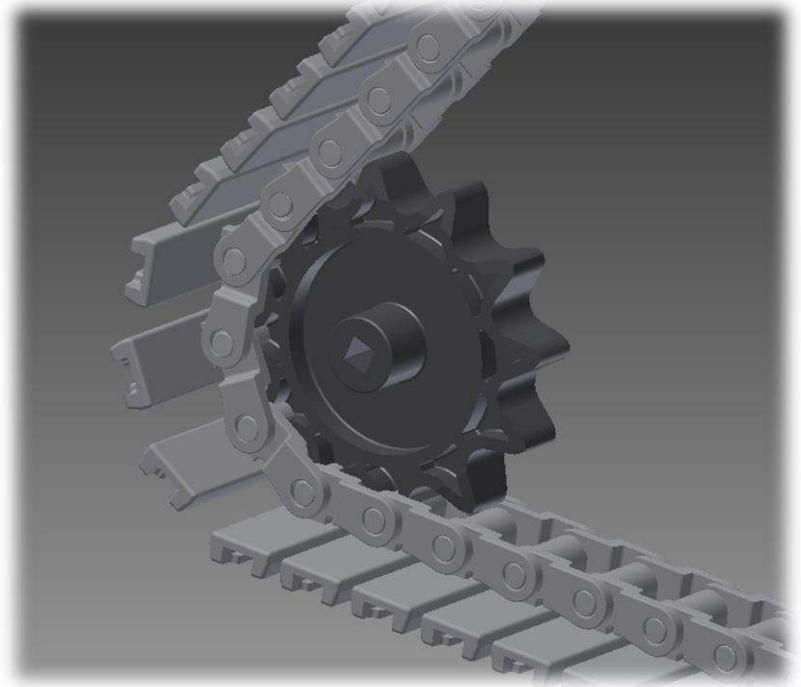


Tank Tread Tutorial



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1. Preface

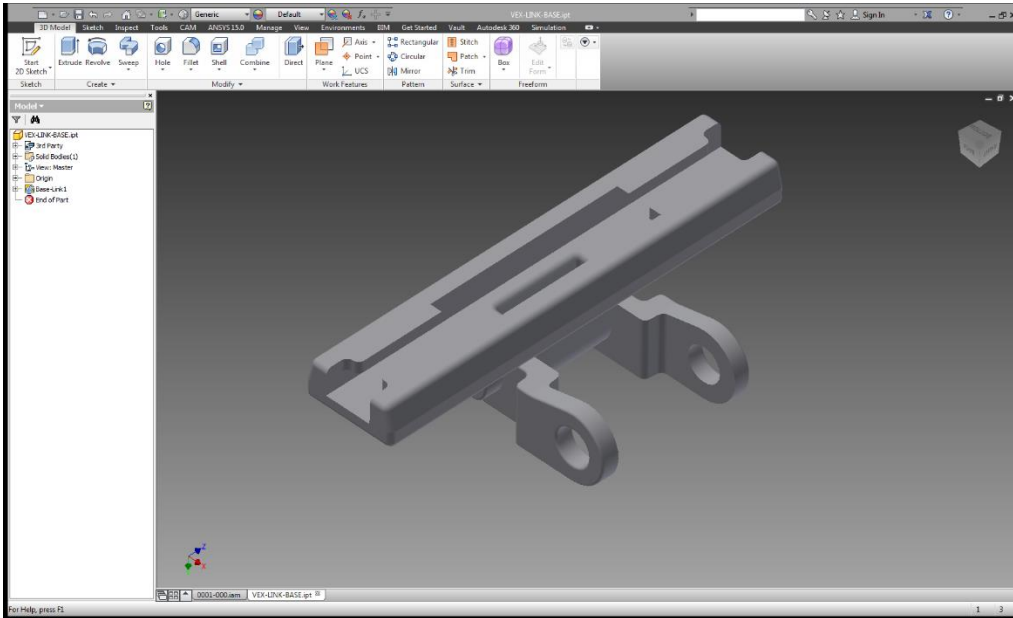
This guide is written for intermediate user of Autodesk Inventor, but the steps should be easy to follow if you know the basics of the inventor. For the tutorial, VEX tank tread and sprockets were used, but the method can be used for any other chain or tread application. The ultimate objective of this guide is to show you and teach some advanced way of patterning components or features, and using adaptive drawing/part to create modifiable tread assembly. If you have any problem, question, or suggestion, feel free to contact me.

skim449@illinois.edu

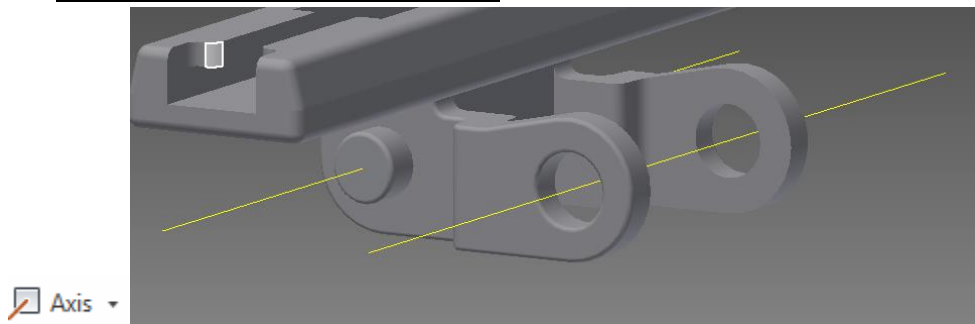
2. Prepare the Link

A. Open the link

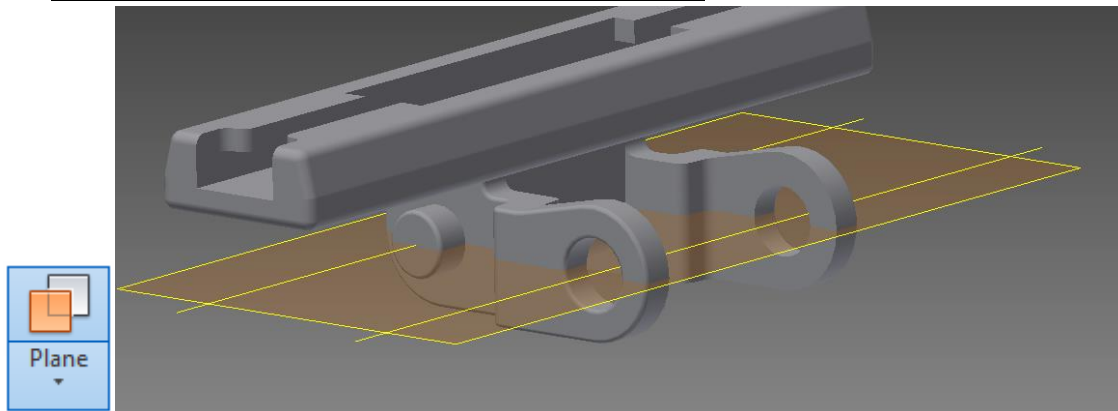
(VEX-LINK-BASE for example)




B. Create two axis through the link

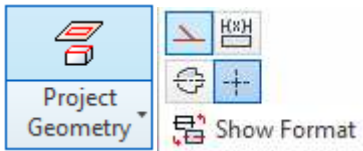


C. Create plane: through two lines (Work Plane 1)

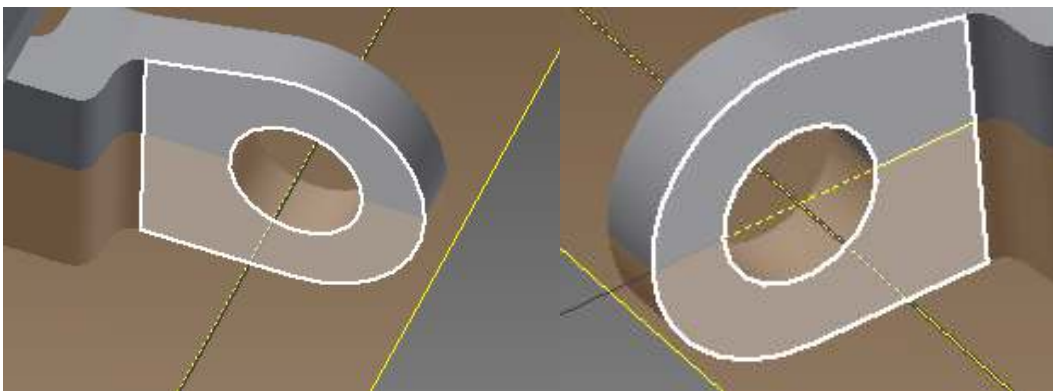



D. Create 2D Sketch on the Work Plane 1

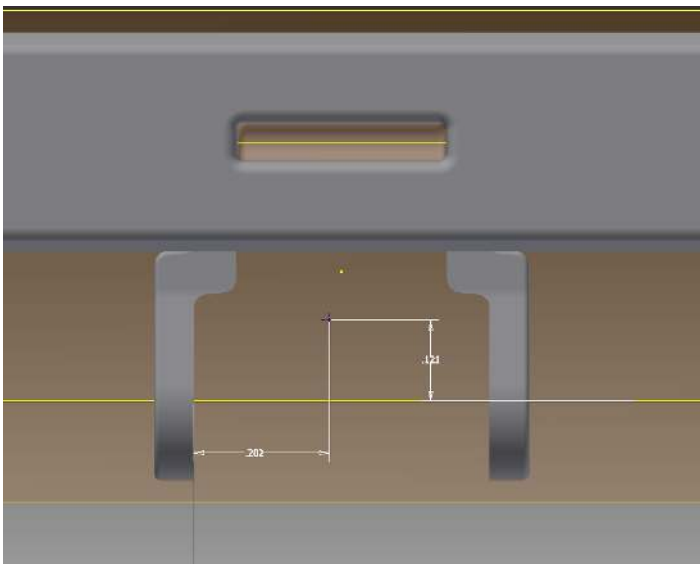
- i. Click 'Start 2D Sketch'
- ii. Click Work Plane 1 on the Model Tree
- iii. Click 'Project Geometry' and 'Construction' 



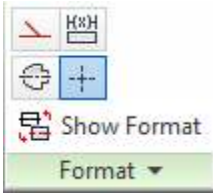
- iv. Click two axis and two surface on the top



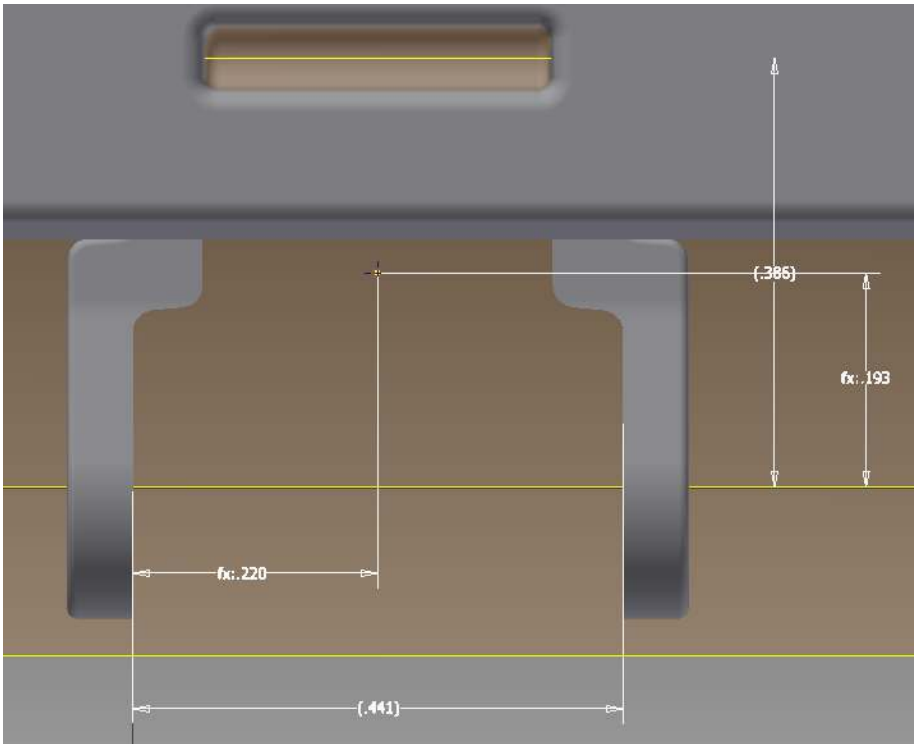
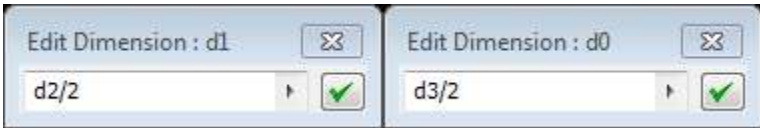
- v. Right-click & hold the mouse, click Ok. (or esc)
- vi. Create a point and two dimensions  Point




- vii. Create reference dimension 



viii. Set the point in the middle (Click the reference dimension and divide it by 2)



ix. Go to 'Parameters'  (Manage – Parameters)

Parameter Name	Unit/Typ	Equation	Nominal Value	Tol.	Model Value	Key	Comment
Model Parameters							
d0	in	d3 / 2 ul	0.193000		0.193000		
d1	in	d2 / 2 ul	0.220472		0.220472		
Reference Para...							
d2	in	0.441 in	0.440945		0.440945		
d3	in	0.386 in	0.386000		0.386000		
User Parameters							

$E = mc^2$ $P + \rho \times \frac{1}{2} v^2 = C$ $E = mc^2$ $P + \rho \times \frac{1}{2} v^2 = C$ E
 $\nabla \times E = -\frac{\partial B}{\partial t}$ $\Delta S_{\text{universe}} > 0$ $\nabla \times E = -\frac{\partial B}{\partial t}$ $\Delta S_{\text{universe}} > 0$

Add Numeric Update Purge Unused Reset Tolerance << Less Done
 Link Immediate Update + ▲ ● -

- x. Rename the distance between the axis (reference parameter d3, for example) to 'LinkLength'

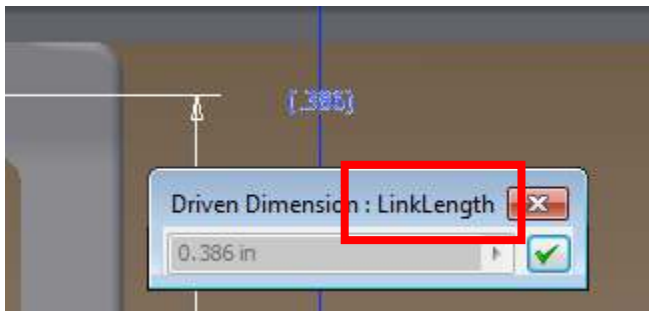
d2	in	0.441 in	0.440945	●
LinkLength	in	0.386 in	0.386000	●
User Parameters				

- xi. Click 'Export Parameter'

d2	in	0.441 in	0.440945	●	0.440945	<input type="checkbox"/>	<input type="checkbox"/>
LinkLength	in	0.386 in	0.386000	●	0.386000	<input type="checkbox"/>	<input checked="" type="checkbox"/>
User Parameters							

- xii. Click Done

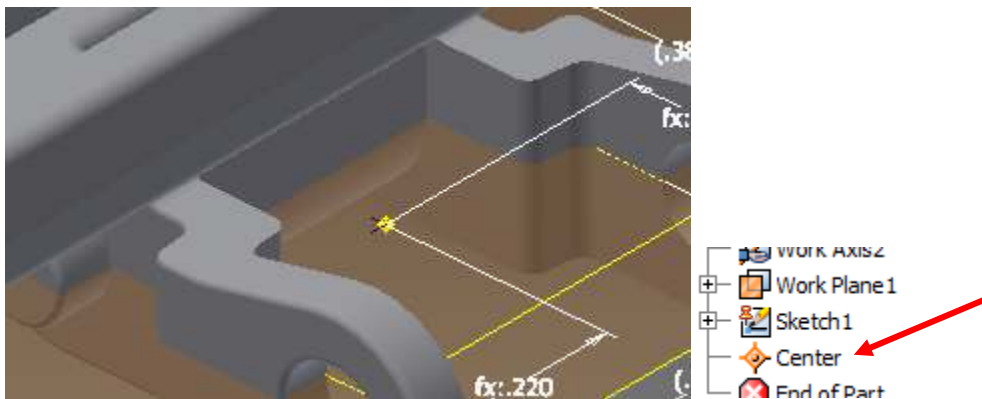
- xiii. Double click the reference dimension and check the name



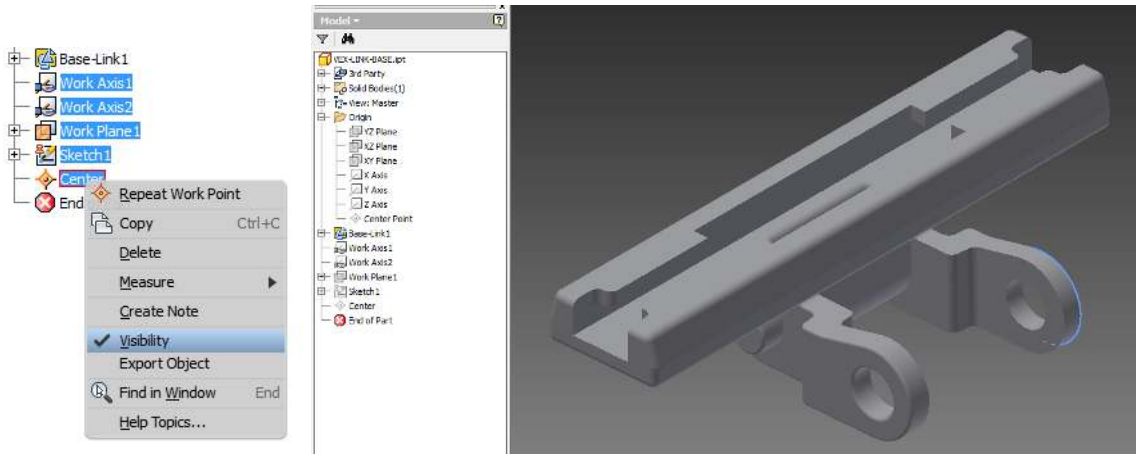
- xiv. Finish Sketch

E. Create Work Point at the sketch point, rename the point to 'Center'

◆ Point ▾

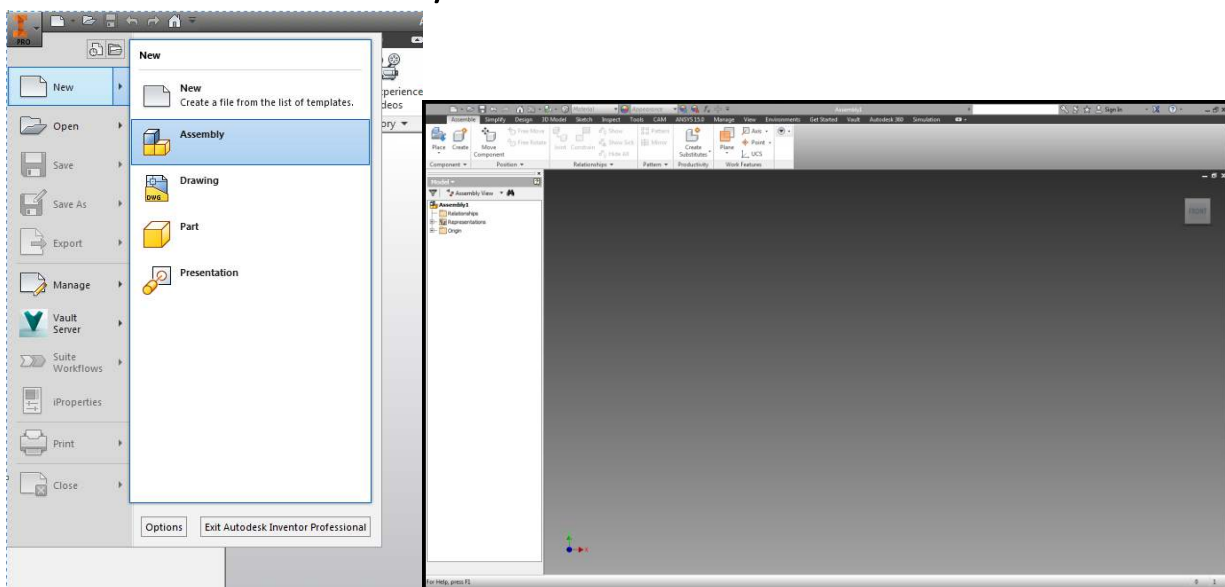


F. Turn off the visibility of everything except the model



G. Save and close the link

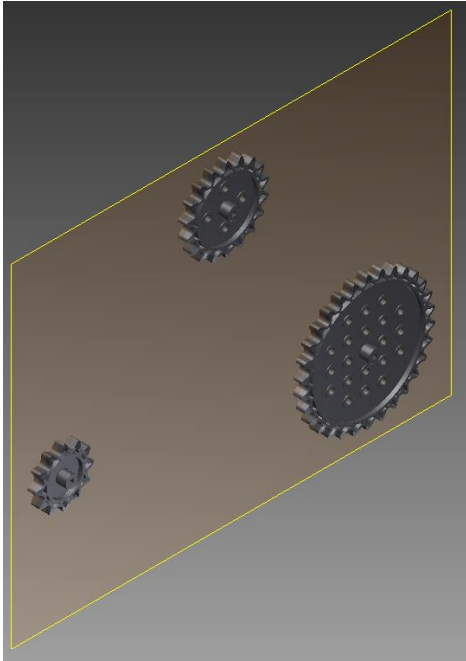
3. **Create the Assembly File and Save**



4. **Place the sprockets**

It is arbitrary: it depend on the design of the tread.

- i. Align all the sprocket in the same plane. (If necessary, make the work plane on the sprocket part file)
- ii. Ground one of the sprocket



5. Create Path File

A. Click Create



B. Name the component and choose the template

(Choose .ipt, standard inventor part extension, for the template)

New Component Name	Template
0001-PATH-01	Standard.ipt

C. Choose 'Reference' for the BOM structure

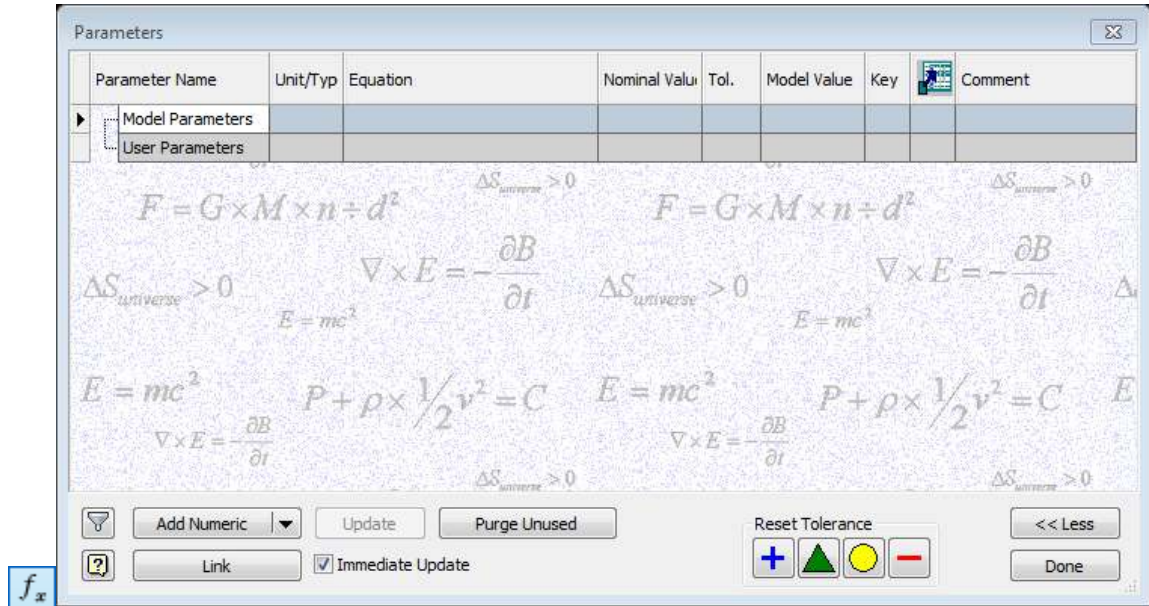
Note: This part file will be used as a reference to make the tread, therefore it should not be included in the BOM list.

Default BOM Structure	
 Reference	<input type="checkbox"/> Virtual Component

- D. Click Ok
- E. Click anywhere on the screen to place and create part
- F. Save

6. Import Dimension from Link

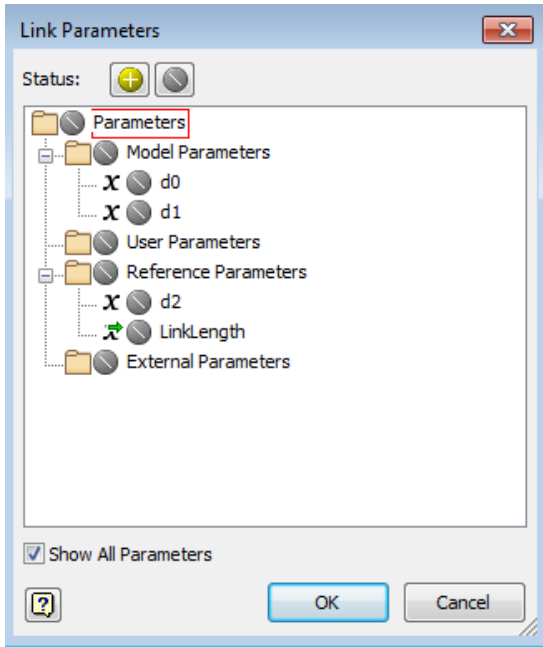
- A. Go to Parameters



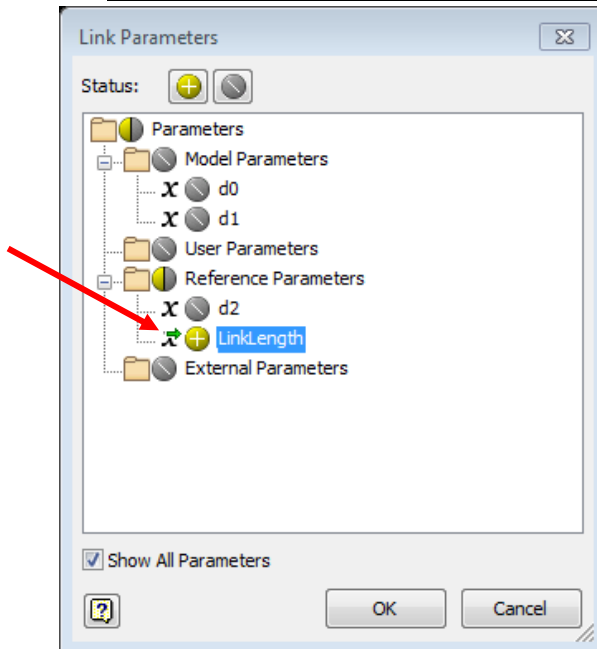
- B. Click 'Link'

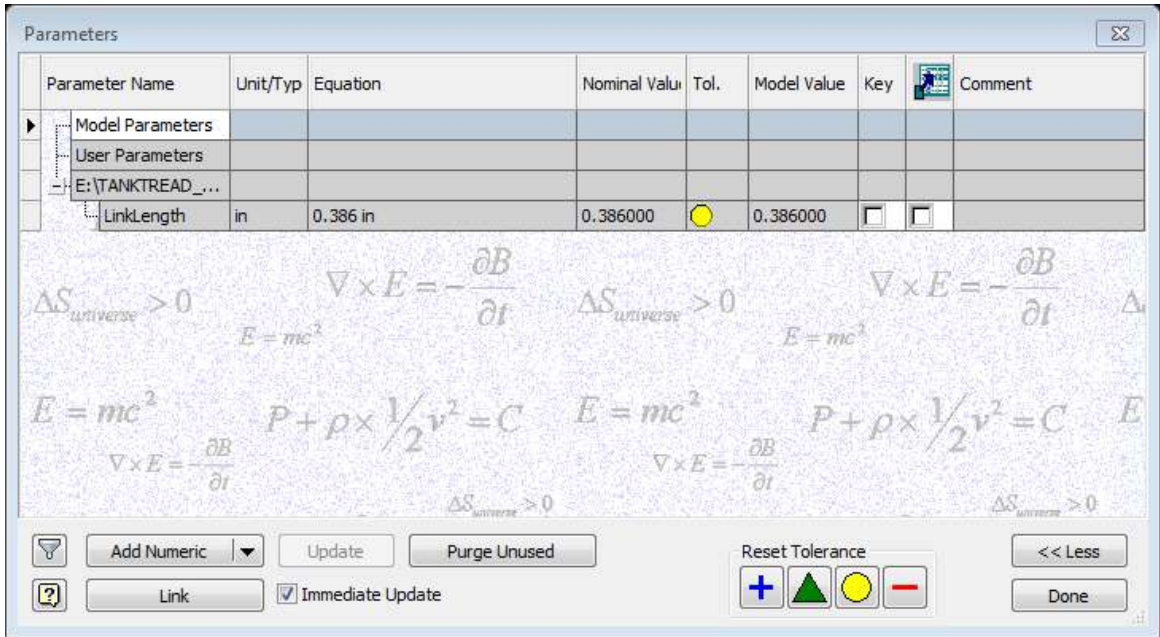


- C. Change file of type to inventor files, and open the link file
(VEX-LINK-BASE for example)



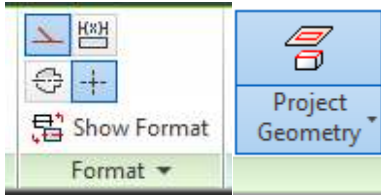
D. Click and derive the dimension 'LinkLength', click Done



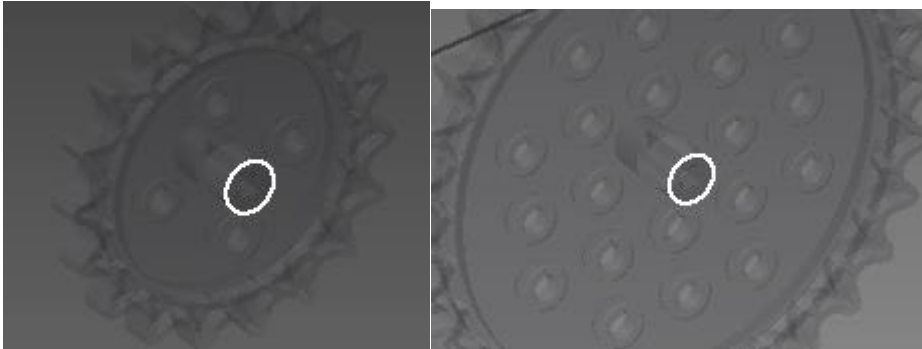


7. Draw the Path

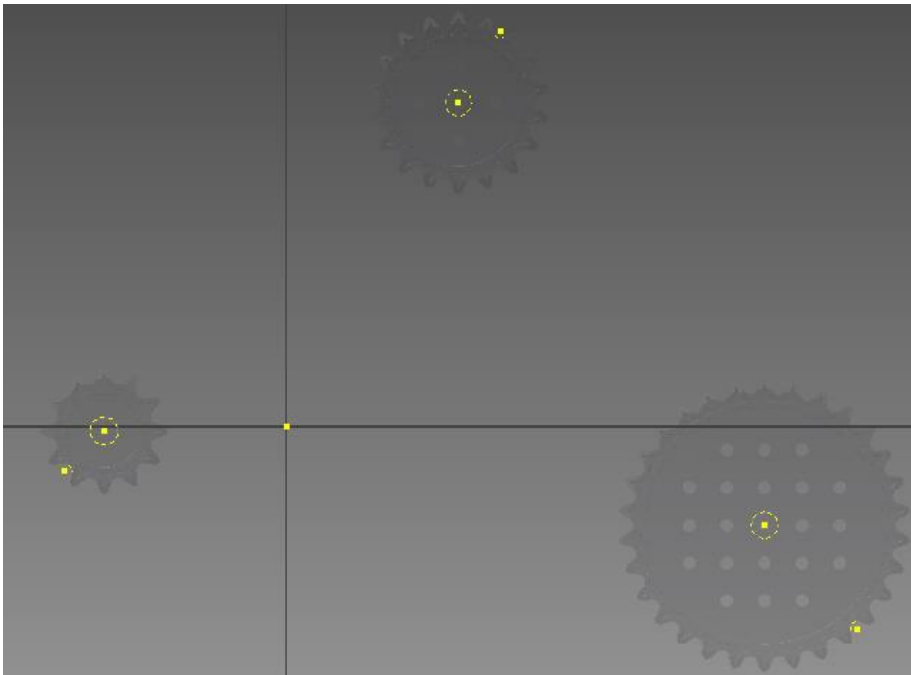
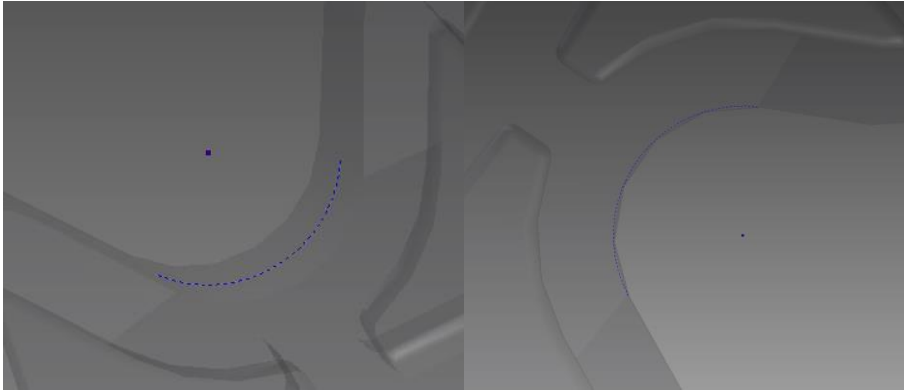
- Click 'Start 2D Sketch'
- Click the plane that is coplanar with sprockets
- Click 'Project Geometry' and enable 'Construction'



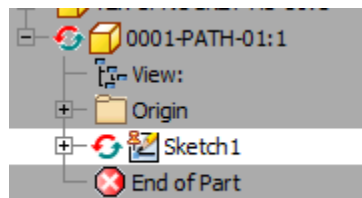
- Create the construction line to find center of the sprockets




E. Create the construction point to find the center of the sprocket teeth

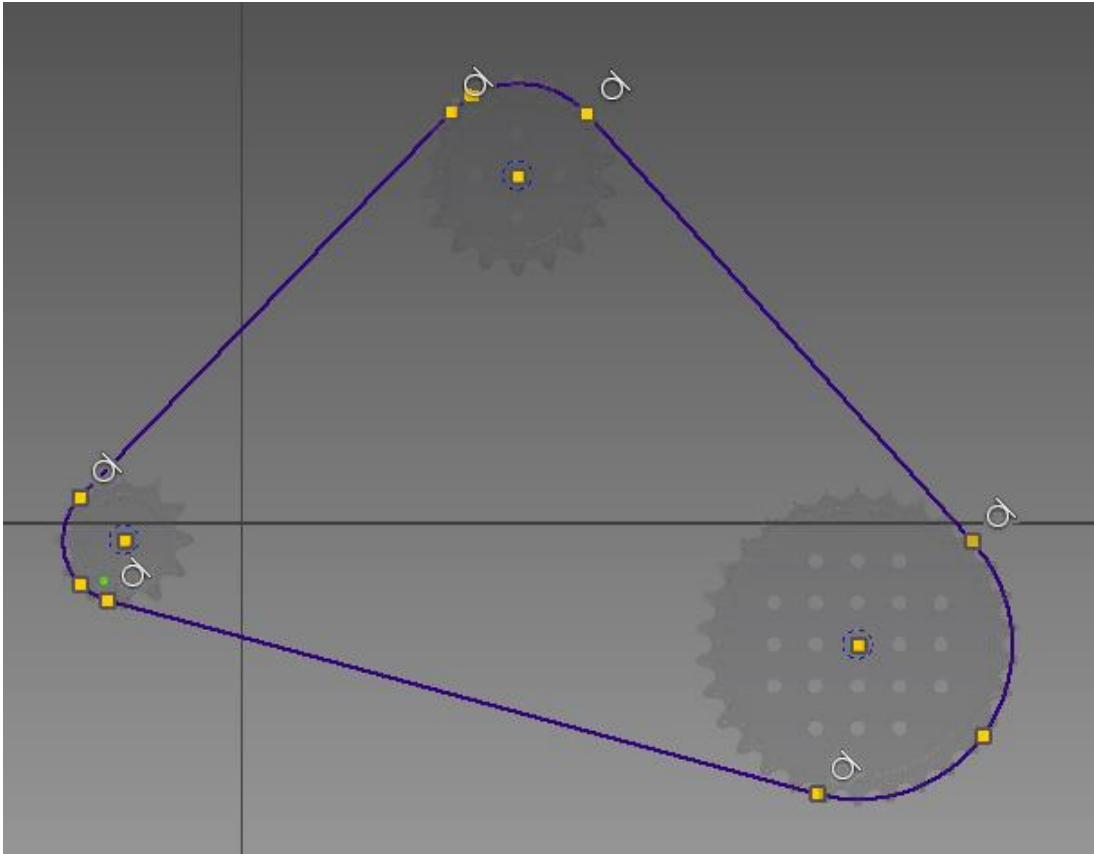


Notice: Adaptive is enabled for the Path file.



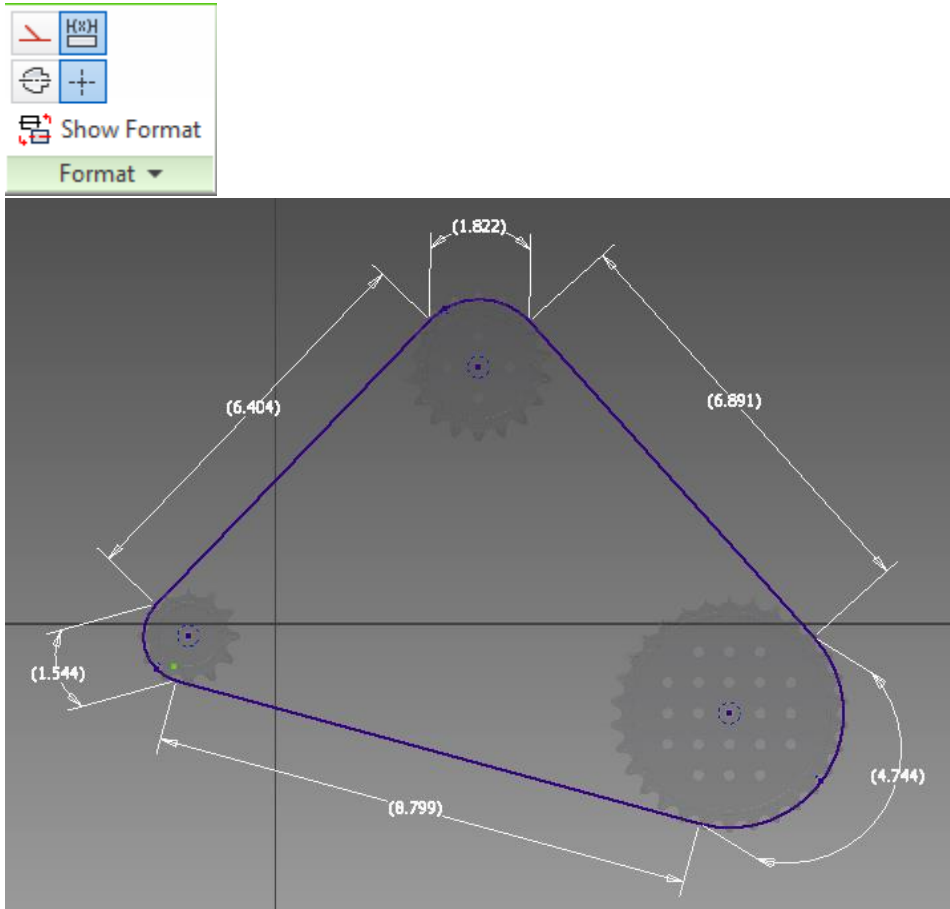
F. Disable the construction 

G. Draw the Path



(Make sure to have tangent constrain between arc (or circle) and lines.)

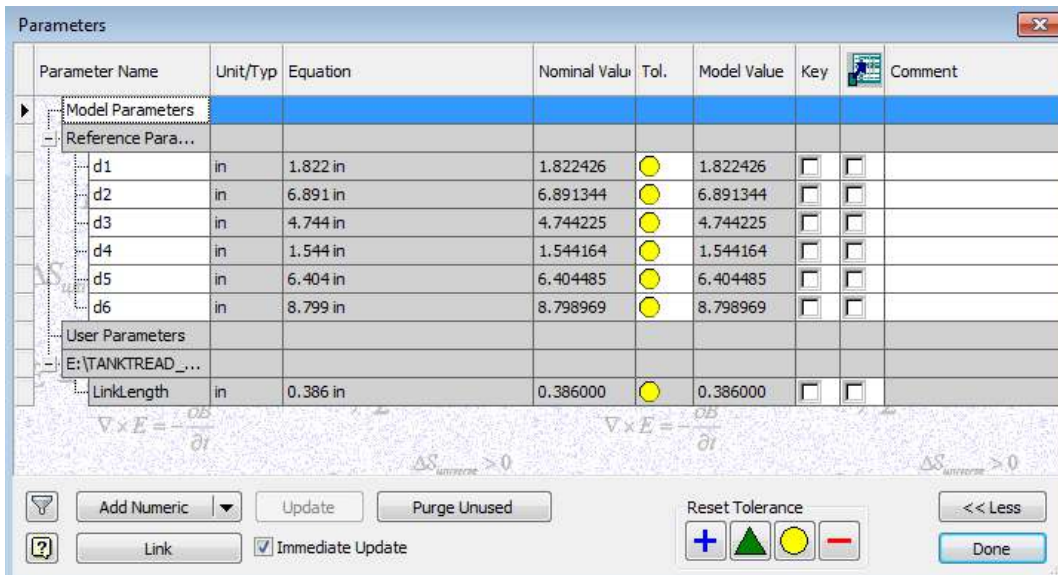
H. Create reference dimension of the path



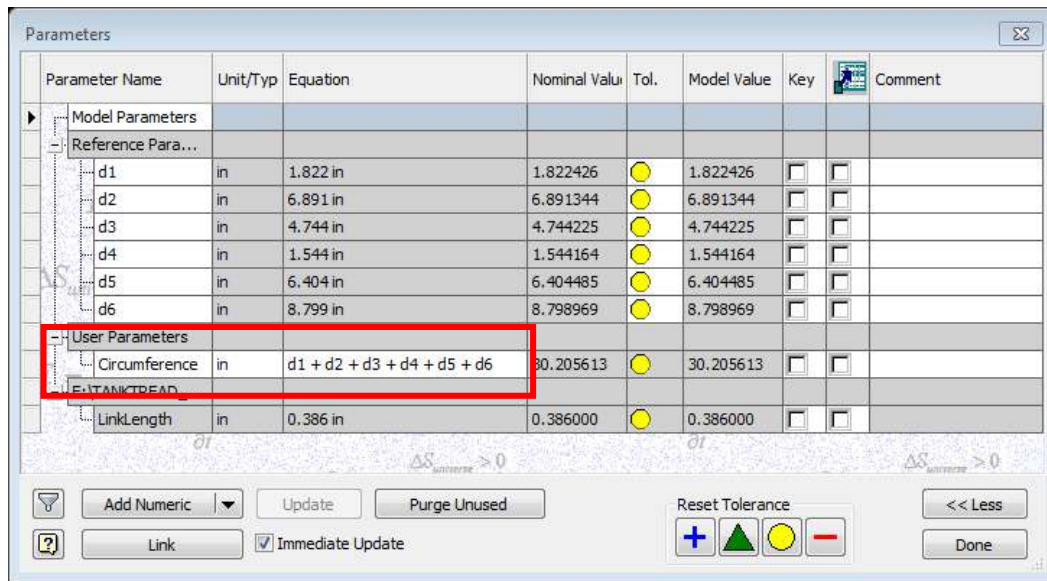
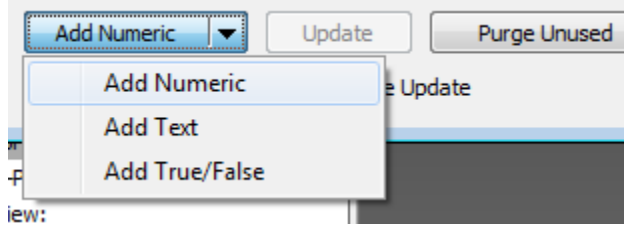
Note: While dimensioning, you can change the type of dimension (ex. aligned, arc length) by right-click.

I. Finish the sketch

J. Open Parameters



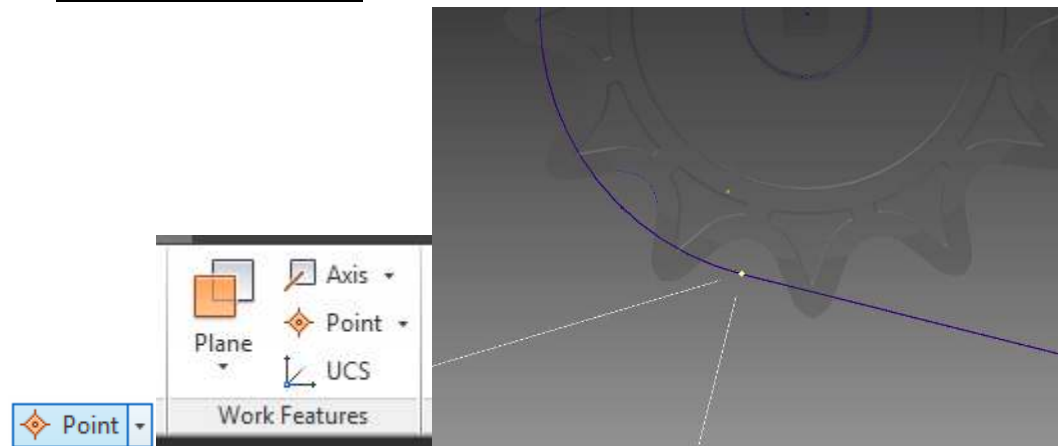
K. Add Numeric User Parameters for 'Circumference'



L. Click Done

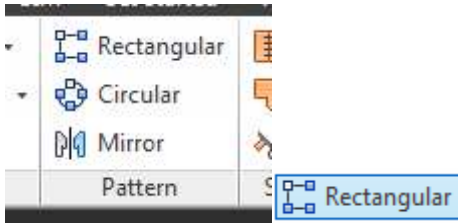
8. **Create Point Pattern**

A. Add point on the path

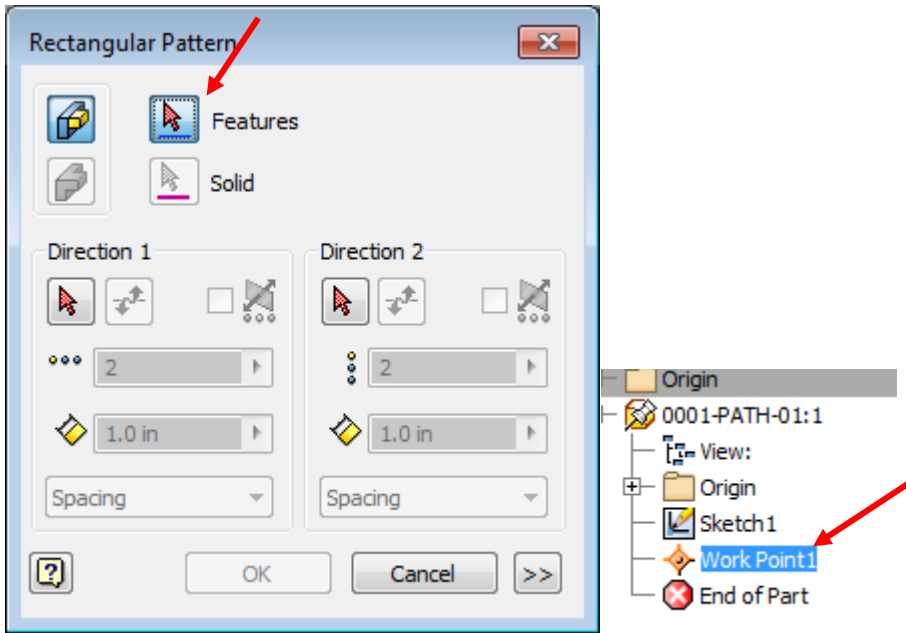


B. Pattern the work point

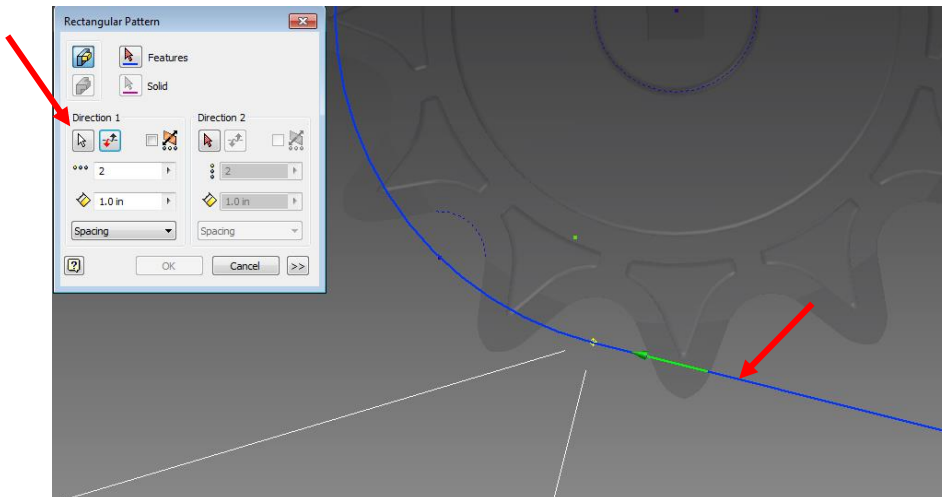
i. Click Rectangular Pattern




ii. Click Work Point on the model tree for Features

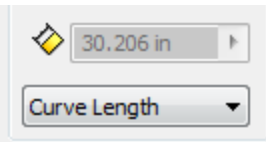


iii. Select Direction 1 (Click  and click the path sketch)

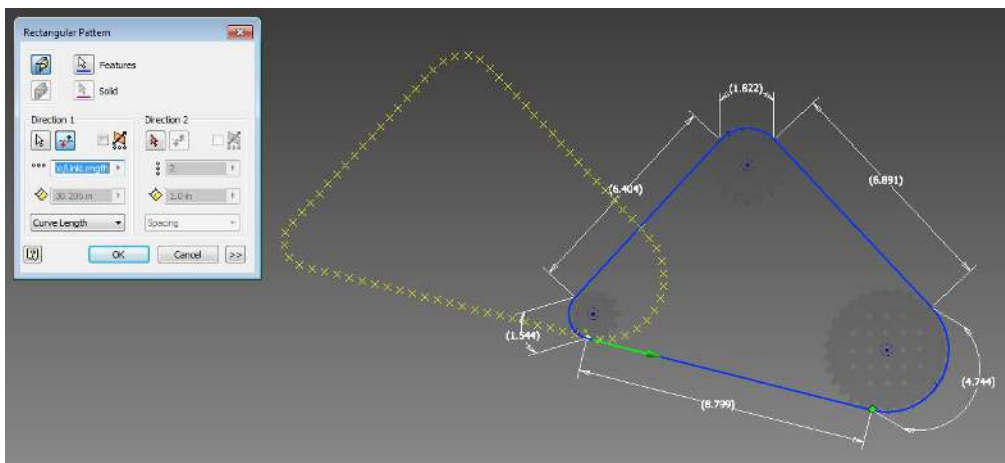
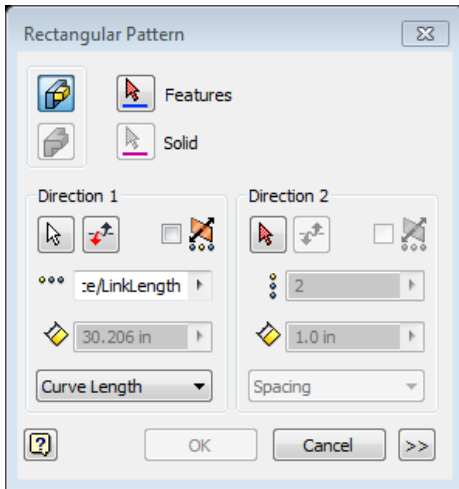


Notice the green arrow for the direction. You can flip the direction with 

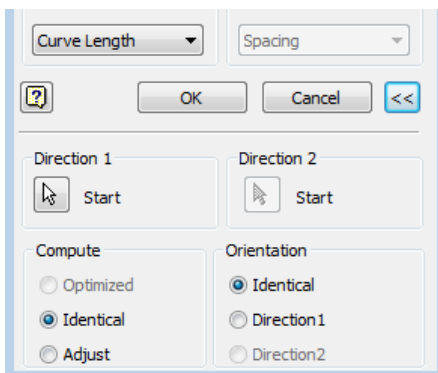
iv. Change 'Spacing' to 'Curve Length'




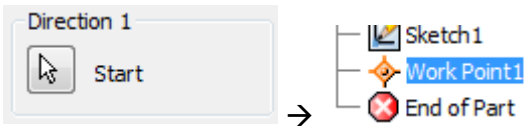
v. Type 'Circumference/LinkLength' for spacing



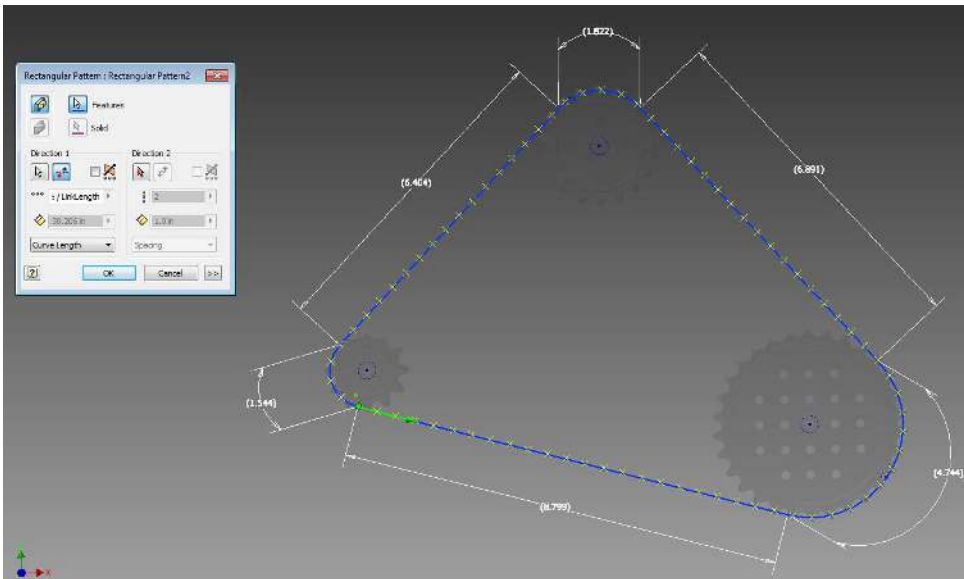
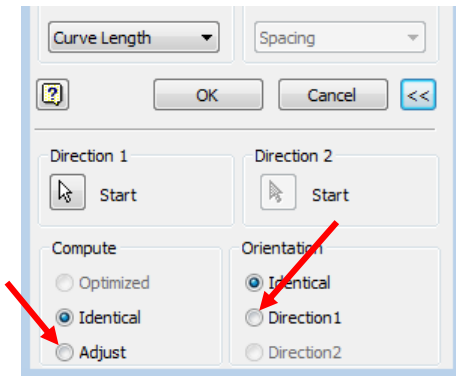
vi. Open detail settings



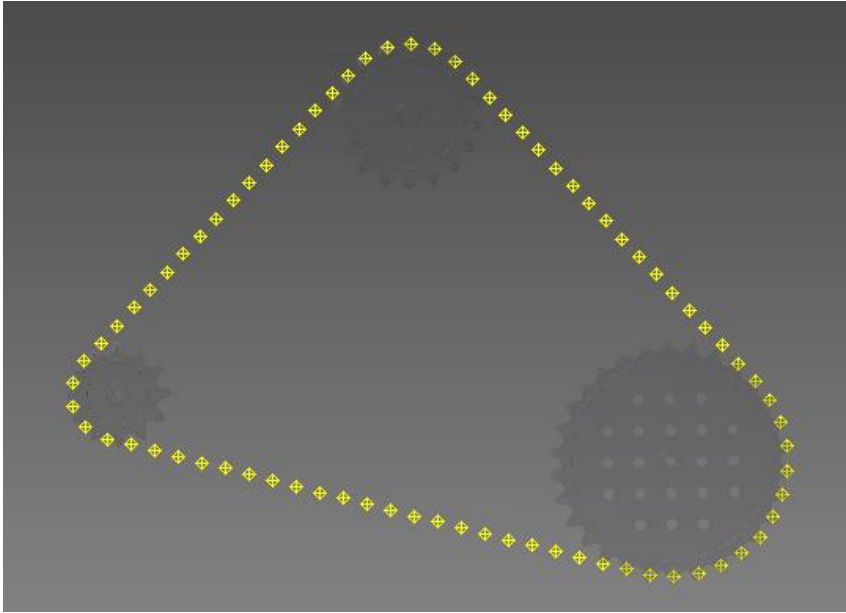
- vii. Change the starting point of the direction 1 to the work point
(Click  Start and click Work Point on the Model Tree)



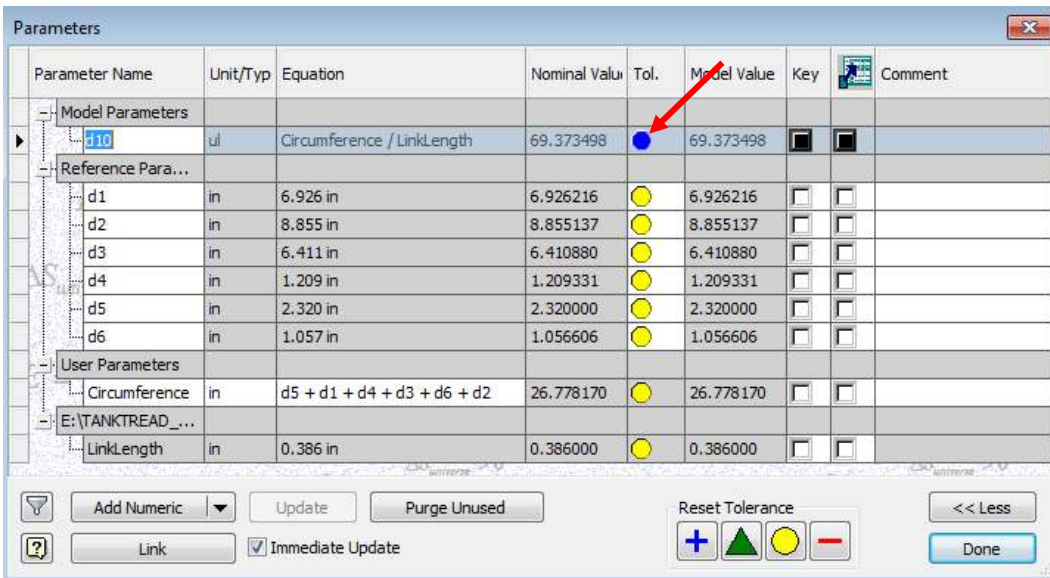
- viii. Change Compute to 'Adjust', and change Orientation to 'Direction1'



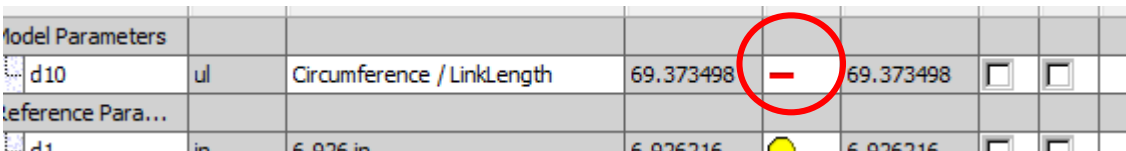
- ix. Click Ok



x. Open the Parameters



xi. Change the tolerance of d10(number of link) to less



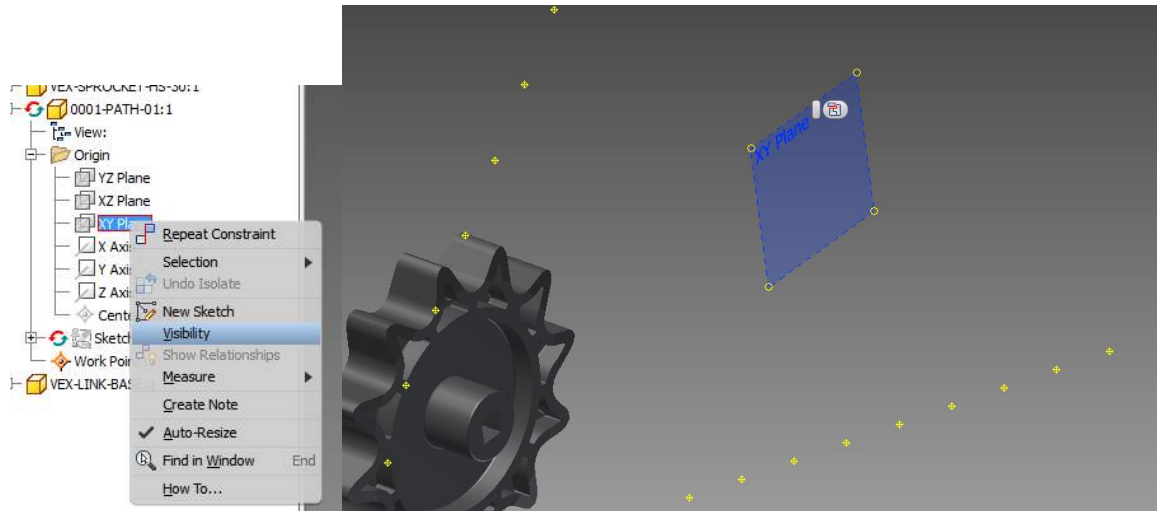
xii. Click Done

C. Return to the top assembly

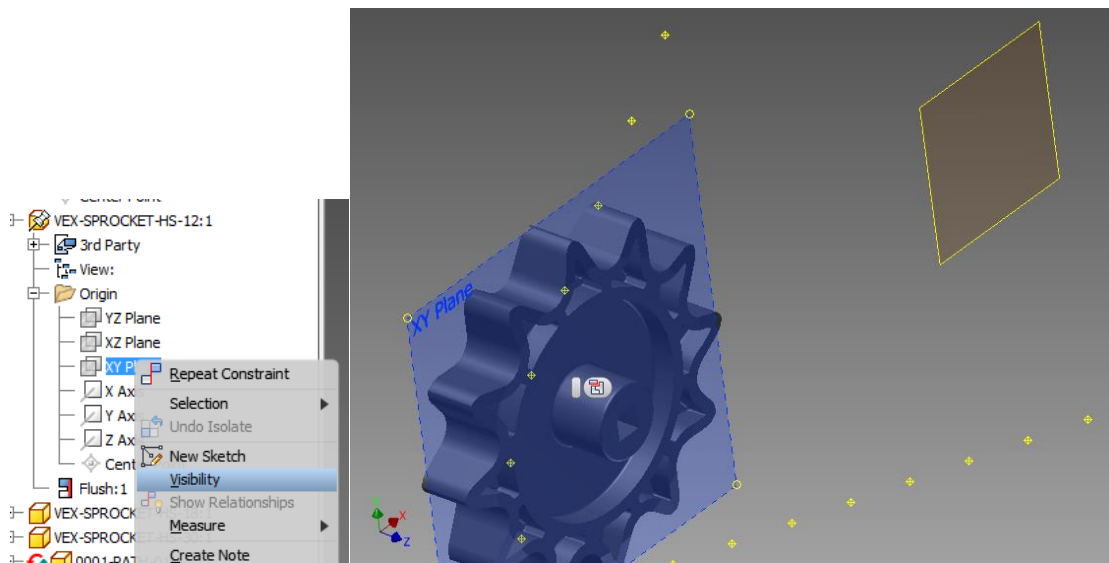


9. Constrain the Path

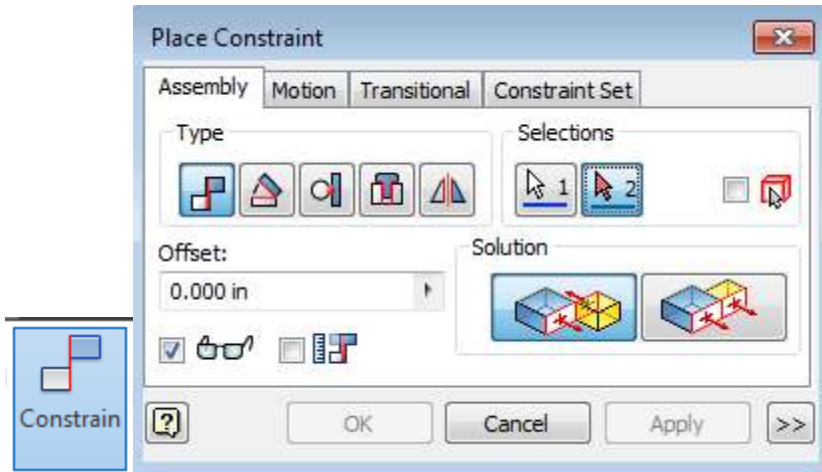
- i. Click the plane with the points, and turn on the visibility.



- ii. Find the parallel plane on the grounded sprocket, and turn on the visibility



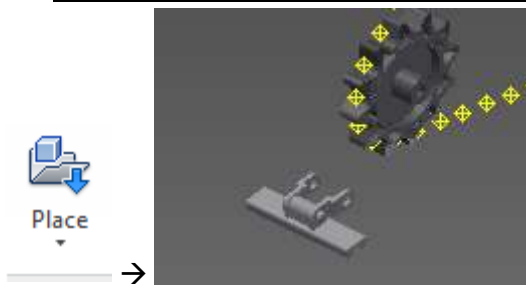
- iii. Constrain using Flush



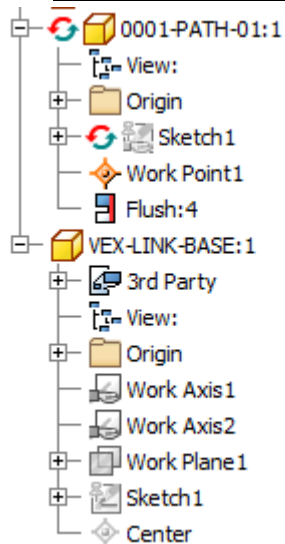
- iv. Turn off the visibility

10. Place the Link

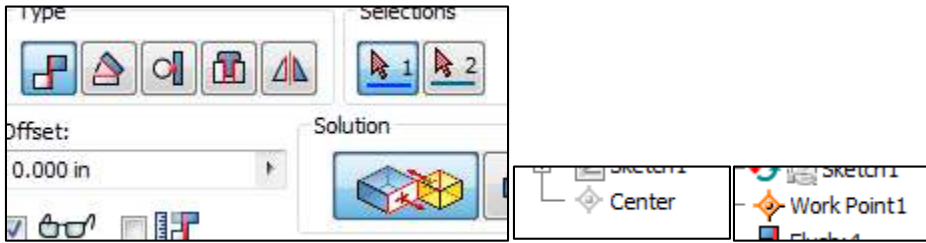
- A. Click place, select the chain/tread link, and place on the screen



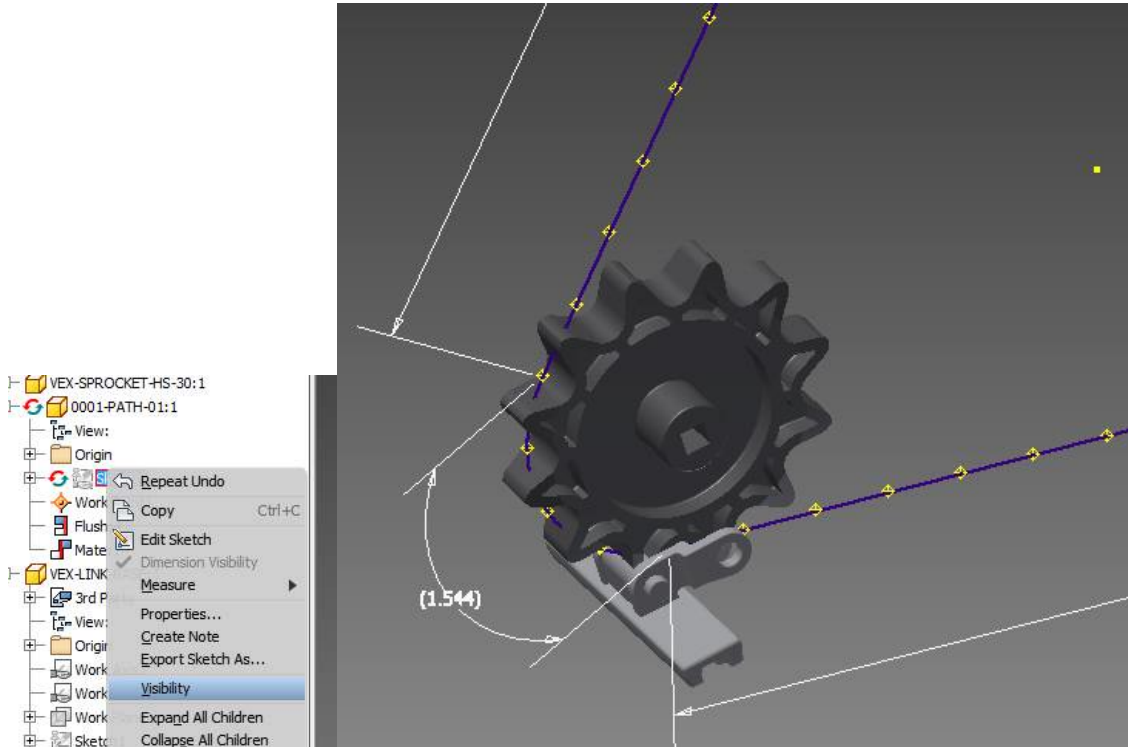
- B. Expand the model tree for Path and Link



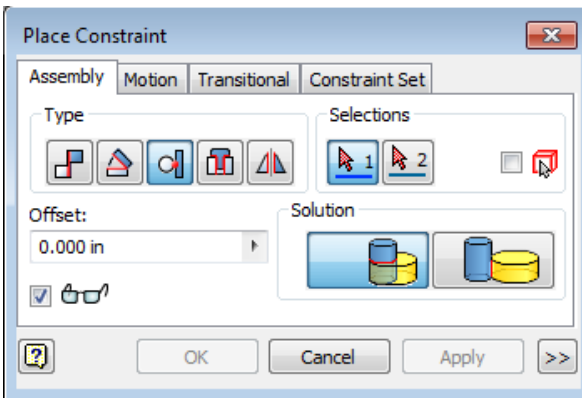
C. Using Constrain, mate the 'Center' from the Link to 'Work Point' on the Path

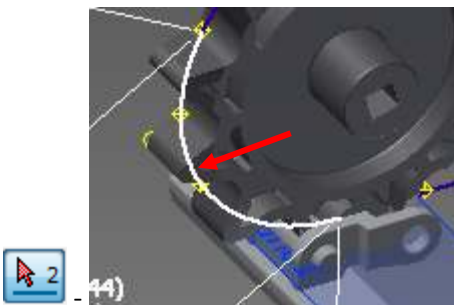
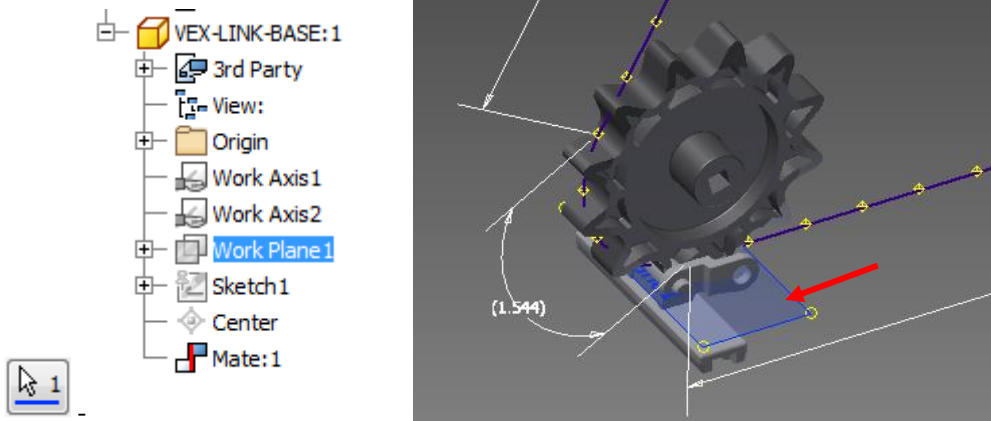


D. Turn the visibility of the sketch of path



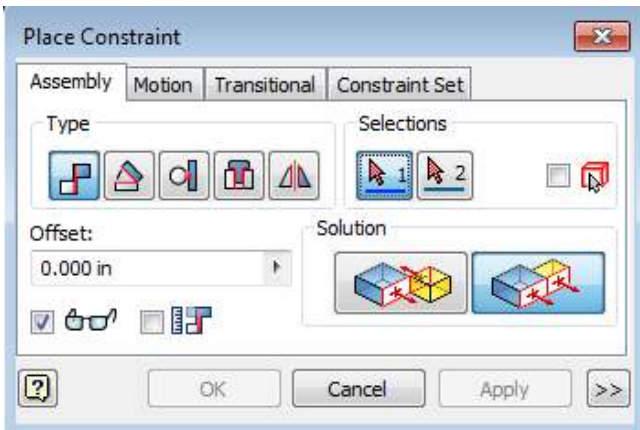
E. Create tangent constraint between Work Plane of the link and the arc path

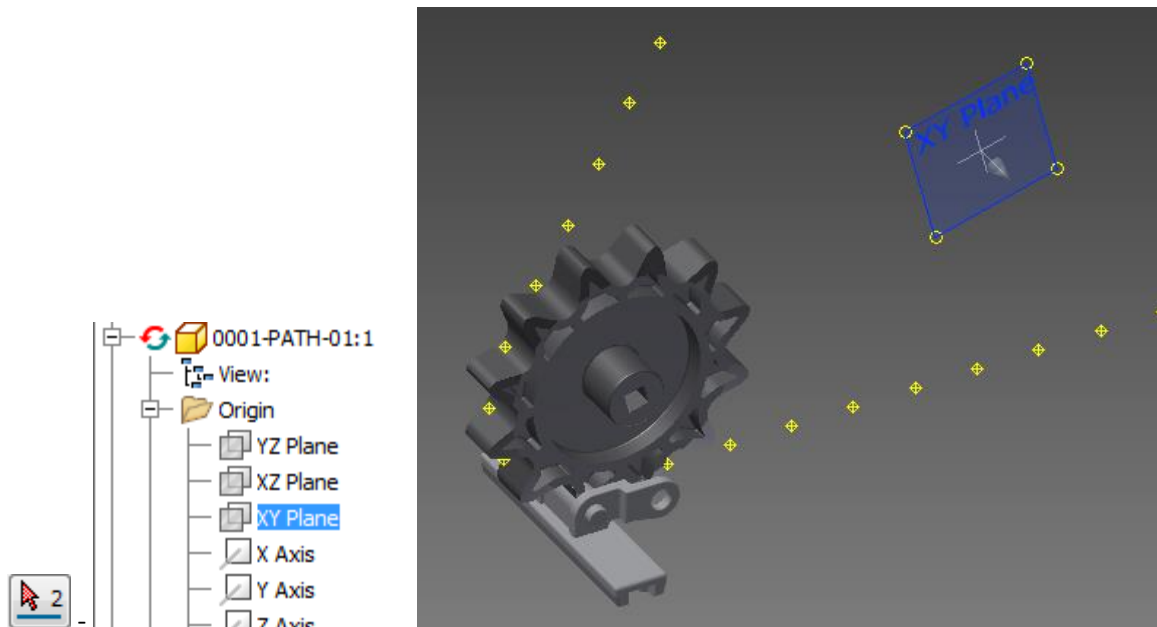
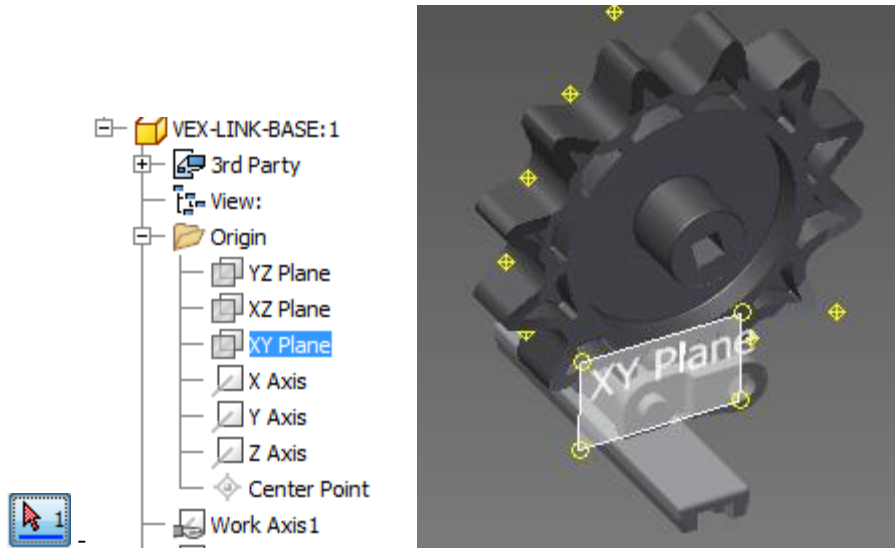




F. Turn off the visibility of the path sketch

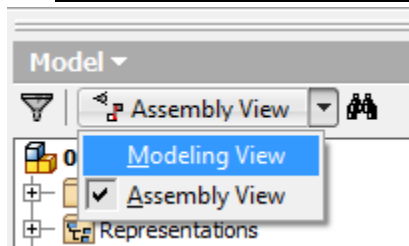
G. Constrain (flush) the vertical plane of the link with the plane of the path





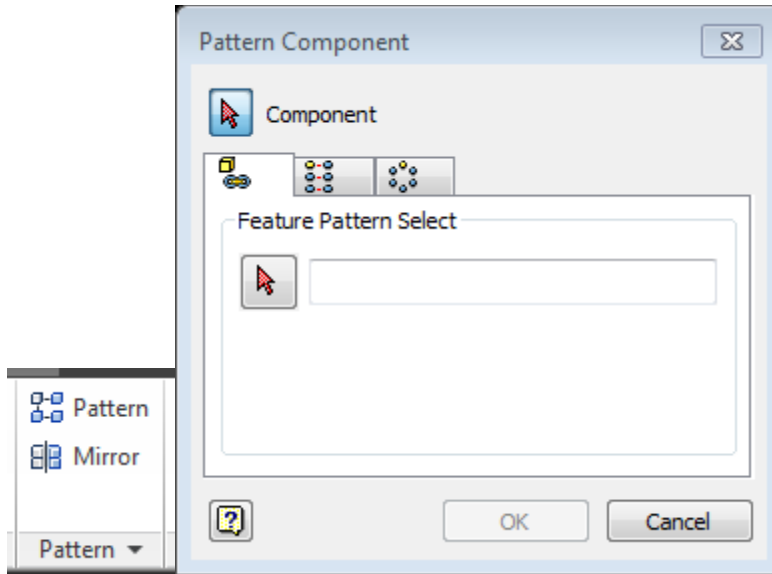
If this constraint gives an error, it means sprocket, path, or link are not aligned.

H. Change the model tree from 'Assembly View' to 'Modeling View'

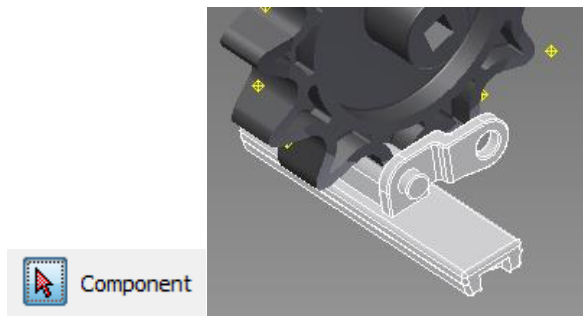


I. Create Link Pattern around the path

i. Click Pattern



ii. Click the link for the Component

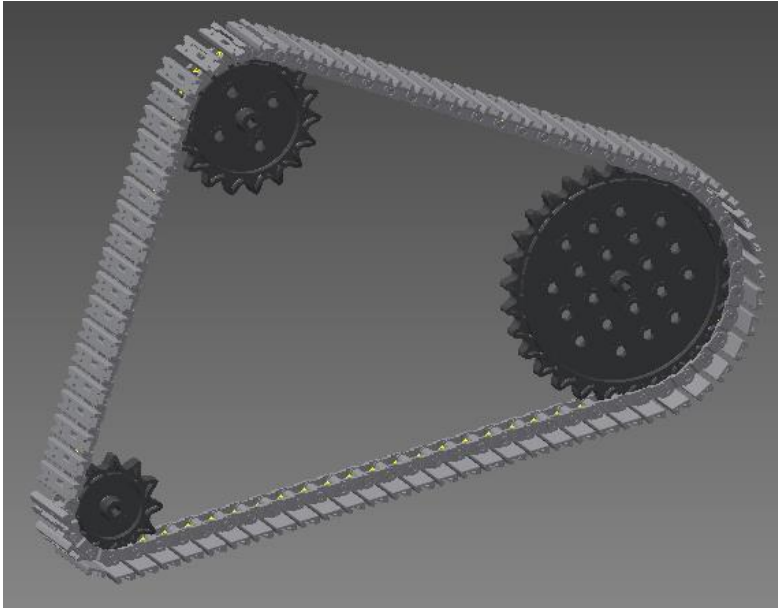


iii. Click the rectangular pattern in the Path for 'Feature Pattern Select'

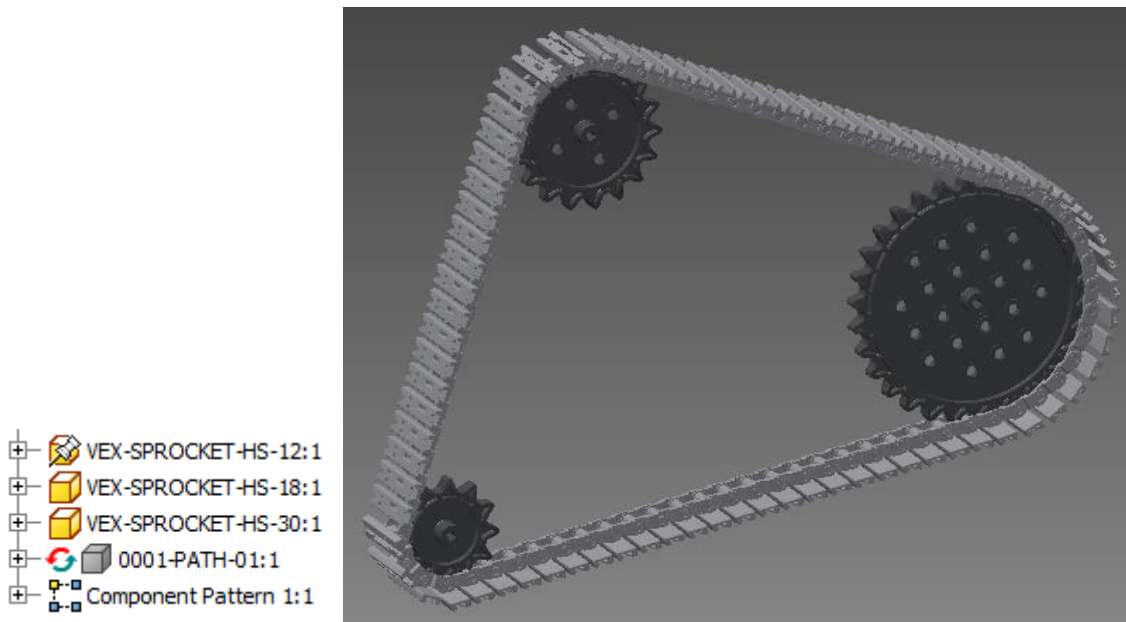


Note: If you don't see the pattrer, make sure you change the model tree to 'Modeling View.' (step 10.H)

iv. Click Ok





- v. Turn the visibility off for the Path

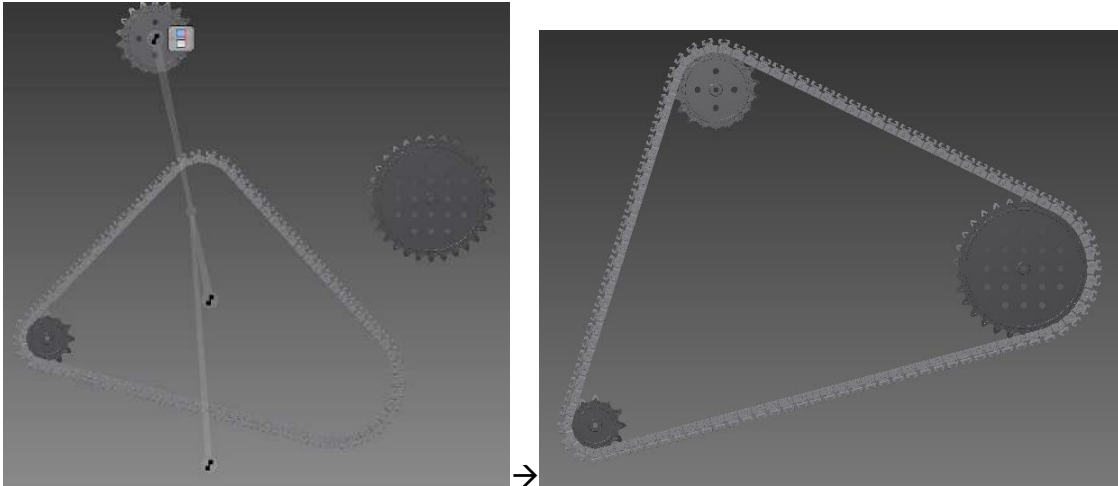


11. Capability and limitation of this method

A. Capability

- 1) Easy to change the location of the sprocket

For testing, try to move the non-grounded sprocket using  Free Move . By pressing update  (Manage – Update), the path and tread will be recreated around the sprocket at new location.



This is because of the Path is 'adaptive,' and the pattern is completely depend on the path file. The adaptive drawing is referencing the location of the sprocket, which is why moving the sprocket will regenerate the path drawing.

This feature is useful when the length/shape of the tread is not yet defined, but still want to construct a CAD file that can be adapted to any design.

2) Reduced amount of computation

In the Autodesk Inventor, the main reason of slowing down the computer is because of constrain. Every single constrain is calculated in the matrix to find the location and orientation of each part. In this case, or any other chain/tread example, if each link of chain or tread takes 3 constrain to be fixed at some location, the amount of computation increases exponentially. This method only constrain one link to the point, and all other parts are copied by the pattern, which reduces the amount of calculation significantly.

3) Presentation

Visually, it creates almost perfect linkage around the path. The limitation usually comes when the curvature of arc is too large.

4) No limit on the number of sprocket

B. Limitation

1) No Constrain

Since all the chains are held without constrain, any mechanical and static/stress analysis are impossible to do in the software. Also, technically parts are not held together.

2) No Animation

Practically, there is no easy way of making tread moving animation in the inventor in this way. It is possible to make a fake animation, or make it seems to be

moving around the path, which is not included in this guide, but it is impossible to make the tread to actually move around the path. (If you really want to make one, try to move the starting point around the path)

12. Some Tip

- 1) Don't include any other component in the tread assembly, and only keep sprockets, path, and link on the assembly.
 - a. Include shaft, motor, support, or any other parts in the parent assembly file, and use this tread assembly as a sub-assembly from the parent assembly.
- 2) Turn the additivity of the path off when it is not in use: it takes more computation to calculate adaptive part or features.