A V A T E C H S O L U T I O N S, I N C.

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Animating a Chain in Autodesk Inventor

Procedure

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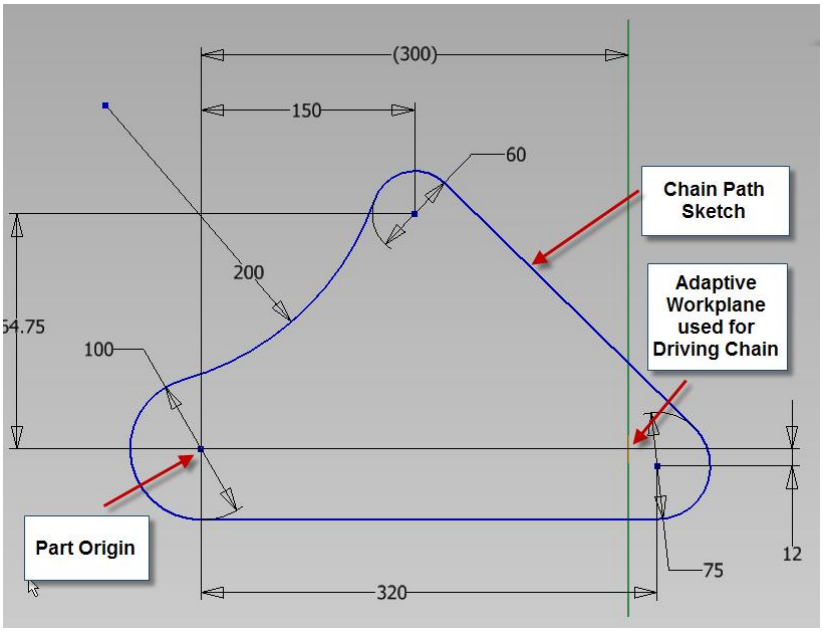
Introduction

The process of animating a chain in Inventor has been documented on the Internet but I have found the instructions lacking or sometimes in a foreign language which often lose clarity when translated. I have also found the occupying graphics do not fully describe the process leaving gaps that often lead to failure. This document does not in any way claim to be the only method to animate a chain in Autodesk Inventor, but my intention is to clearly describe one. All screen shots and procedures are captured from Autodesk Inventor 2010.

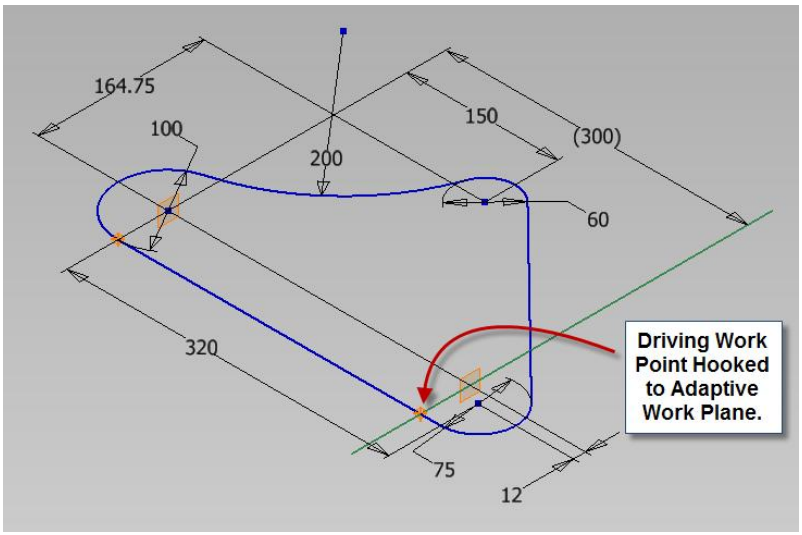
I would like to acknowledge my two resources that helped me put together this procedure. The first is a procedural document by John Tellema which is located at this Internet address:
http://www.inventorwizard.be/tipntricks/assembly/assembly_werkende_ketting.pdf
This document is written in Dutch. The second resource is Stan Wile of Avatech Solutions who translated this document and sent it to me.

Basic Theory Behind the Animation

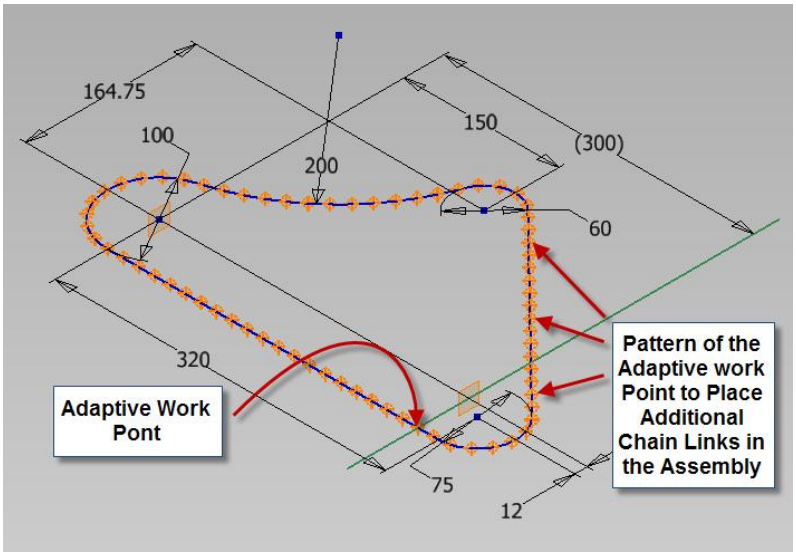
The process begins by creating an Inventor part containing a fully constrained sketch which represents the chain path for your application. This sketch is constrained to the origin by the center of one of the chain sprockets. An adaptive work plane is placed in the sketch which will be hooked to one link of the chain. When this sketch plane is constrained and driven in the assembly the chain will be driven along the chain path.



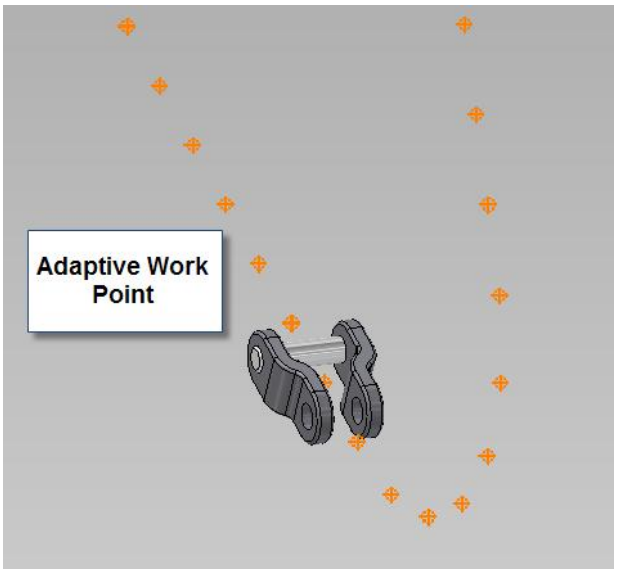
A driving work point is parametrically hooked to the adaptive work plane using a pattern feature. The driving work point will be used in the assembly to constrain the first link of the chain. When the adaptive work plane is driven, the first link of the chain will move along the chain path dragging the rest of the chain with it.



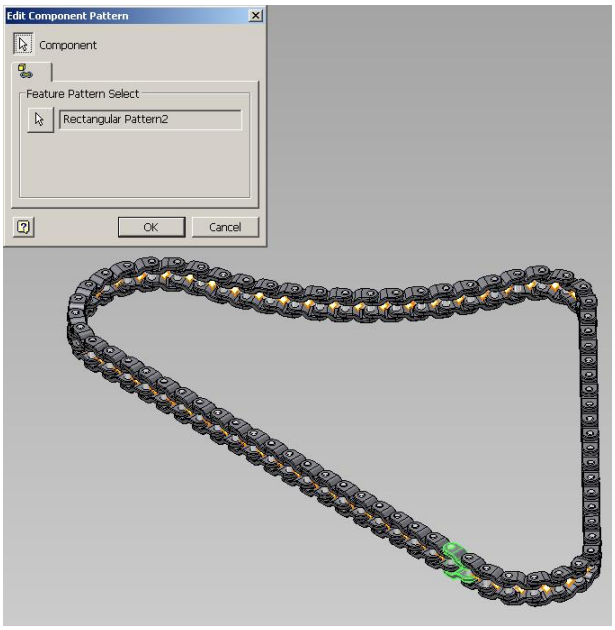
The adaptive work point is pattern to represent the pitch of the chain and will serve as attachment points for the rest of the links in the chain.



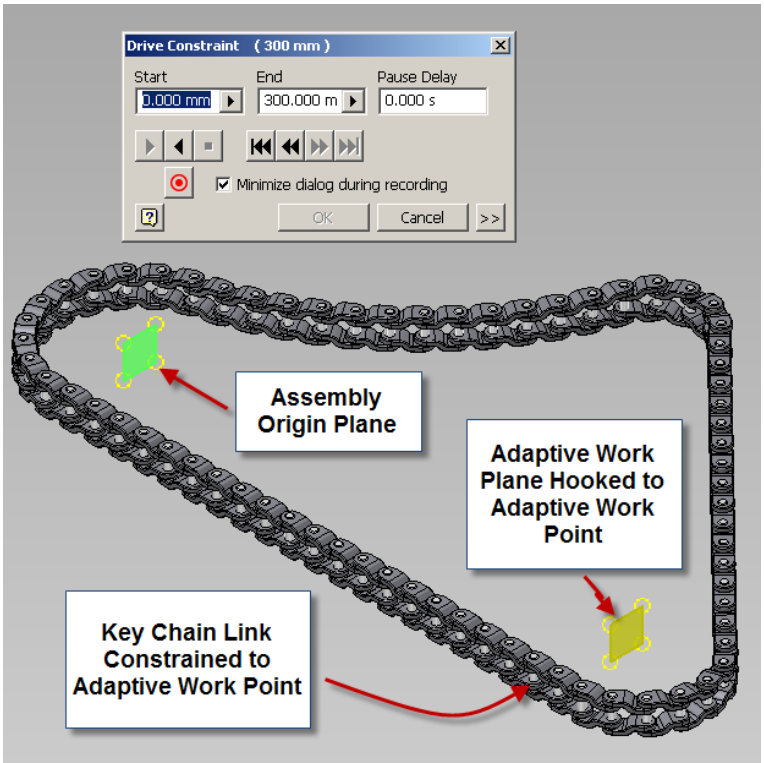
The last step is to create the chain assembly by placing the chain path part and one link of the chain into the assembly model. The link is constrained by a key point to the adaptive work point and oriented to the assembly origin using constraints.



An associative pattern completes the chain along the path. Your chain assembly is finished.

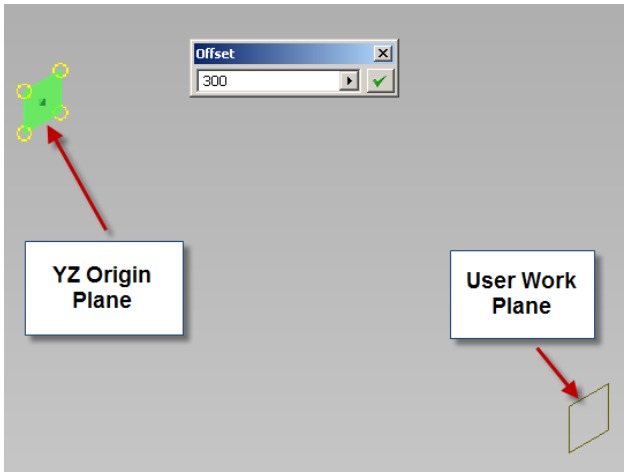


Driving the assembly constrain attached to the adaptive work plane drives the chain along the path.

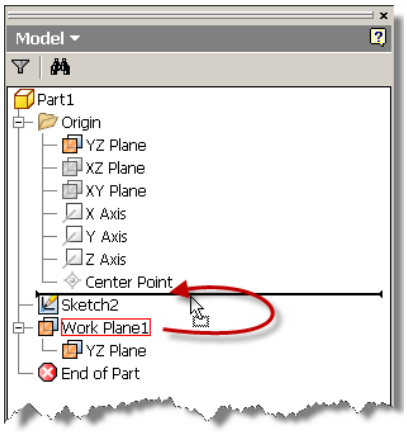


Creating the Chain Path Part

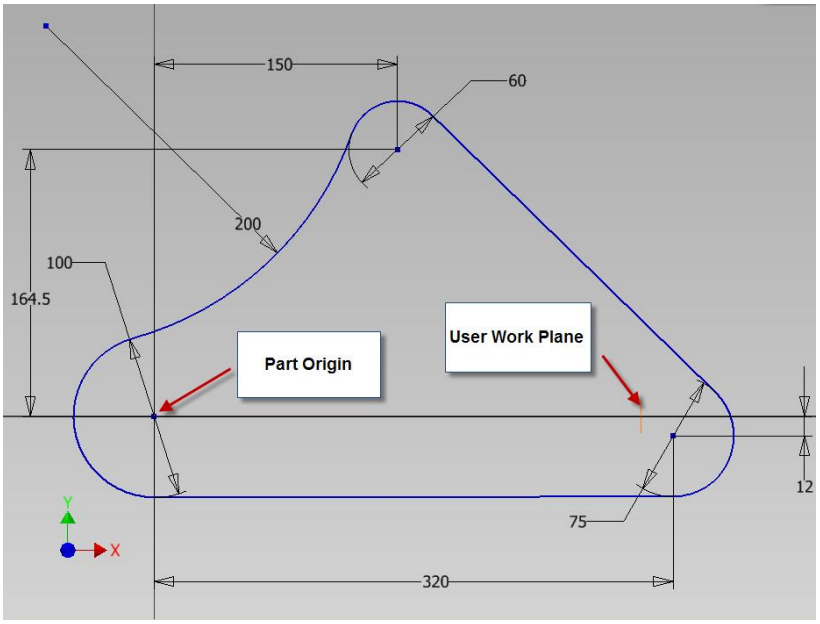
Start a new part and save it as “Chain Path.ipt”. My part is metric and the units are millimeters. Turn on the visibility of the YZ origin plane. Create a user work plane offset from the YZ origin plane by a distance around one chain pitch less than the distance from the first chain sprocket to second. In our case the distance between the first sprocket center and the second is approximately 320 millimeters so we will use an offset distance of 300 millimeters (refer to sketches above for reference).



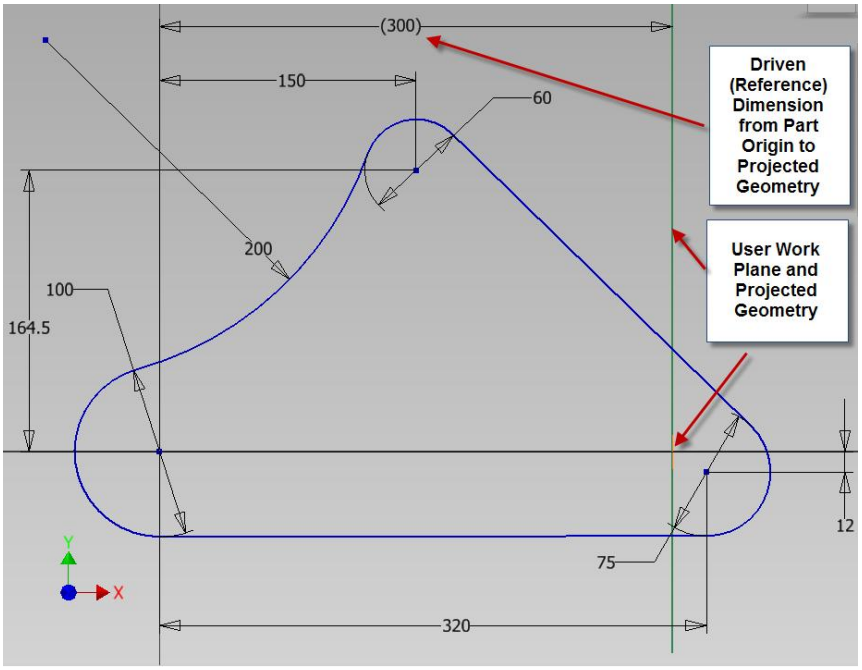
Drag the User Work Plane above the existing sketch in the browser. This step in only necessary if your Inventor Application Options create a new sketch during part creation. This makes the User Work Plane available to the sketch.

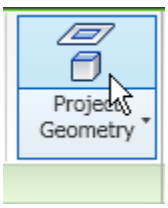


Edit the existing sketch or create a new one if needed and sketch and constrain the chain path sketch. One sprocket center should be constrained to the origin of the part. My sketch plane is the XY plane. It is recommended you fully constrain the sketch. Do not constrain any part of the chain path to the User Work Plane. The top sprocket center dimensions can be adjusted later to adjust the pitch line distance of the chain for a whole number of links.

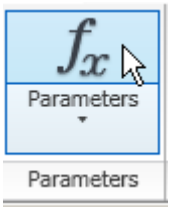


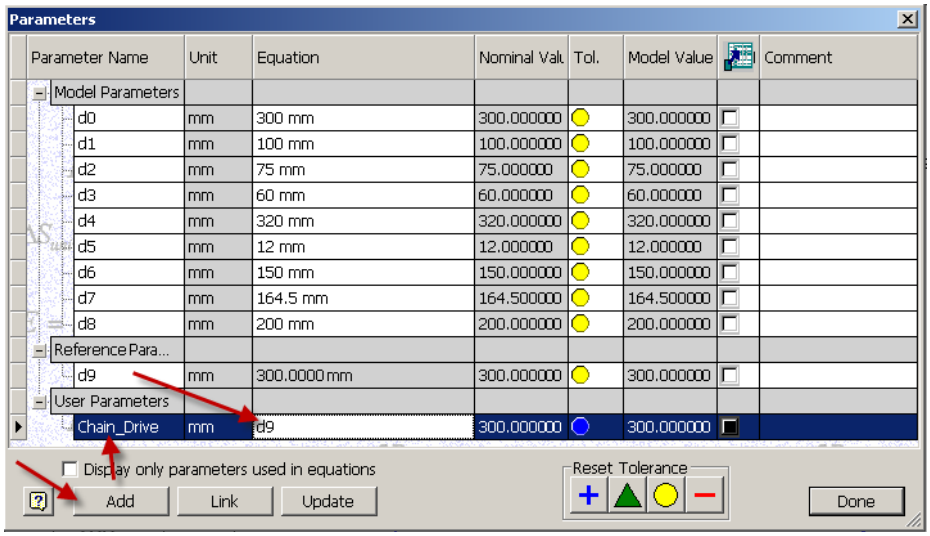
The next step is to project the User Work Plane into the sketch using the Project Geometry command. Add a driven (Reference) dimension between the part origin and the projected geometry.



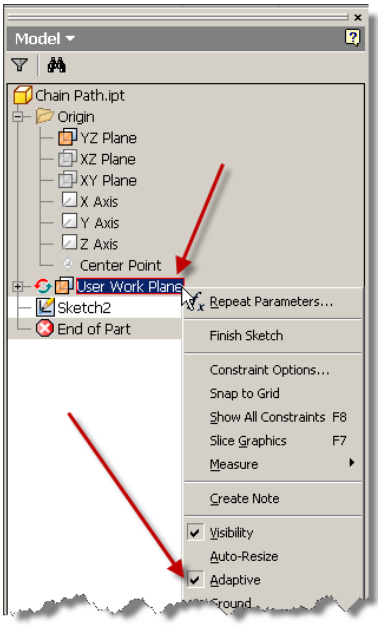


Click on the Parameters command and add a User Parameter called “Chain_Drive”. Assign this parameter to the Reference parameter that you just created, in my case it is “d9”.

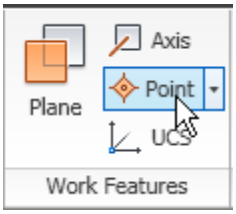


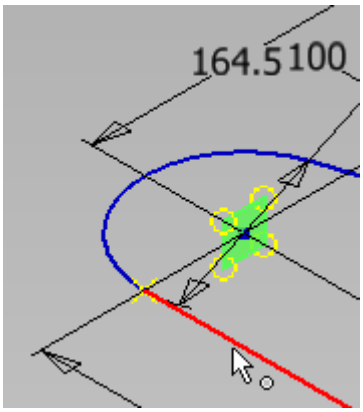


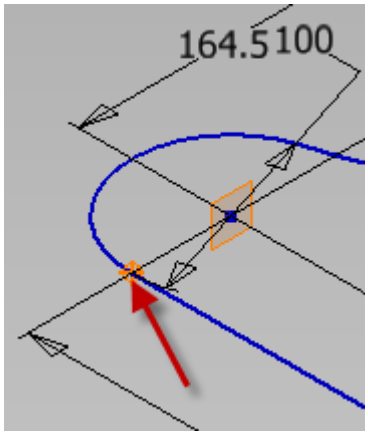
The next step is to make the User Work Plane adaptive. Right click on the User Work Plane in the browser and select “Adaptive”. Finish the sketch.



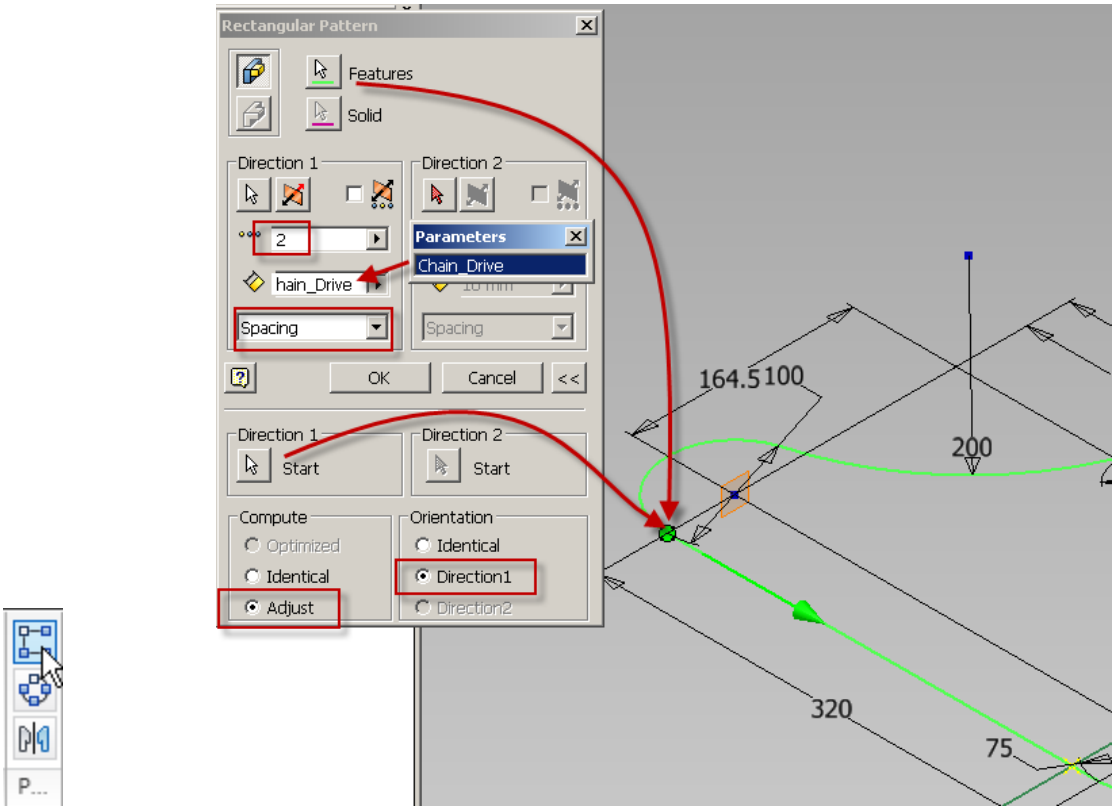
We need to add a work point to our chain path that will be parametrically linked to our User Work Plane. Place a User Work Point at the intersection of the YZ origin plane and the lower line of the chain path.

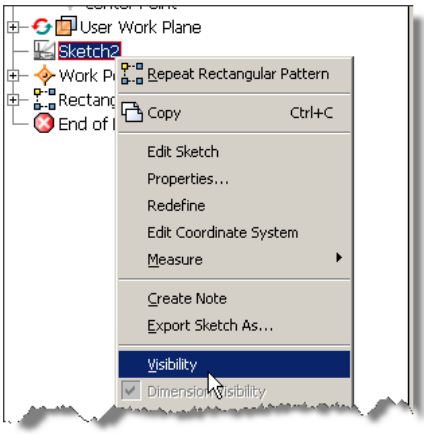






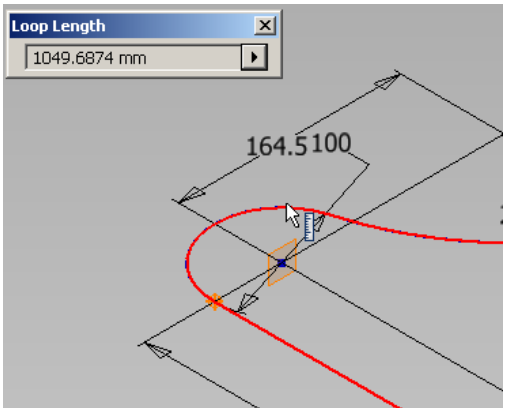
Use the Rectangular Pattern command to pattern this work point using the chain path as "Direction 1". Flip the pattern direction as needed to follow the chain straight path. Fill out the Pattern command dialog box as shown below. The parameter "Chain_Drive" will be used for the pattern spacing. Note "Direction 1" Start is selected and indexed to the work point being patterned. Also not the "Direction 2" has been set to use "Direction 1". The "Adjust" button in the "Compute" section has been selected. After creation of this pattern the chain sketch will become invisible, I suggest you make it visible again from the browser.



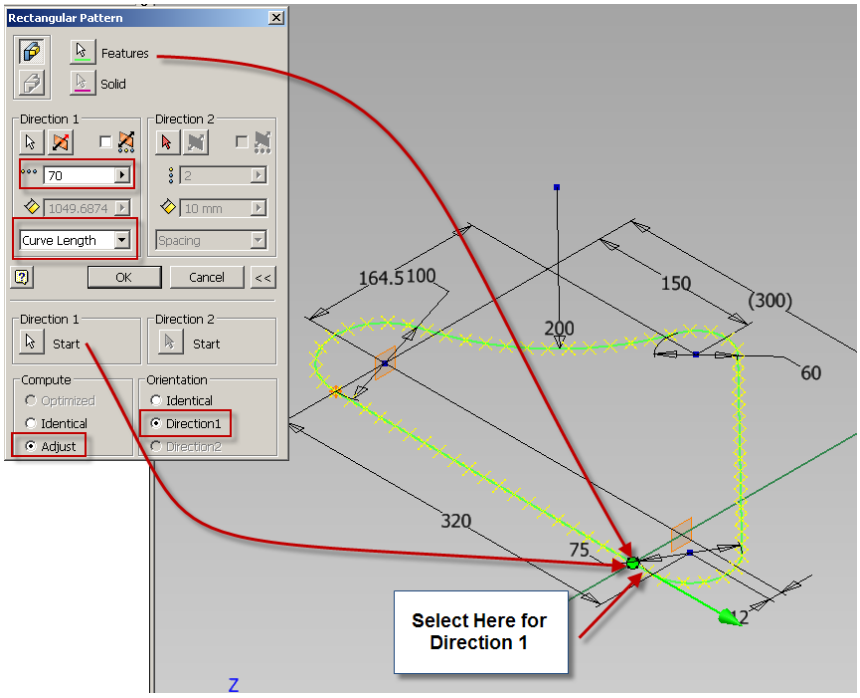


The next step is determine the number of links in your chain and adjust your sketch dimensions to come close (exact is not required) to a whole number of links. We can use the "Measure" command with the loop option to find out the length of our chain path. Divide this by the pitch length (15mm in my example) of your chain

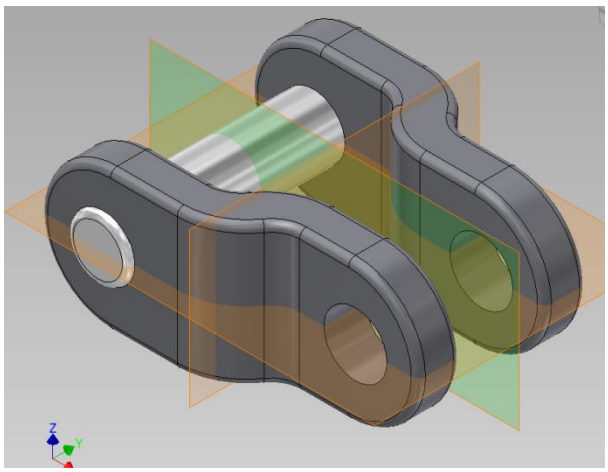
and adjust your sprocket take-up location as needed. In my example the number of chain pitches comes very close to 70 (1049.6874/15=69.97961). That is close enough; no one will see the slight gap between the beginning and ending chain link.



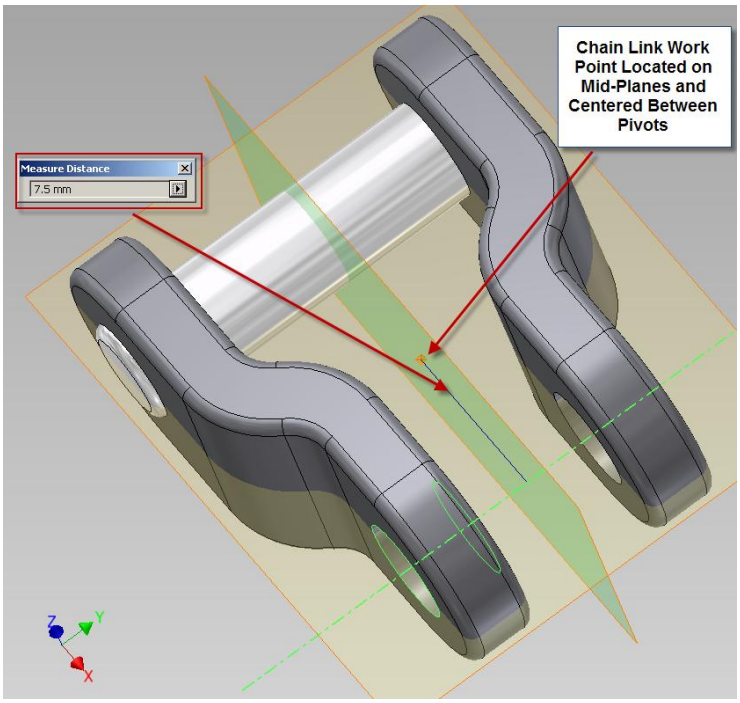
The next step is to make another rectangular pattern using the work point created by the rectangular pattern above as the feature to be patterned. Select the work point from the graphic display to make sure you are selecting a single work point. Select the "Direction 1" on the right side (continuing along the chain path) and flip the direction if required to point along the chain path. Enter the values as shown below in the Rectangular Pattern dialog box. The main difference this time will be the pattern count will be the number of chain pitches calculated above and the "Curve Length" will be used for the spacing to get an equally spaced pattern. Note the selection of the "Direction 1" Start and the settings for the "Compute" and "Direction 2" dialog locations. NOTE: After creation of the pattern the pattern work points will NOT be visible. You will see their location before you say "OK" to the pattern dialog box and that is all you will need. You will be selecting them from the browser for future operations. If you want to see them, the rectangular pattern must be modified under the "Compute" section. Change the "Adjust" to "Identical". If you do this, change it back before proceeding.



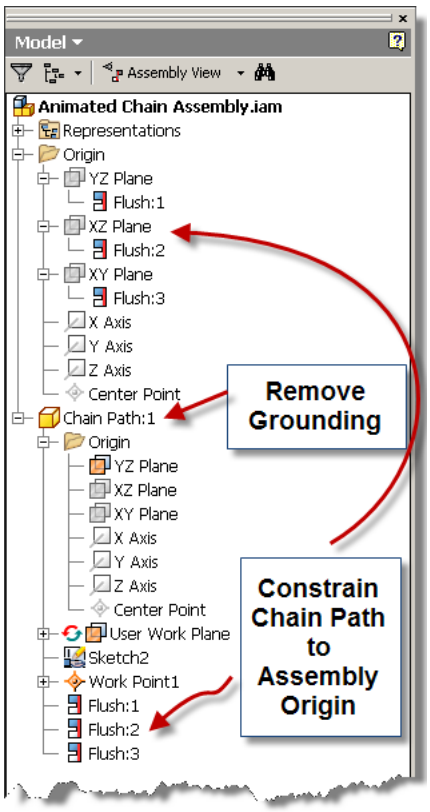
You are finished with your Chain Path.ipt, save it to be used in your Animated Chain assembly model. The next phase is to model your chain link or secure it from the Internet. The main thing to remember when modeling this link is to be sure it is oriented with the part origin planes as shown below.



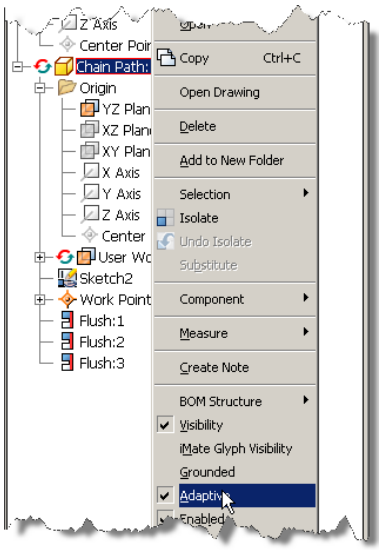
A work point is required in the chain link part located on the mid-planes on the link and centered between pivot axes. In my example, the part was modeled centered on the origin so the work point is the center of the part. Save the part and call it Chain Link.ipt.



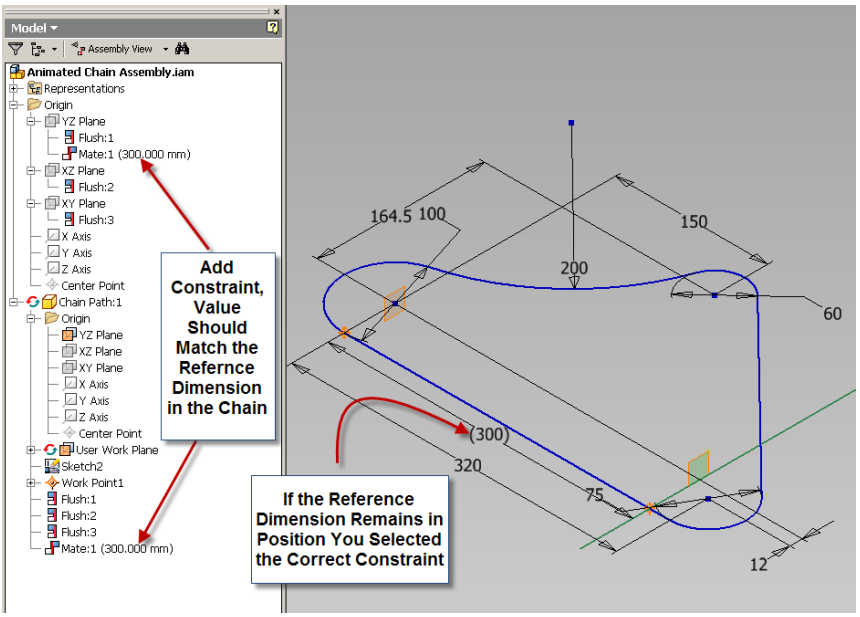
The next step is to create a new assembly, save and name it Animated Chain Assembly.iam. Place the Chain Path.ipt in the assembly and un-ground it. It needs to be constrained in the assembly so match up the origin planes of the assembly and the origin planes of the Chain Path part using three flush constraints.



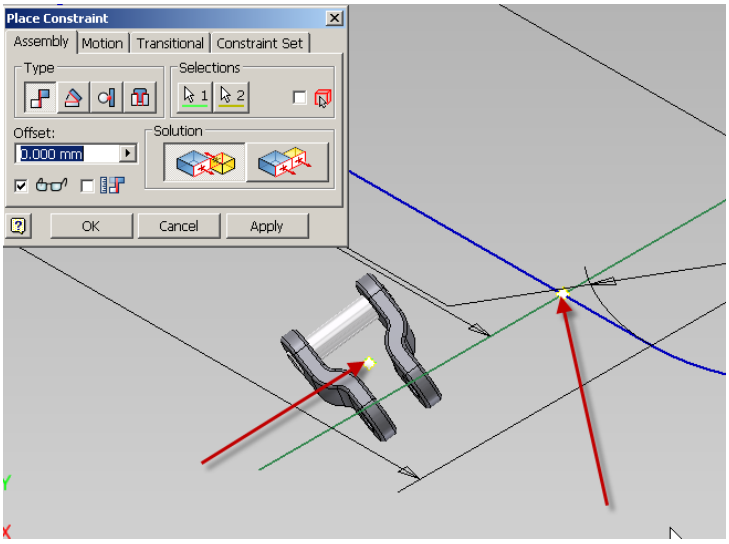
Next make the Chain Path part adaptive in the assembly.



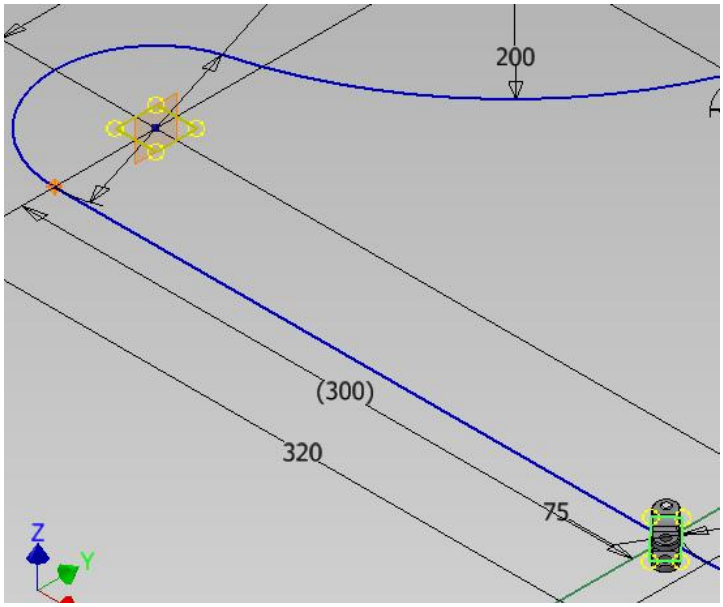
After making the part adaptive, add another constraint to the assembly between the assembly YZ origin plane and the adaptive work plane in the Chain Path part. This constraint will probably be a Mate if you followed my procedure and the offset value should match the reference dimension from the Chain Path part which was 300 millimeters in our example. Open the Chain Path part after placement and make sure the visible work points are positioned along the chain path; if not, the Mate constraint should be changed to a Flush constraint.



Place the "Chain Link" part into the assembly. You first place a Mate constraint between the work point of the Chain Link and the second work point on the Chain Path Part.

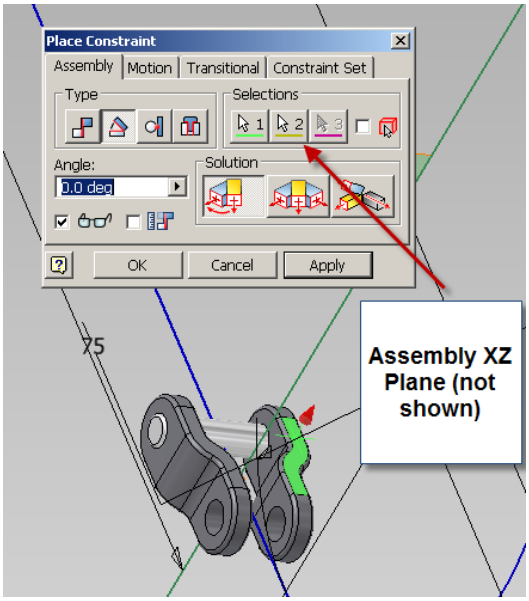


Add a Mate constraint between the Chain Link XZ origin plane and the assembly XY origin plane. Your planes may vary but the objective is to line up the chain link with the chain path as shown below.

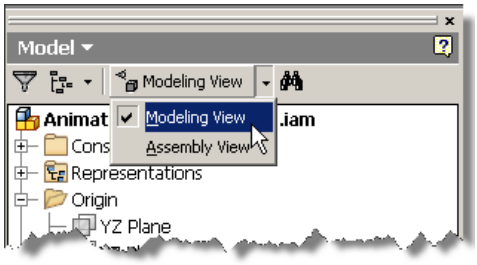


The last constraint will be an angle constraint between on face of the Chain Link part and the assembly XZ origin plane. Again your constraint features may vary but

the goal is to align the part to the Chain Path part. The Chain Link should be fully constrained and aligned with the Chain Path part.

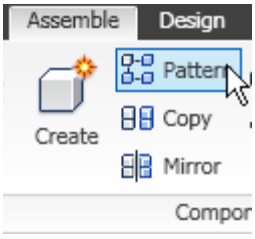


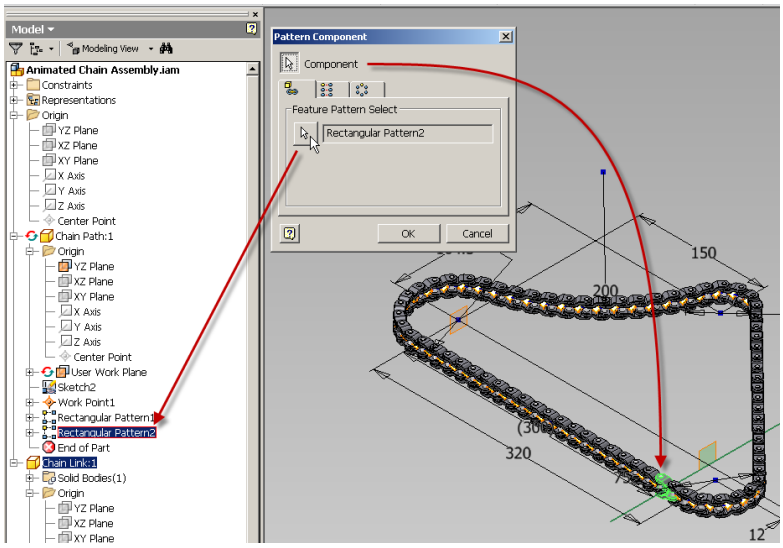
Go to the top of the Browser in the assembly and change from "Assembly View" to "Modeling View".



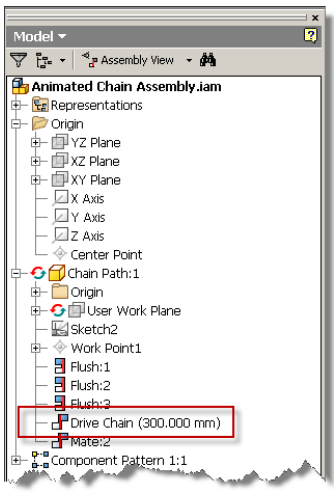
Select the "Pattern" command and select the Chain Link as the component to pattern. Select the "Associative Feature Pattern" button and in the browser pick the

second rectangular pattern in the Chain Path part. Pick "OK" to create the pattern of chain links.

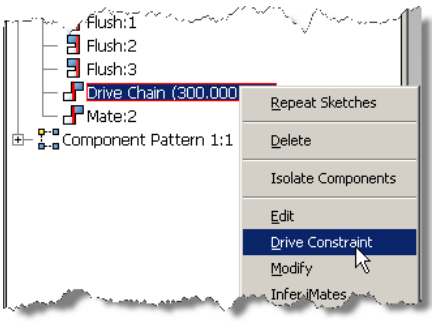




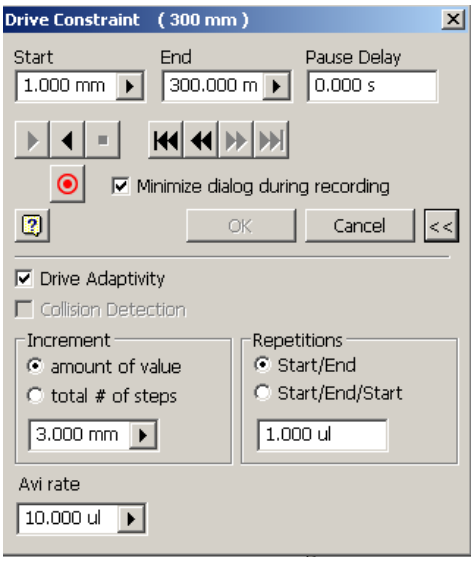
At this point you will probably like to do some clean-up by turning off all visible work features, 2D sketches and switch back to Assembly View in the browser. It is a good idea to rename the chain driving constraint to make it easier for others to find it. I rename mine to "Drive Chain" in this example.



Right click on the driving constraint and choose "Drive Constraint" from the menu.



In the "Drive Constraint" dialog box, change the Start value to 1 mm and the End value to 300. Remember the model is driving the Adaptive Work Plane in the Chain Path part from the beginning location (300 millimeters) toward the assembly origin. If you set the end to zero, the model will possibly fall apart because the work point of the chain link falls of the chain path in this model. Expand the dialog box and place a check mark in the "Drive Adaptivity" box. I changed the "Increment" value in this example to speed up the chain motion. Select the "Play" arrow and enjoy.



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12/26/09
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