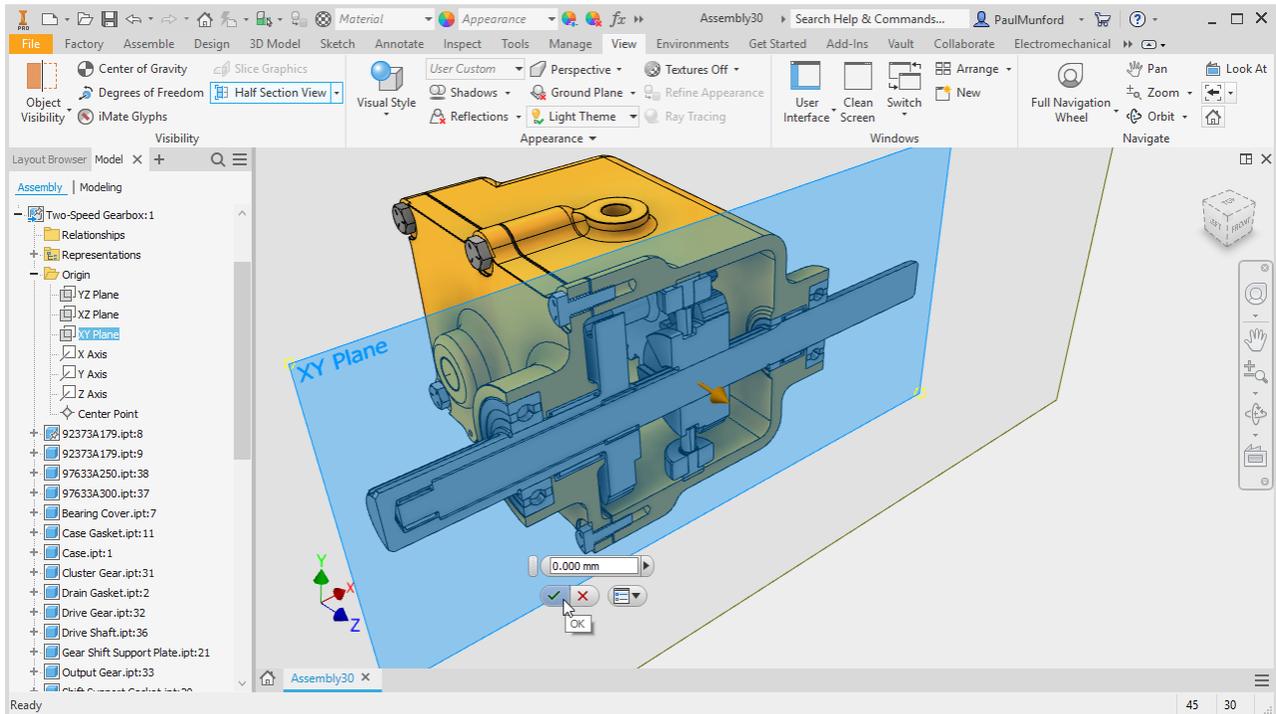
To make it easier to find the components that we wish to analyse, create a section view on the XY Plane.

Navigate to:

'View' Tab > 'Visibility' Panel > 'Half Section View' button

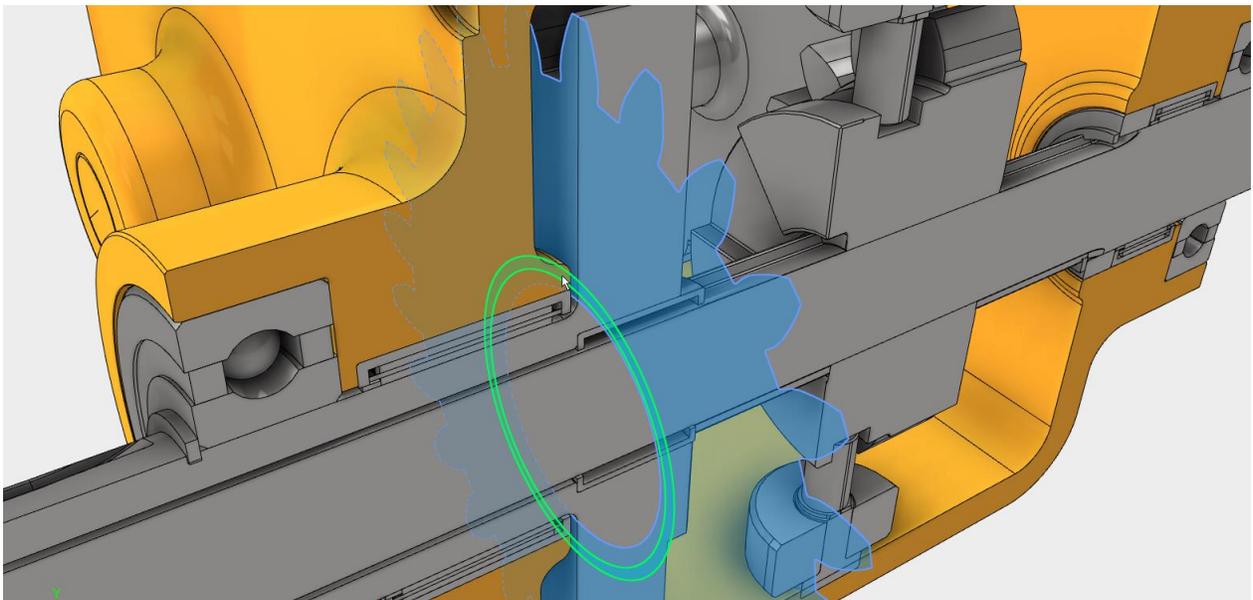
In the model browser, Next to the 'Origin' folder, LMB on the 'Plus' symbol to expand the folder.

Select the XY Plane. Click on the Green 'Tick' ['Check'] ✓ button to complete the command.



## Look for the Gap

We would like to analyse the gap between the Gear and the Housing.



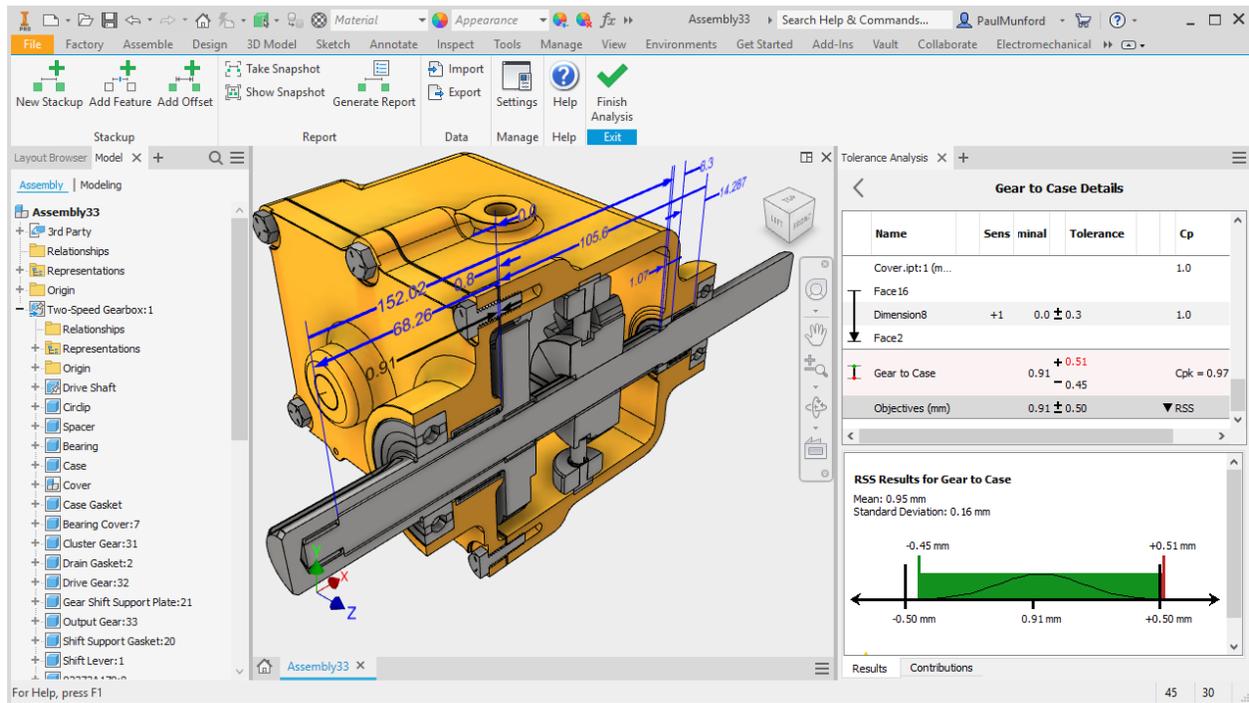
## Understand the Loop

Study the Assembly. Which components will comprise the Loop?

## Create a Stackup

Enter the Tolerance Analysis Environment.

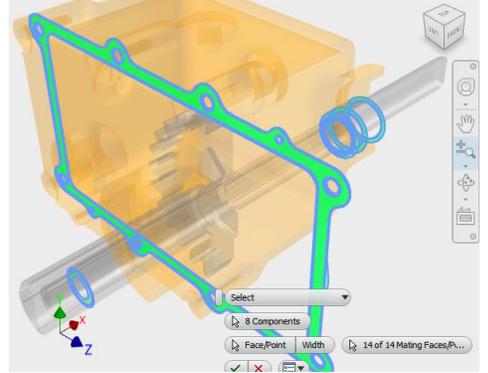
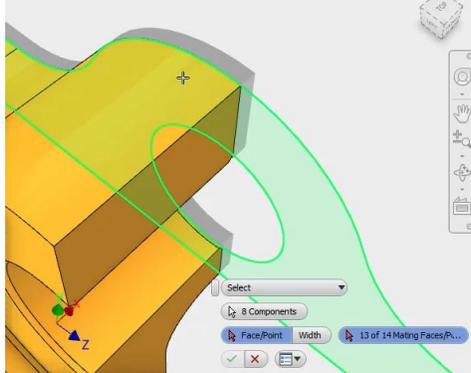
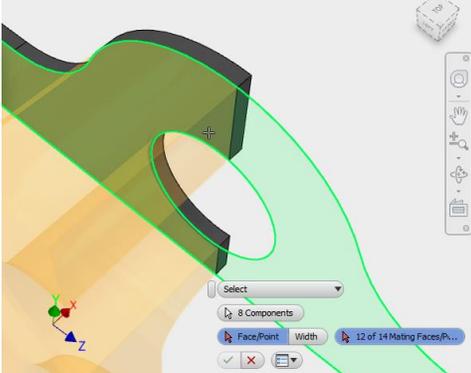
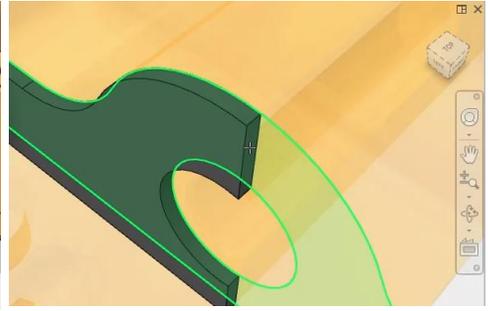
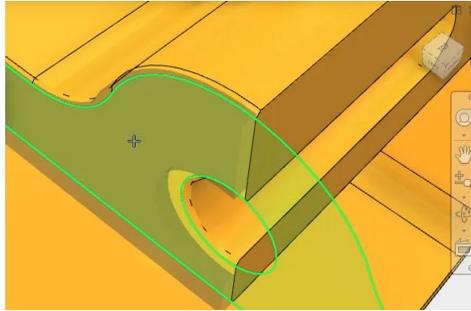
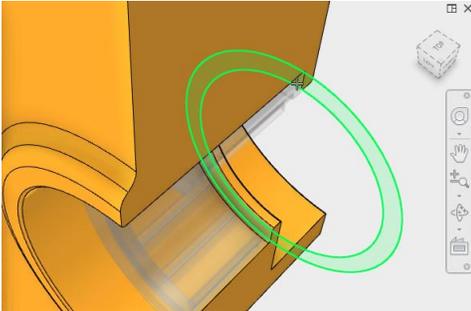
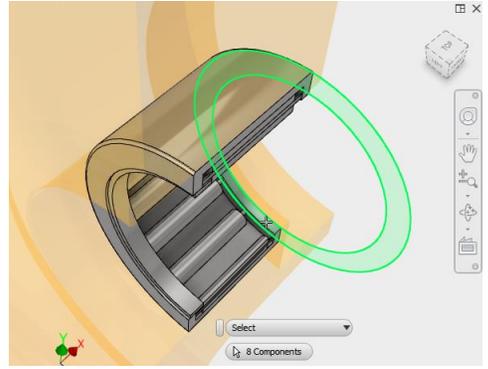
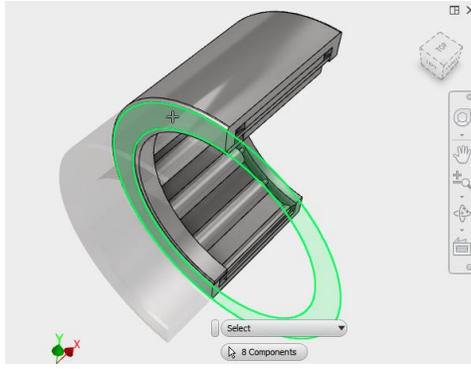
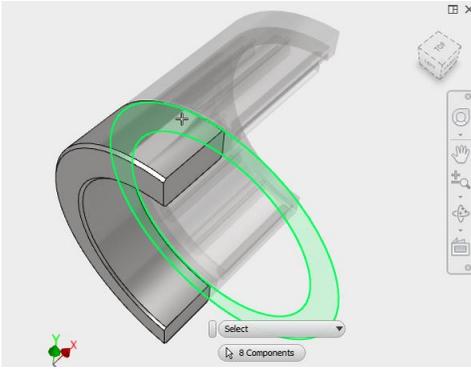
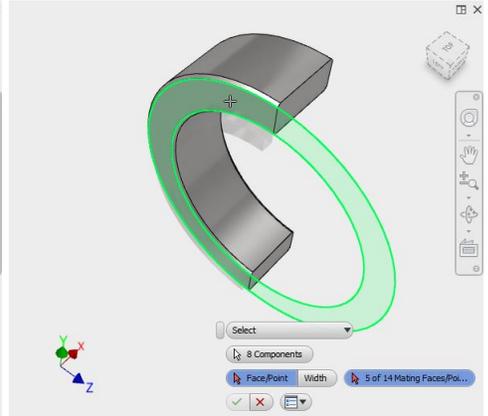
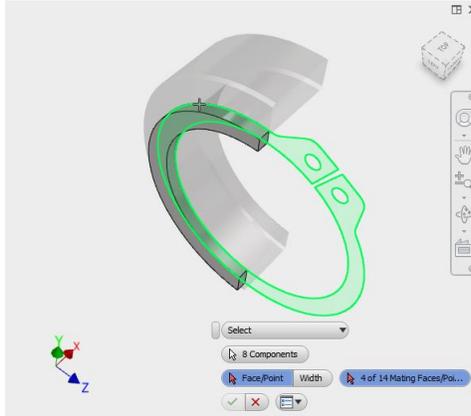
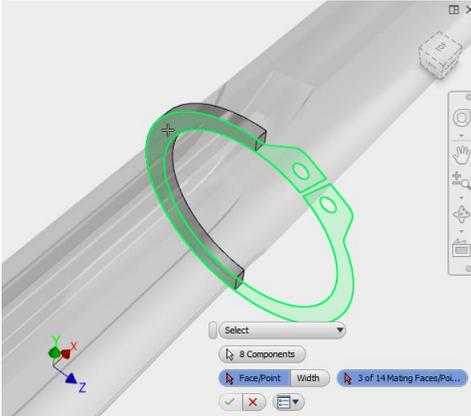
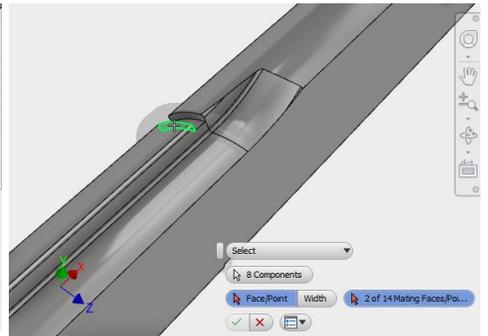
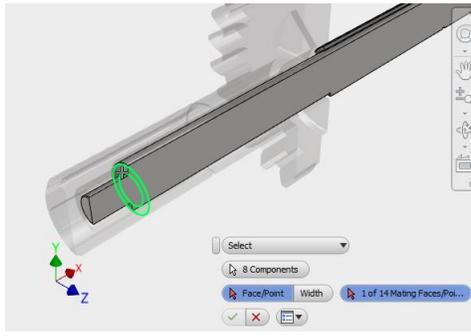
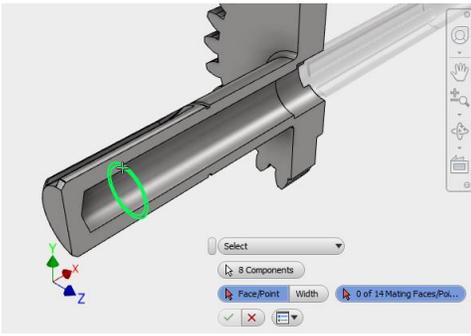
Create a new stack up to analyse the gap between the Gear and the Housing.



Pick the components for the Loop in the following order.

1. Drive Shaft
2. Circlip
3. Spacer
4. Bearing
5. Case
6. Case Gasket
7. Cover

Pick the mating faces on the components in the following order (Shown on next page).



## Review the Tolerances

Tolerance Analysis will add Nominal tolerances to the component dimensions.

This files dimensions are in Metric. Adjust the tolerances for each component:

1. Output Gear       $\pm 0.5$
2. Drive Shaft       $\pm 0.5$
3. Circlip            Max +0.0 Min -0.05
4. Spacer              $\pm 0.2$
5. Bearing      Max +0.0 Min -0.015
6. Case                $\pm 0.5$
7. Case Gasket       $\pm 0.1$
8. Cover               $\pm 0.5$

## Establish the Objective

Solve for a symmetrical objective  $\pm 0.5$

## Compare results

Compare the 'Worst Case' results Graph with the RSS Graph.

## Compare contributors

Which component tolerances are contributing most to the stackup?

## Adjust your tolerances.

Adjust the tolerances on the components until the results are Green on the RSS Graph.

**Note:** *The Bearing and Circlip are purchased components. The tolerances we have inputted are from the supplier and can't be changed.*

## Compare results

Compare the 'Worst Case' results Graph with the RSS Graph. Can you optimise your tolerances any further?

## Create a report

- Name your Stackup
- Create a snapshot
- Create a report

## What we didn't cover!

### Tolerance Analysis settings and defaults

For more information on Tolerance analysis settings, please check out the help:

<http://help.autodesk.com/view/INVTOL/2020/ENU/?guid=GUID-93BA3B66-FC45-4B99-8C05-FB4D165314F2>

### Cp and Cpk definition

Thanks very much to Stephen Werst of [Sigmetrix](#) for providing this explanation.

The relationship among these values is defined in the following equations:

**C<sub>p</sub>:**

$$C_p = \frac{(UTL - LTL)}{6\sigma}$$

**C<sub>pk</sub>:**

$$C_{pk} = \frac{(UTL - LTL) - 2|\mu - T|}{6\sigma}$$

Where,

(UTL-LTL) = Tolerance bandwidth

$\sigma$  = Standard deviation

$\mu$  = Mean

T = Target (or nominal) value

*The most common assumption of Cp=1.0 stems from the assumption that manufacturing will select a manufacturing process that will place the defined tolerances at +/- 3 standard deviations from center of the tolerance zone, assumed to be the mean, so that the probability of a part complying to the required tolerances is 99.7%. For all statistical analyses Tolerance Analysis assumes that manufacturing will target the midpoint of the tolerance range, thus the mean is assumed to be the midpoint of the tolerance range.*

It's best to think of Cp as the *capability* of a given process to manufacture parts within a desired tolerance range.

Since the historical assumption that a process whose standard deviation is 1/6<sup>th</sup> the range between the limits (i.e. +/- 3 standard deviations from the midpoint) is "good enough" to produce a specific feature on a part at "acceptable" defect levels, this becomes the baseline of a Cp = 1.0.

All other process variability to tolerance ratios are then compared to this baseline to discern "how good" the parts produced by the process will be (i.e. the ratio of variation that will occur to the range of tolerances).

Cp is most interesting BEFORE parts are made, because it only addresses the ability to make a good group of parts. Something can still happen during set-up that moves the mean of the distribution of the parts produced well outside the specification limits, but if the variation between the parts is still measured to be small relative to the range of the specification limits, the measured Cp would still appear to be “good” despite the fact that most of the parts were produced outside of one of the spec limits.

Enter Cpk. It effectively calculates the ratio of the variability of the process to the smallest distance between the mean and the nearest tolerance limit. When the mean of the distribution is at the midpoint of the spec limits,  $Cpk = Cp$ , but as the mean moves towards either limit,  $Cpk < Cp$ .

Note that, since the mean is part of the calculation, it's most applicable for referring to produced parts, else one must assume a mean. There's typically no reason to assume a mean other than at the tolerance midpoint a priori, hence Cp is a better metric to use before parts are produced and Cpk a better one after. It's a better estimator of the defect rates that will occur should the process continue without change.

Here's a simple example in the hopes of helping to explain why both metrics are needed.

Let's assume a feature must be made on a part with a dimension of 25 +/- 1mm. Manufacturing may have 3 different processes capable of making the part, each having different variability and costs.

Let's assume the standard deviations of the three processes, from cheapest to most expensive, are .5, .3, and .1. The first is ruled out as likely to create too many defective parts. The second, with a Cp of 1.1, seems like a good candidate.

During setup a part is made with a size of 25.3, and the next one is at 25.2, so the operator adjusts the process with an offset of -0.25 to try to bring the anticipated mean back down to 25. However, as the nature of any random process, it may be that these initial two parts happened to be on the higher end of the actual process distribution and thus the operator only made matters worse.

Let's say that the initial process mean was 24.8 and now the operator adjusted it down to 24.55. If the rest of the parts were made with no other monitoring or adjustment, the Cpk of the batch would be .6 with an anticipated defect rate of 3.4% instead of the 0.1% expected originally. This is only one simple example of the impact mean shifts have on quality.

Note that both metrics are only useful when the distribution is normal, or Gaussian. As the distribution become less and less normal, neither of these are good predictors of defect levels.

## Thank You

Thank you very much for attending this Lab at Autodesk University. I hope that you now feel confident to use Autodesk Inventor Tolerance Analysis at work and adjust the workflows we've learned to suit your company's needs.

I'd like to thank my Boss Lars Christenson and my colleagues at Autodesk, Chris Hall, Luke Mihelcic and Jim Byrne for helping me to learn Tolerance Analysis and put this class together.

I'd also like to thank our awesome Lab assistants, Steve Dennis, Steven (Mingwei) Gao, Stephen Wurst, Steven (Peter) De Strijker and Carlos Lastrilla.

My thanks to Steve Dennis and Steven Gao for checking this handout for errors – any remaining mistakes are my own.

Finally, I'd like to thank Stephen Werst for answering all my Tolerancing questions.

## Resources

If you would like to learn more about Autodesk Inventor Tolerance Analysis, I can recommend the following resources:

Overview Page and download a Trial:

<https://www.autodesk.com/products/inventor-tolerance-analysis/overview>

Help and Release Notes:

<http://help.autodesk.com/view/INVTOL/2020/ENU/>

Join the Beta to learn what's coming next for Inventor Tolerance Analysis

<https://Beta.Autodesk.com>

Try the Guided Tutorial:

