Computer Aided Design 2

First Robotics 4669
Galileo High School
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Setting Up

Create a project called CADTraining in Inventor

Select the project

Copy the Parts folder from

https://drive.google.com/open?id=1TJT36WsmA4ypVmiz7wZCV1-ilftgkibB
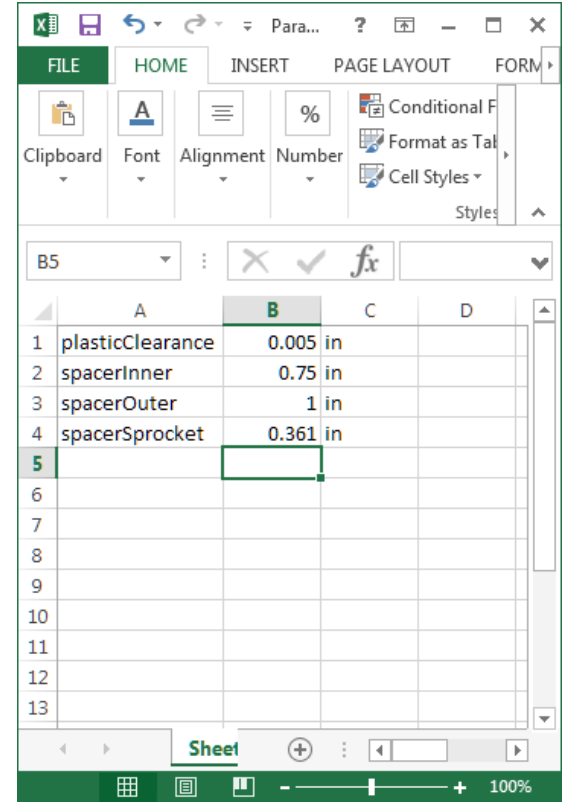
into the project

Setting Up The Parameter Worksheet

Start Microsoft Excel

Select New Blank Workbook

Enter the plasticClearance, spacerInner, spacerOuter, spacerSprocket

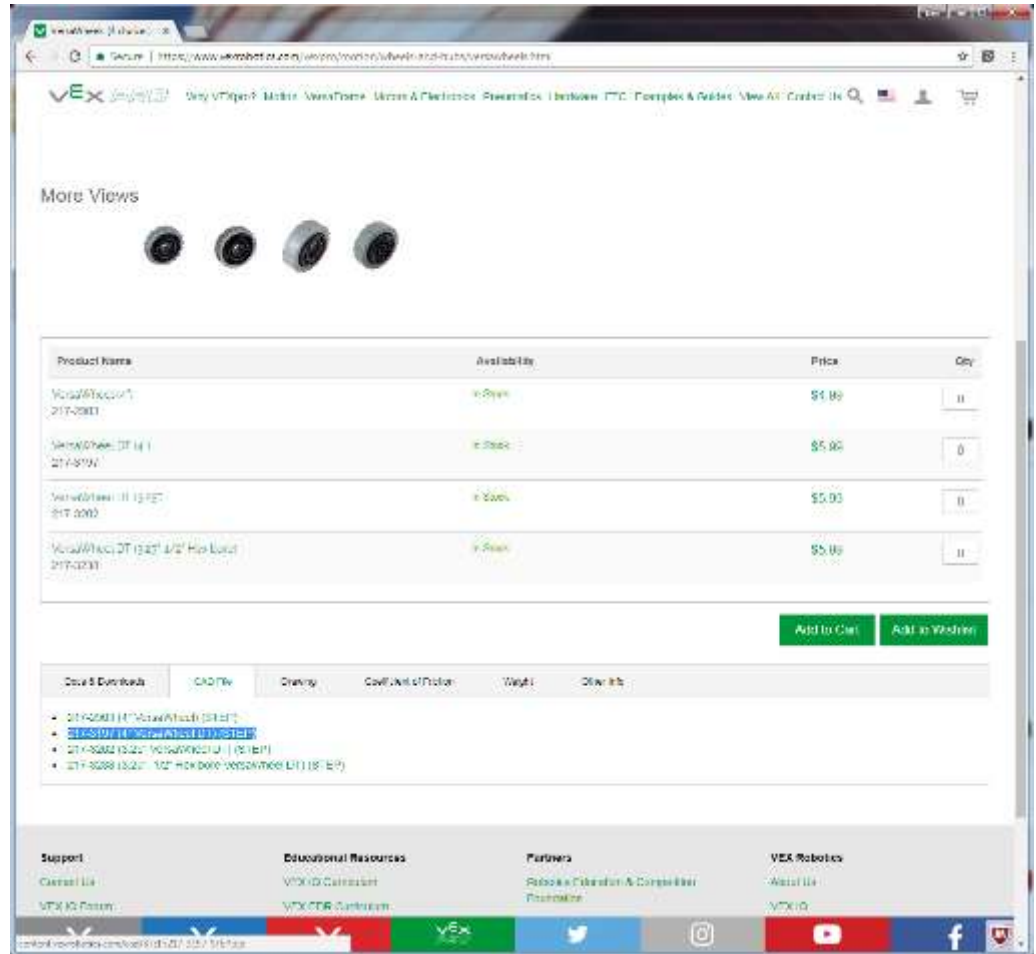Save the worksheet with the name "Parameters" in the Inventor Project directory you just created

Adding A New Part

Find the VersaWheel Part on the vexrobotics.com website

Download the 4" VersaWheel DT STEP file
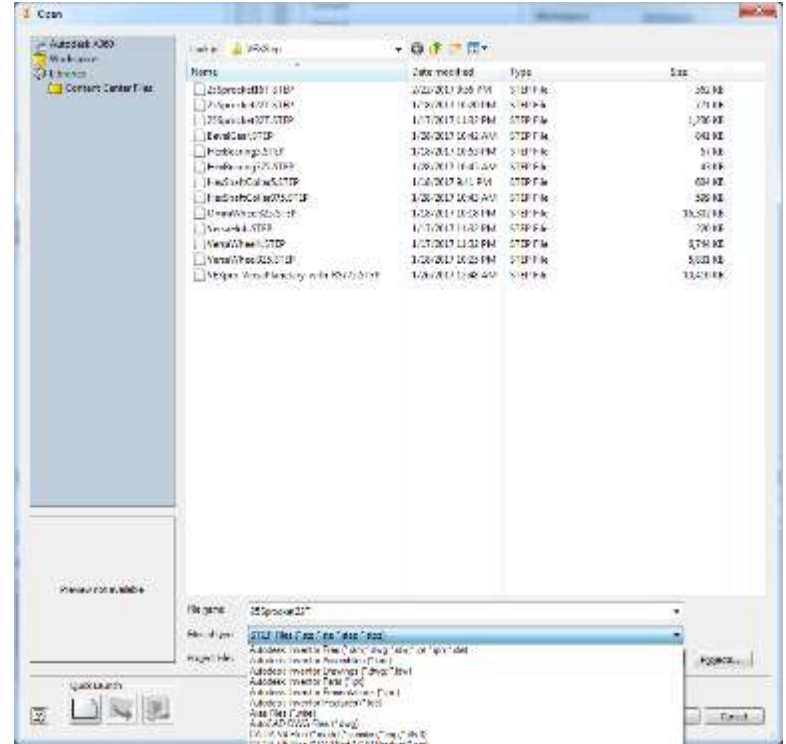
Opening the STEP File

Click on the Open Button in the Toolbar

Select the File type as STEP Files

Select the file you downloaded

Click OK on the Import dialog

Save the imported part as VersaWheel4.ipt in the Parts folder
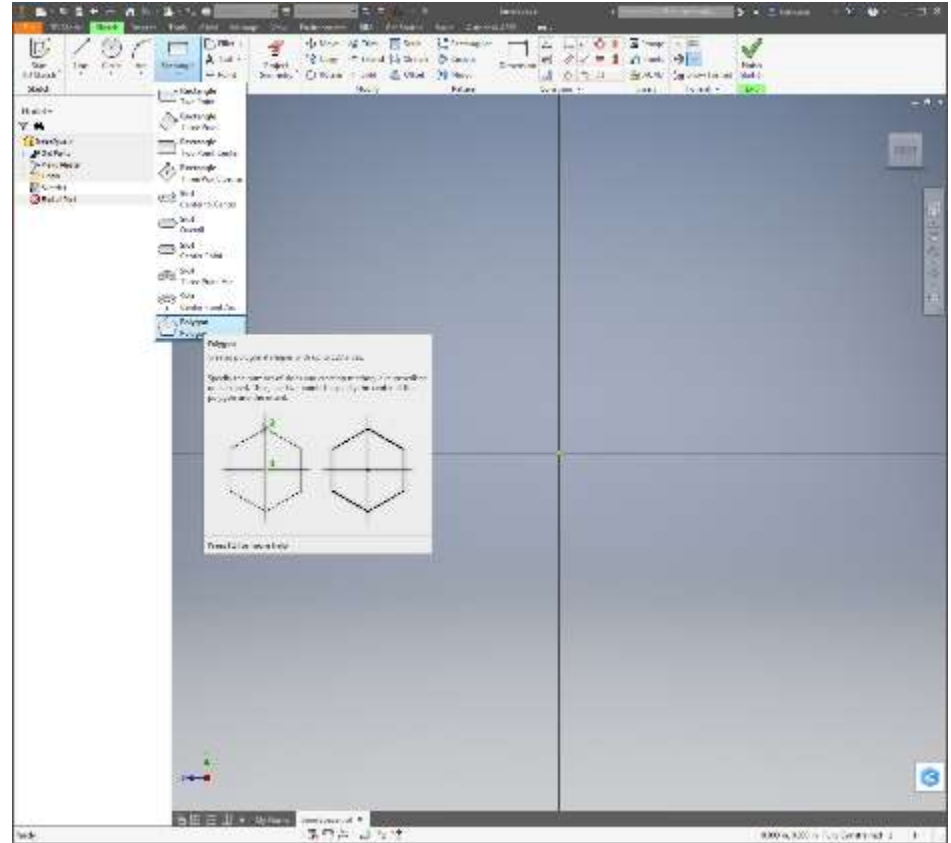
Create A Spacer

Create a new part

Save As InnerSpacer.ipt

Link the Parameters worksheet to the part

Create a new sketch in the YZ plane

Select the Polygon Tool from the Sketch toolbar

Setup the Polygon

Select the second option called circumscribed

Make sure the number of sides is 6
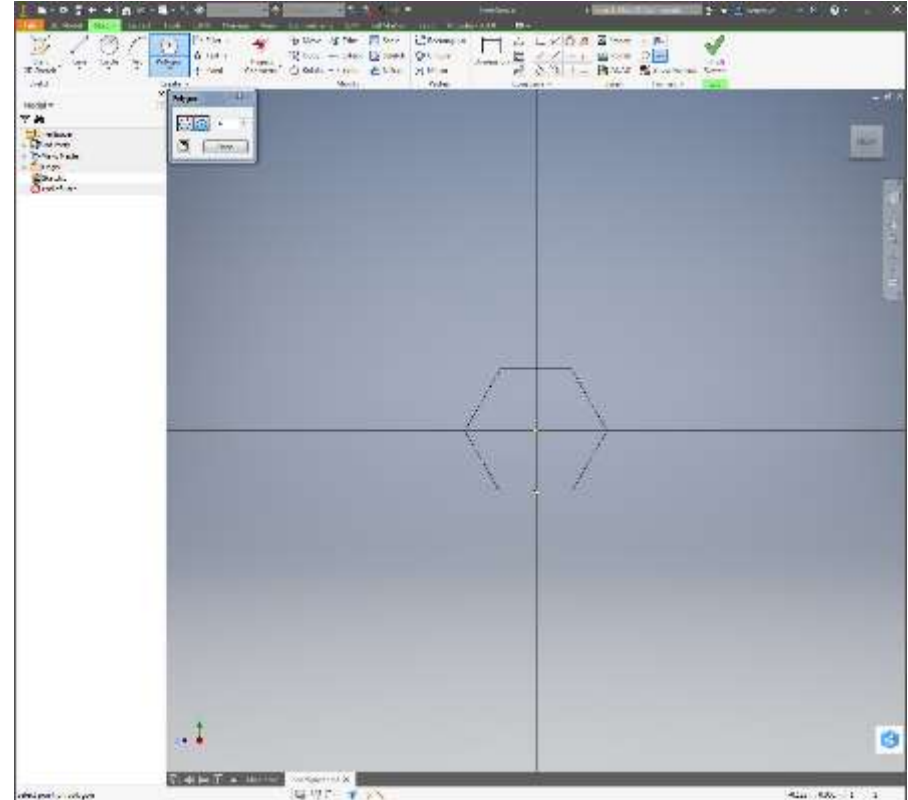
Do not click Done

Draw Hexagon

Click on the yellow dot in the center of the part window

Move mouse straight down and stay on the vertical line

Click to create hexagon
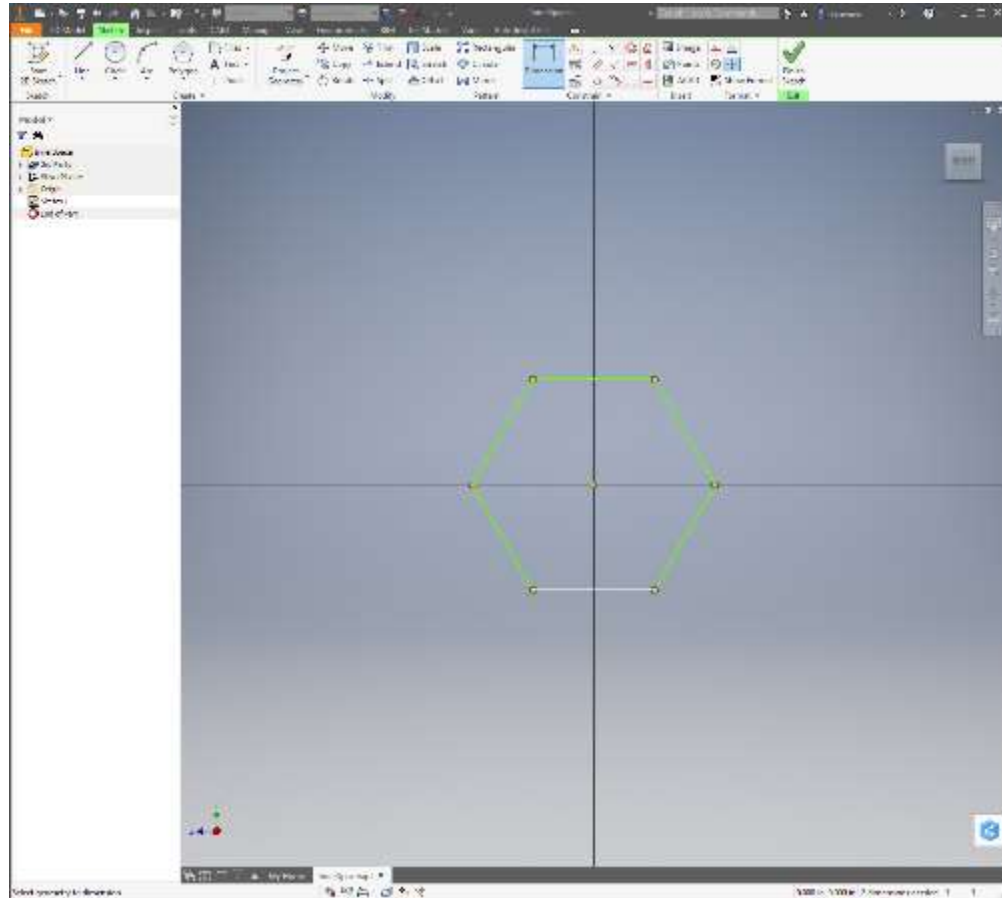
Size The Hexagon

Click on the Dimension button on the Toolbar

Click on the bottom segment of the hexagon

Click on the center of the hexagon

Move mouse down and to the right and Click

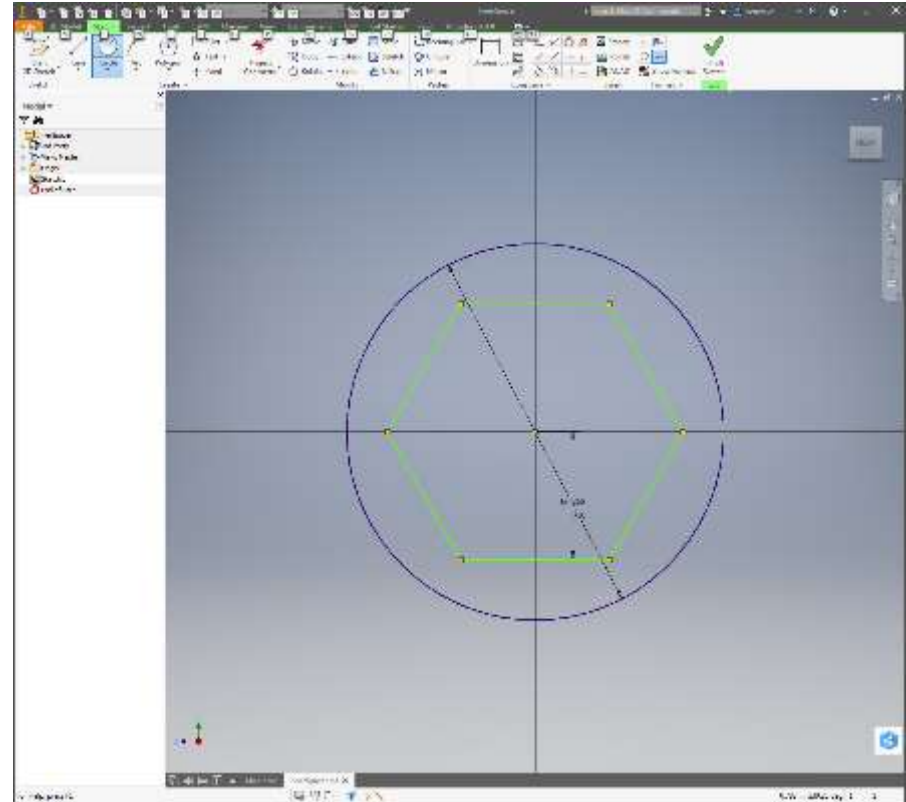Type plasticClearance + .25 and press Enter

Draw Circle

Select the Circle Tool

Click on the center

Move mouse down and to the right. Do not click

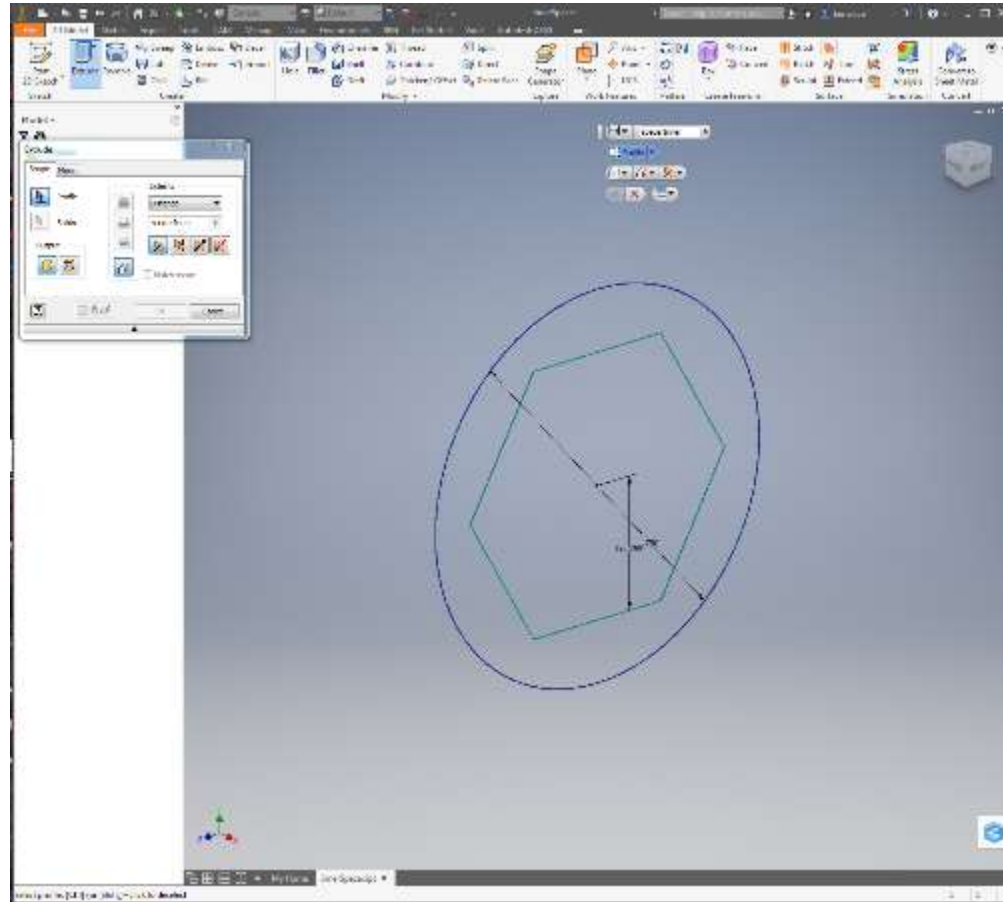Type .75 and press enter

Click on Finish Sketch

Extrude

Click on Extrude

Type spacerInner into the box below the Distance in the dialog

Click on the area between the hexagon and the circle

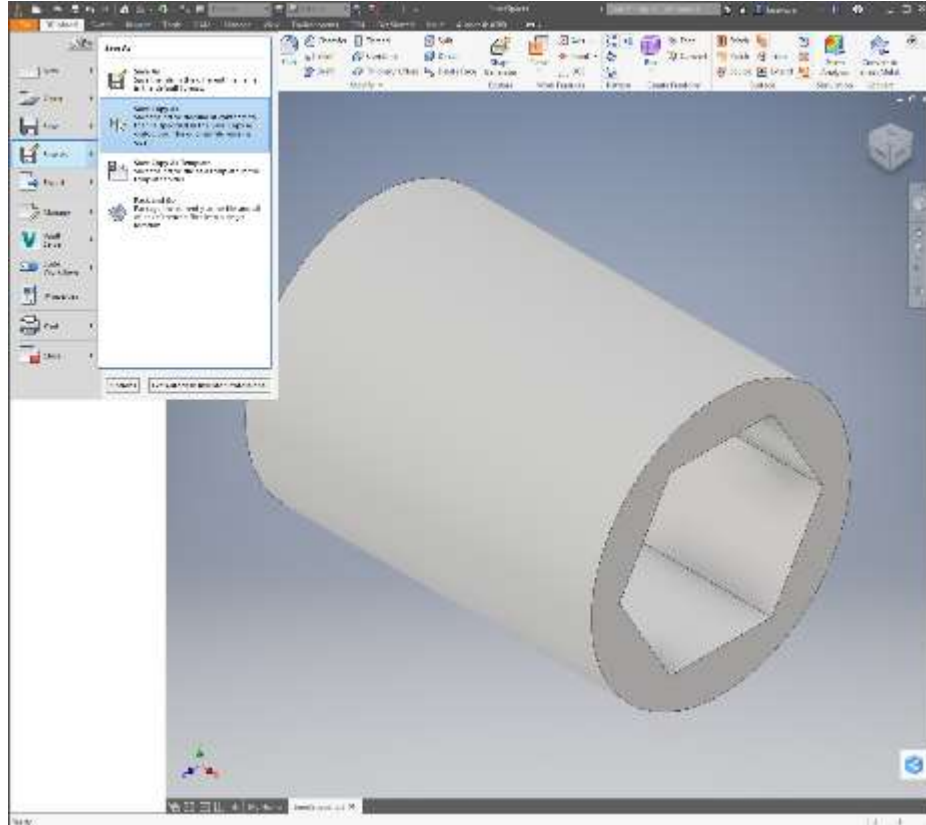Click OK in the dialog

Save the part

Create The Outer Spacer

With the InnerSpacer open, select Save Copy As

Type OuterSpacer as the Filename in the File dialog
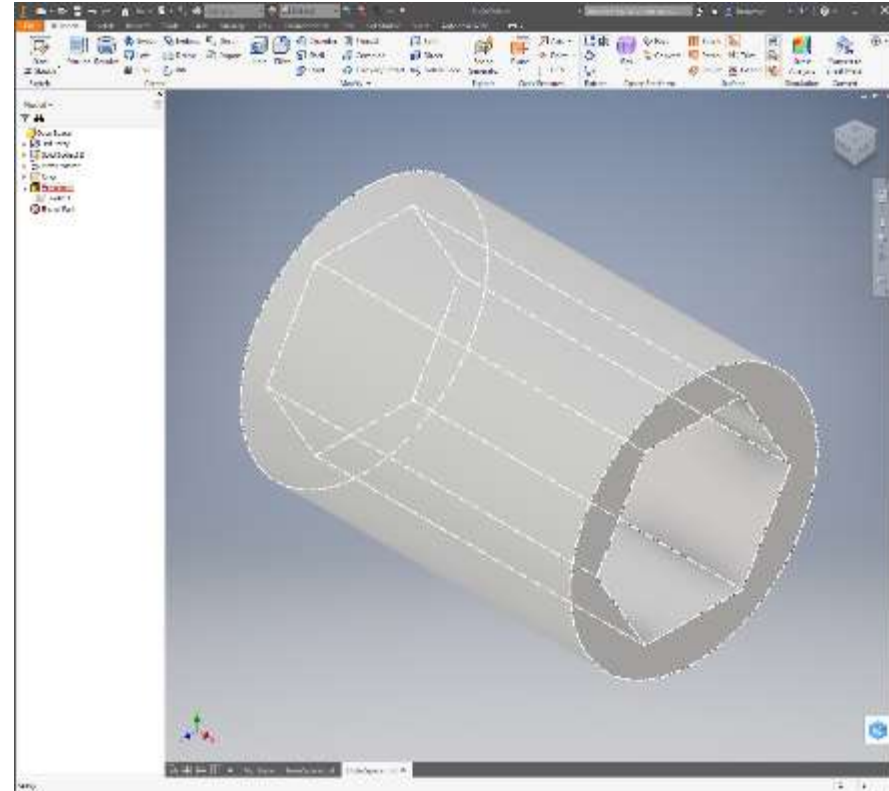
Open the new OuterSpacer.ipt file

Change The OuterSpacer Length

Double click on the yellow cube next to Extrusion1 to bring up the Extrusion dialog

Change the box below the distance to spacerOuter
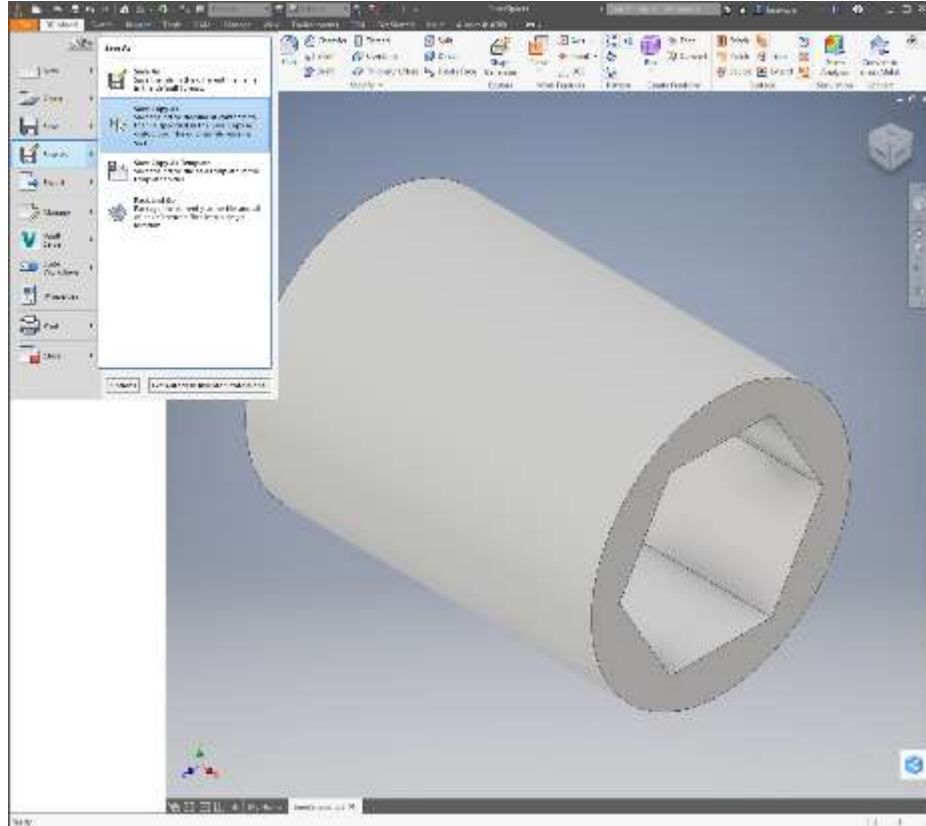
Save the part

Create The SprocketSpacer

With the OuterSpacer open, select Save Copy As

Type SprocketSpacer as the Filename in the File dialog
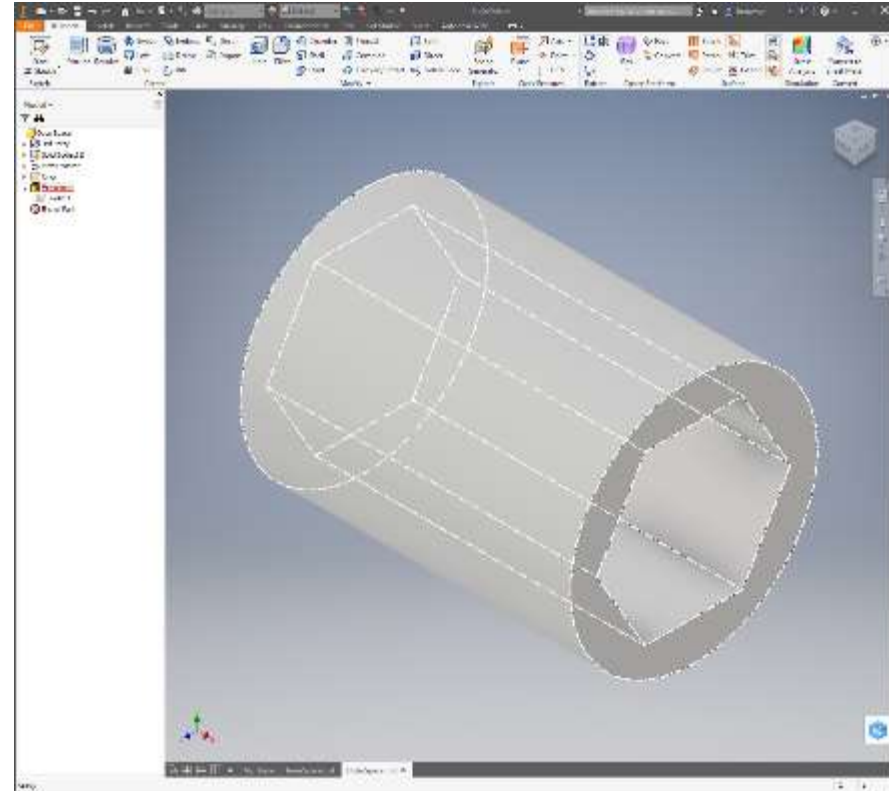
Open the new SprocketSpacer.ipt file

Change The SprocketSpacer Length

Double click on the yellow cube next to Extrusion1 to bring up the Extrusion dialog

Change the box below the distance to spacerSprocket
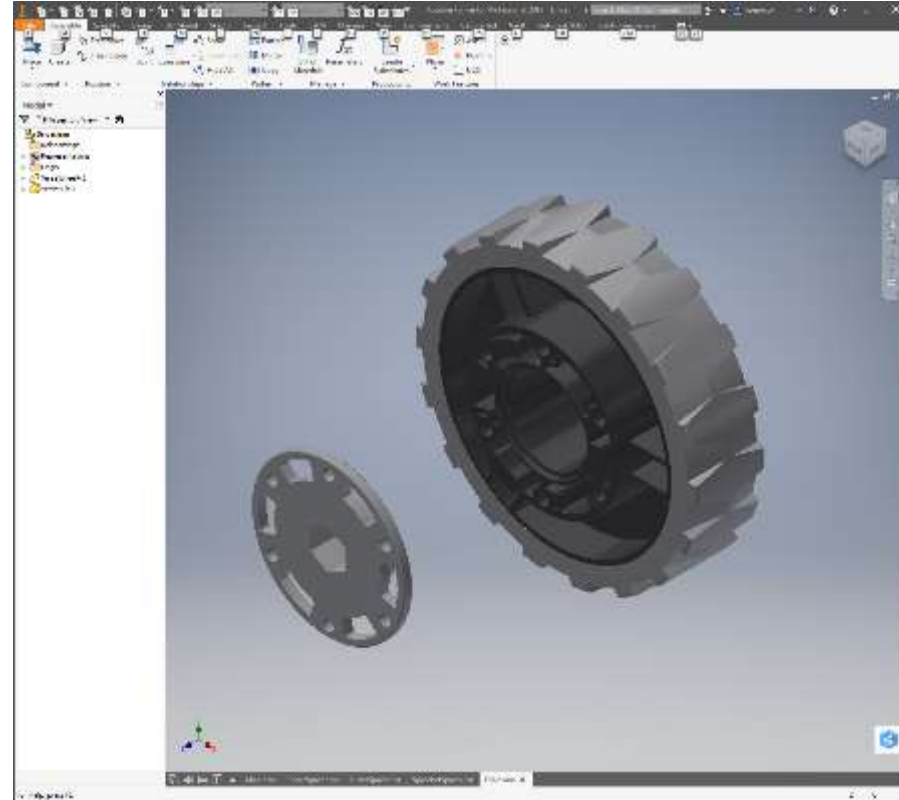
Save the part

Create Drive Assembly

Create a new assembly

Save the assembly as Drive.iam

Place the VersaWheel4.ipt from the Parts folder

Place the VersaHub as shown using the right click options to rotate the part
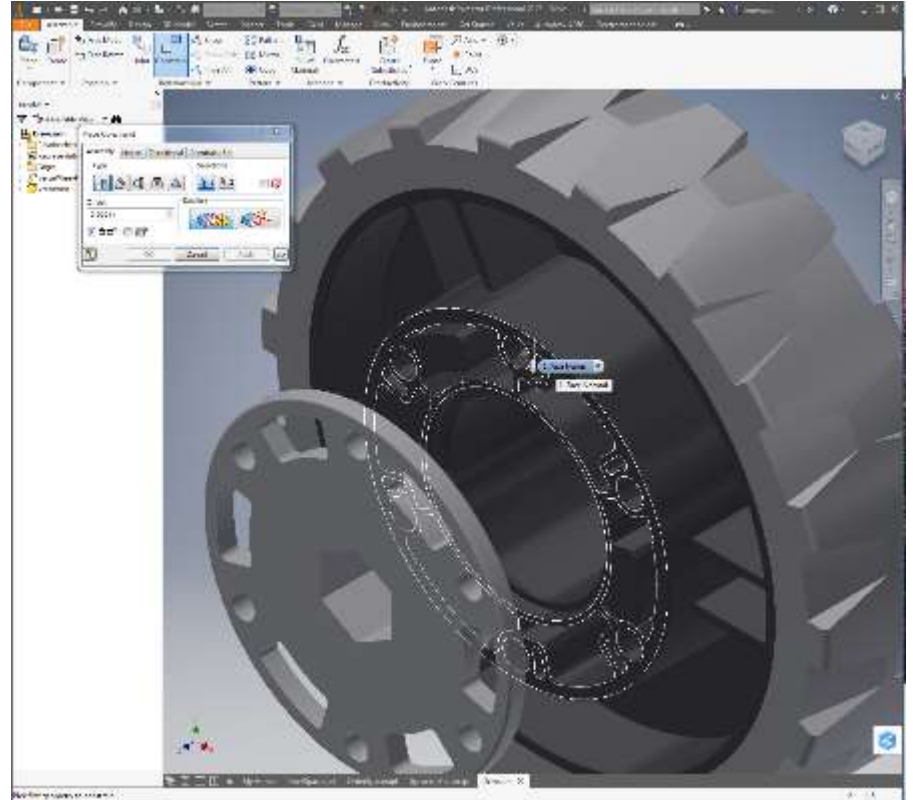
Constrain The Surfaces

Click on the Constrain button

Make sure the Mate option is selected

Click on the face of the VersaWheel
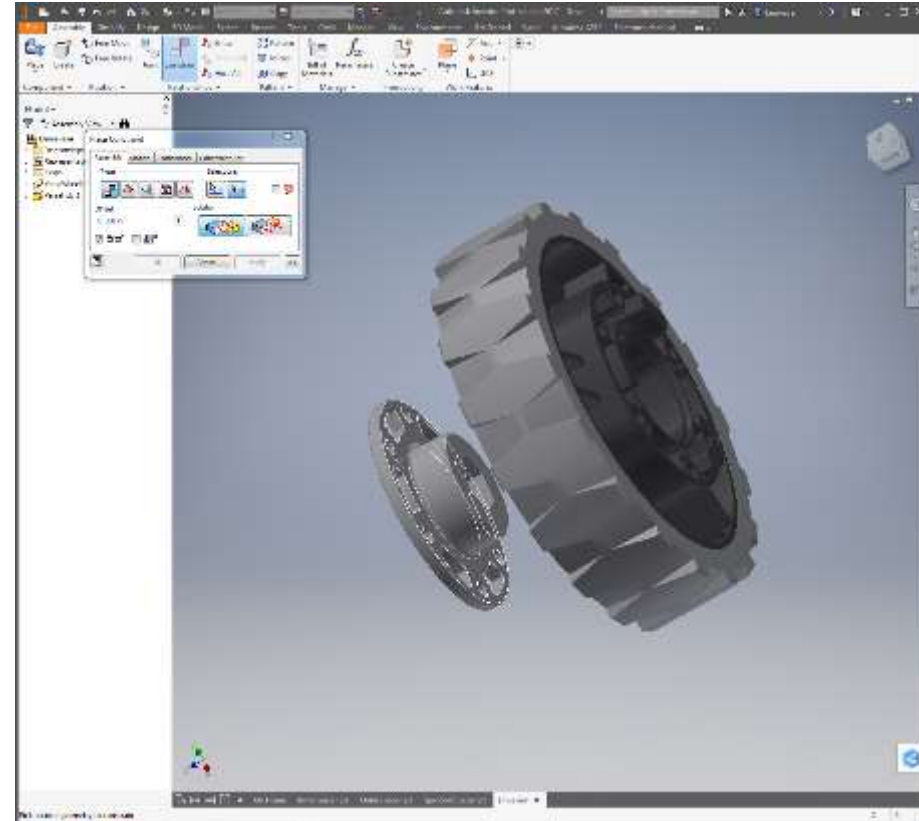shown in the picture not the small tabs
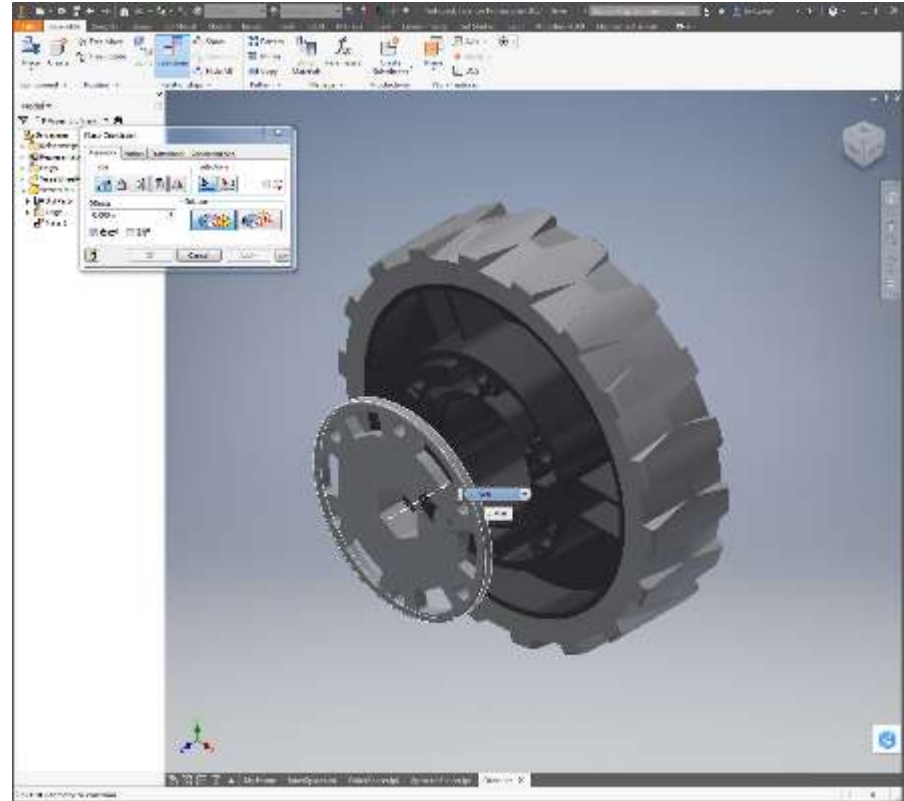
Constrain The Surfaces

Rotate the parts

Click on the face of the VersaHub shown in the picture
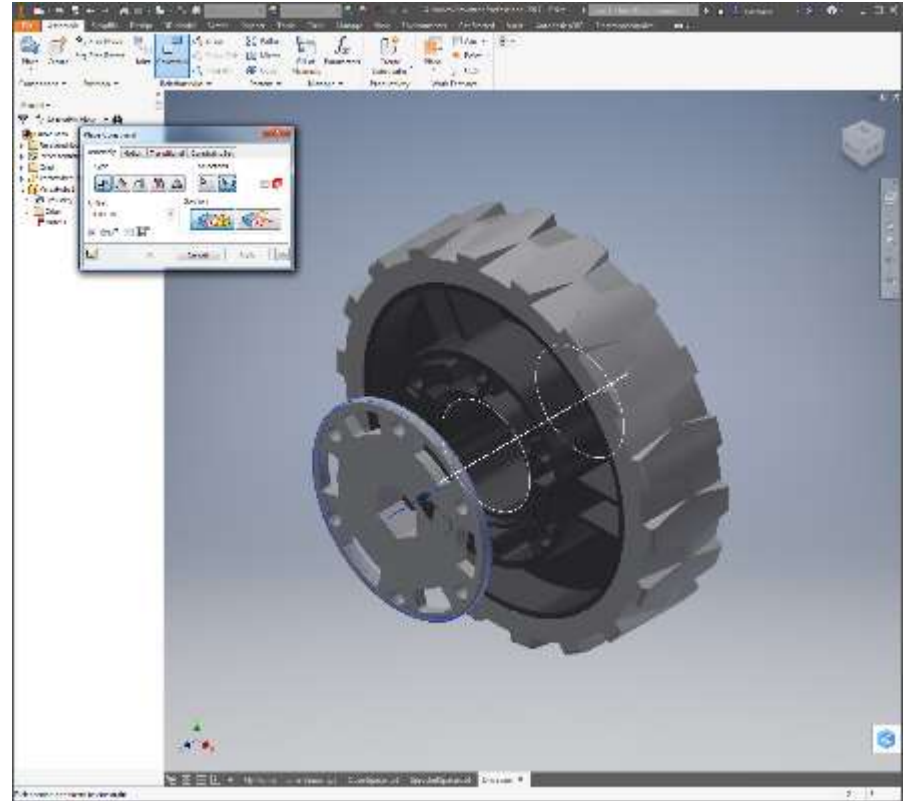
Click Apply

Constrain the Axis

Click on the outside cylindrical surface
of the VersaHub when a line appears
through the center axis

Constrain the Axis

Click on the inside cylindrical surface of the VersaWheel when a line appears through the center axis
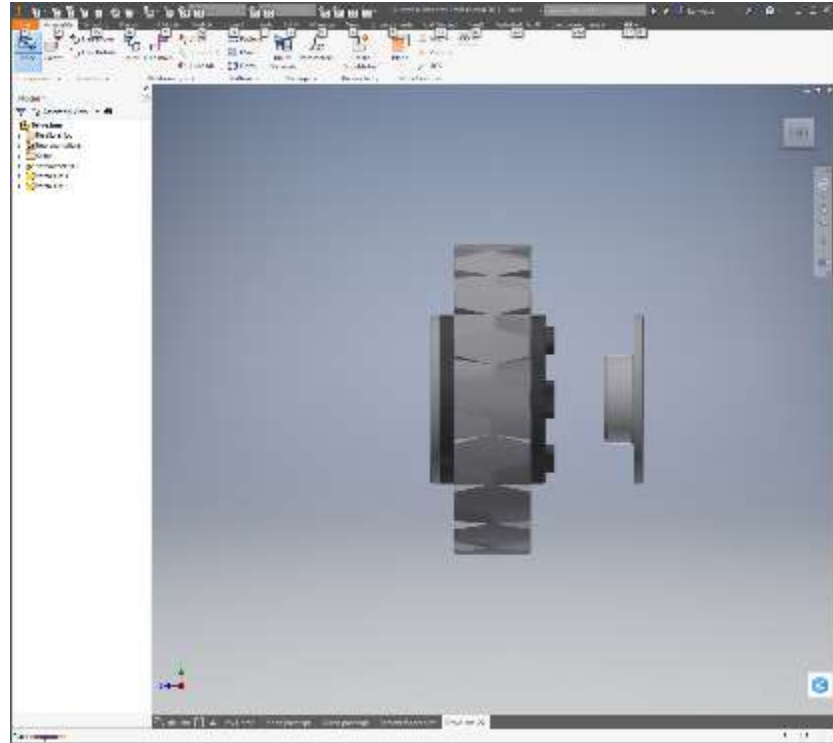
Click OK

Add VersaHub To Other Side

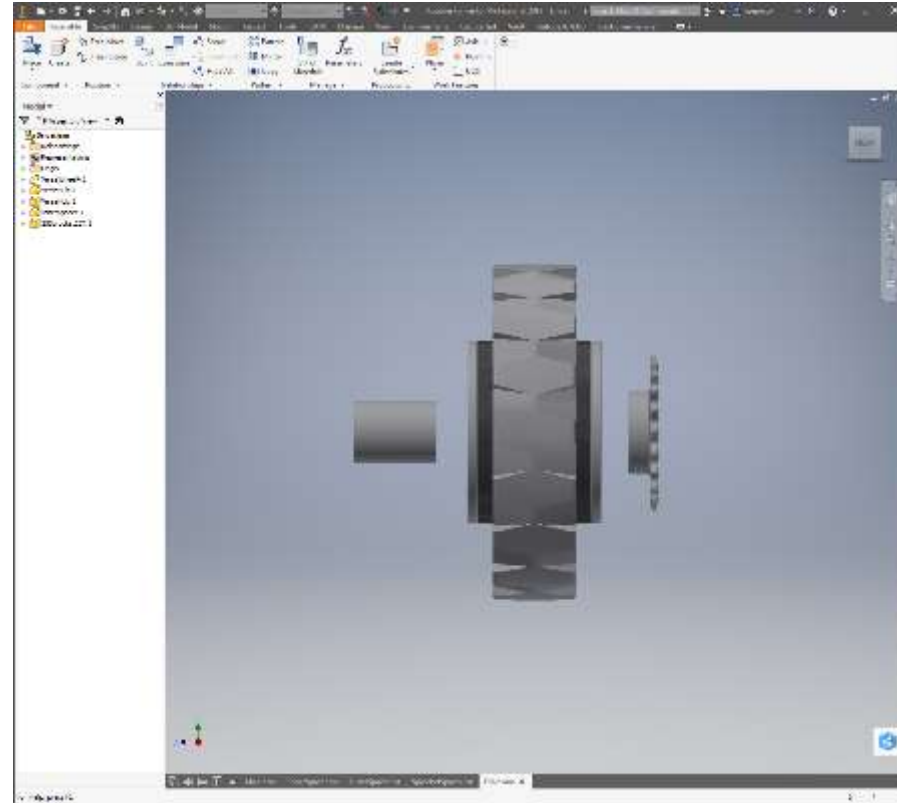Place a VersaHub on the other side of the VersaWheel

Constrain the VersaHub using the same axis and surface constraints
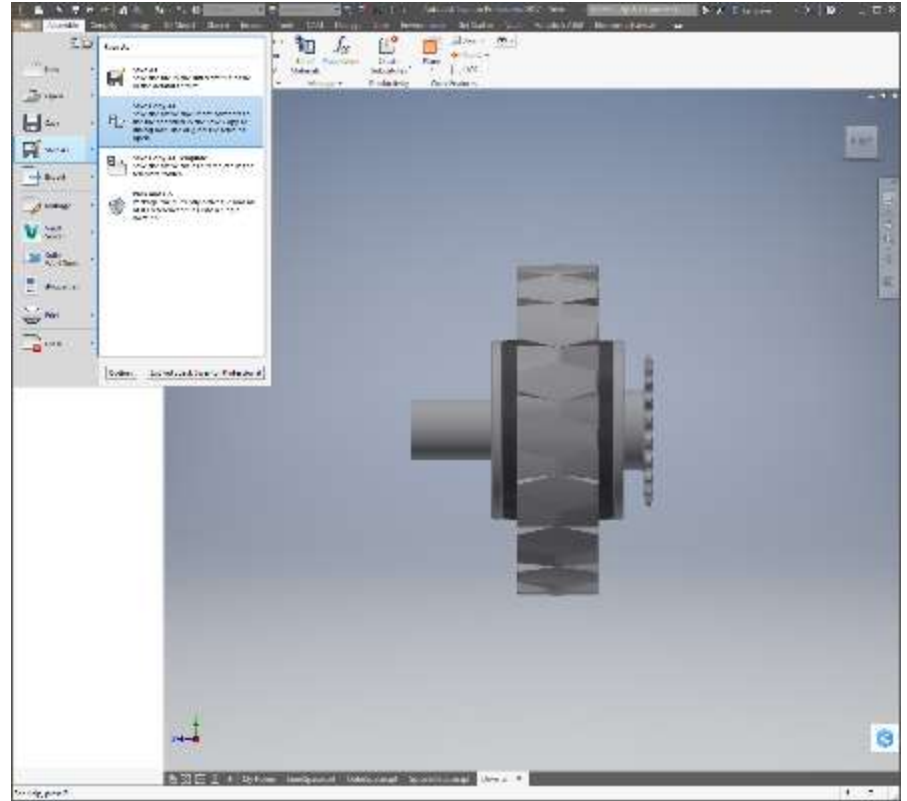
Add InnerSpacer and Sprocket

Place an InnerSpacer.ipt and a
25Sprocket22T.ipt

Use axis and surface constraints
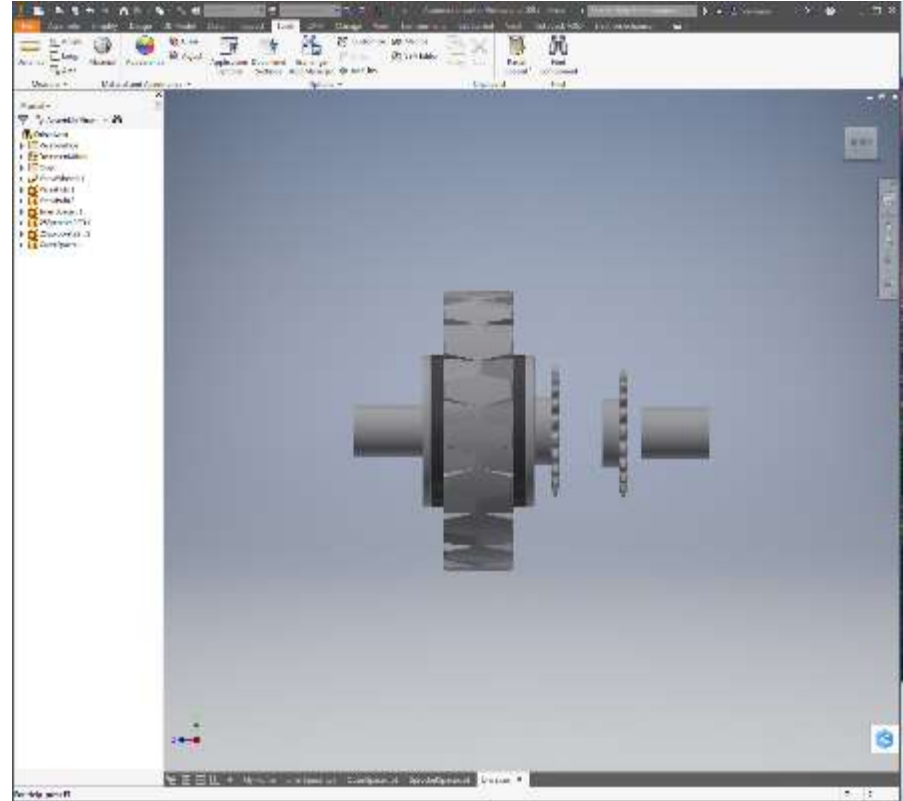to put the parts together

Save A Copy As FrontDrive.iam

Save A Copy As FrontDrive.iam

Add OuterSpacer and Sprocket

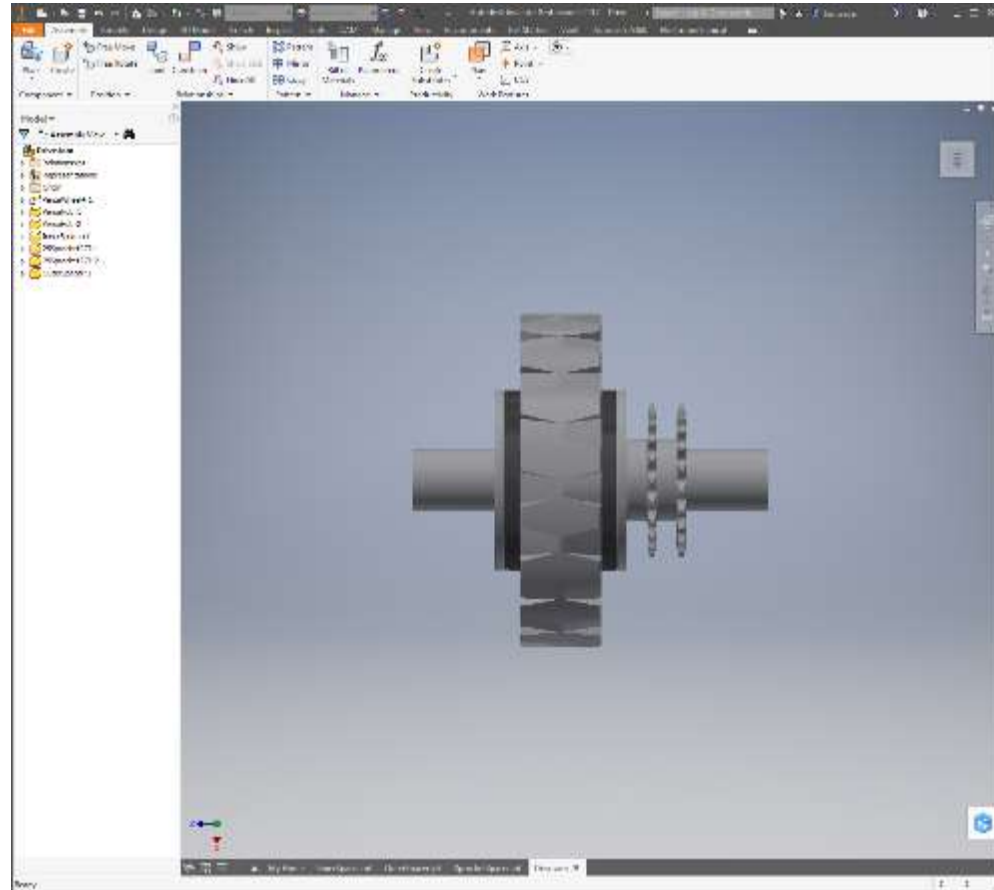Place an OuterSpacer.ipt and a
25Sprocket22T.ipt

Use axis and surface constraints
to put the parts together

Save Drive.iam
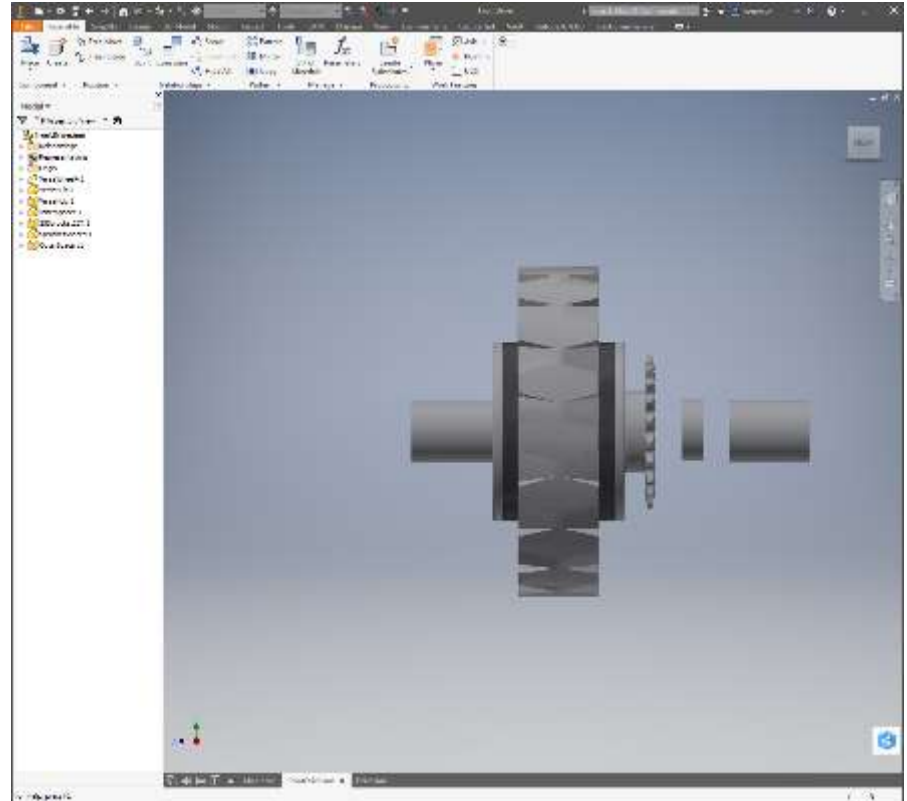
Save Drive.iam

Open FrontDrive.iam

Add SprocketSpacer And OuterSpacer to FrontDrive

Place an OuterSpacer.ipt and a
25Sprocket22T.ipt

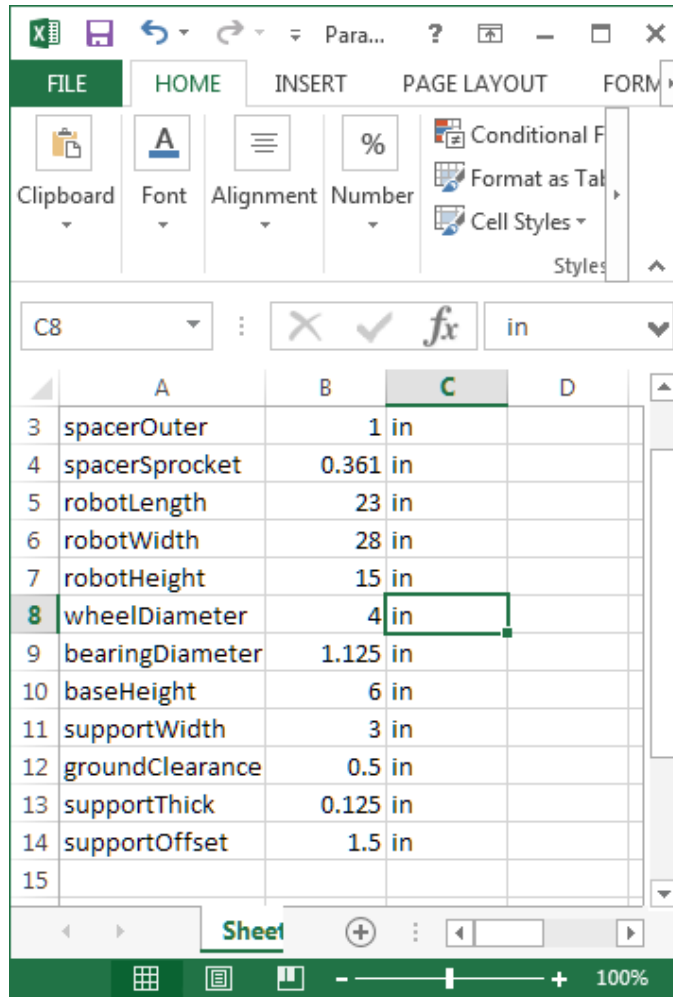Use axis and surface constraints to
put the parts together

Save FrontDrive.iam

Add Parameters

Add Parameters

Save Worksheet

Create Frame Width Bars

Open 20x20Rail.ipt from the Parts folder

Save A Copy as Width.ipt

Close 20x20Rail and open Width.ipt

Link the Parameters worksheet into the Width.ipt part

Edit the Length parameter equation to be robotWidth-40mm

Click Done and save part
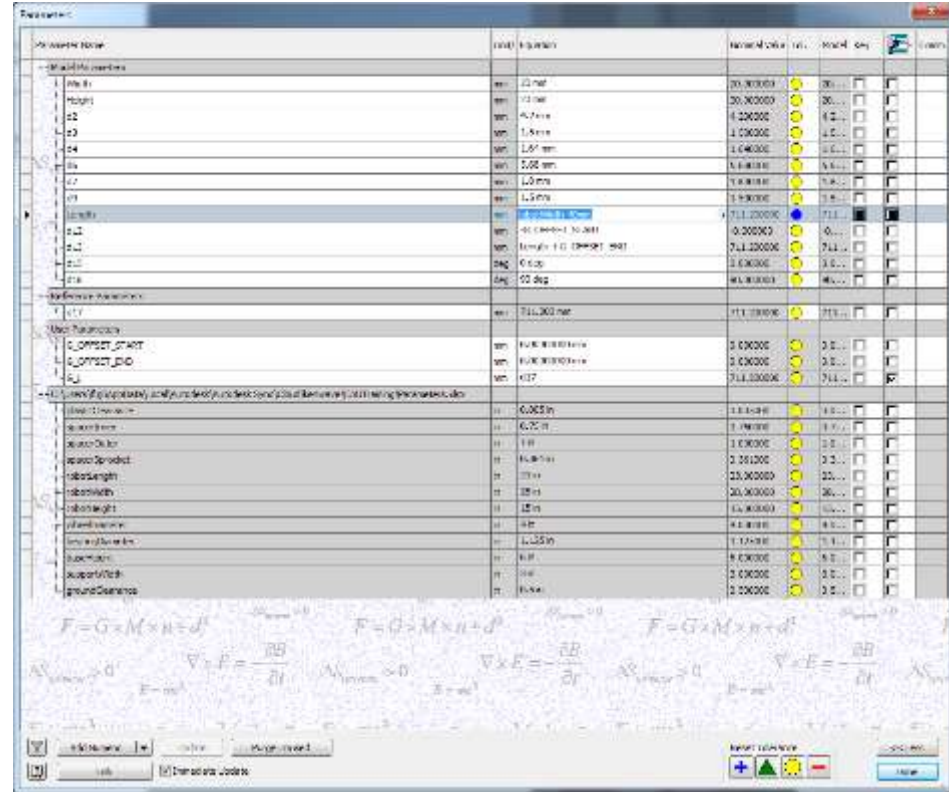
Create Frame Length Bars

With the Width.ipt part open, Save A Copy as Length.ipt

Close Width.ipt and open Length.ipt

Edit the Length parameter equation to be robotLength-40mm

Click Done

Save part

Create Frame Height Bars

With the Length.ipt part open, Save A Copy as Height.ipt
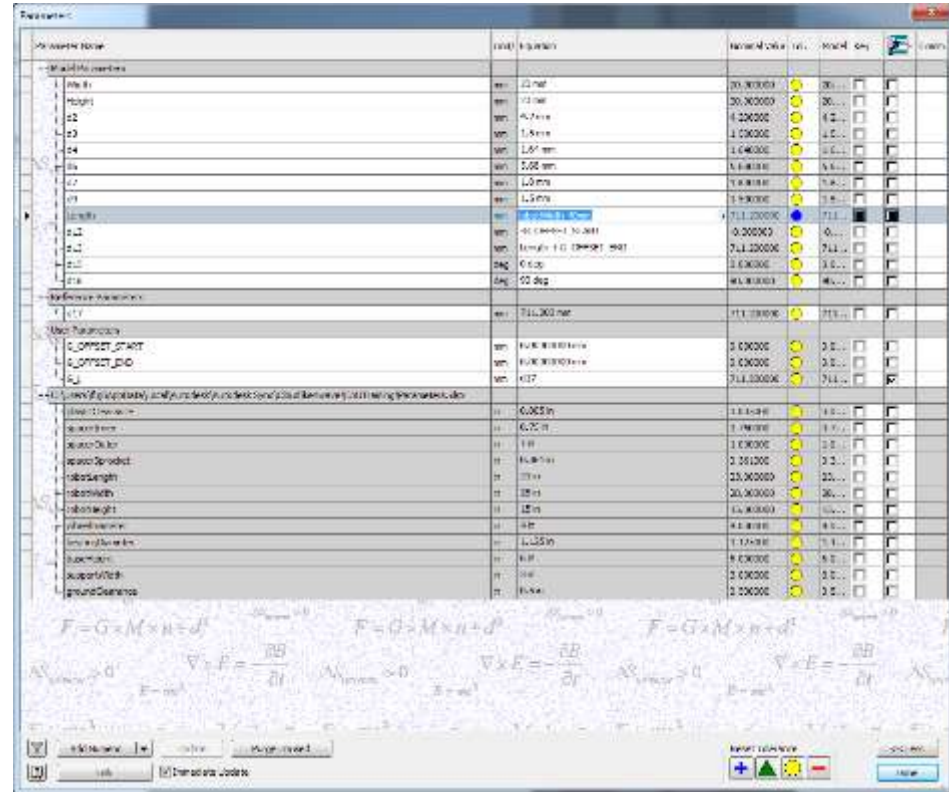
Close Length.ipt and open Height.ipt

Edit the Length parameter equation to be robotHeight-groundClearance

Click Done

Save part
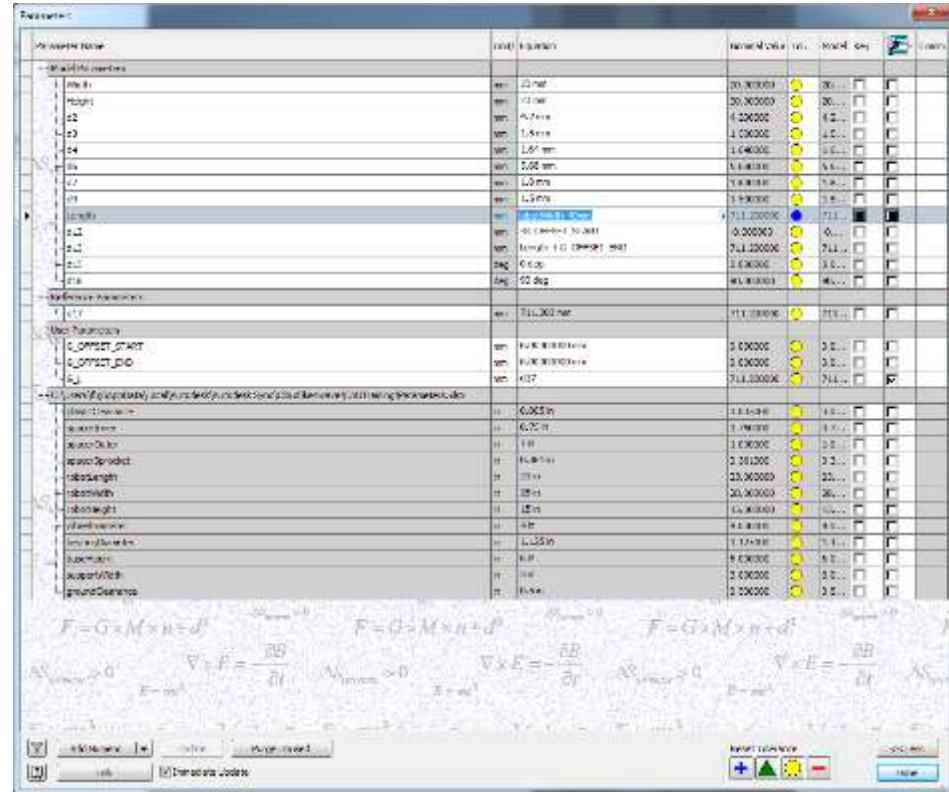
Create Axle Support

Create a new part and Save As Support.ipt

Link the Parameters worksheet to the part

Create a new sketch in the YZ plane

Click Rectangle button in Toolbar

Click on yellow center move mouse up and right

Type baseHeight and press Tab

Type supportWidth and press Enter
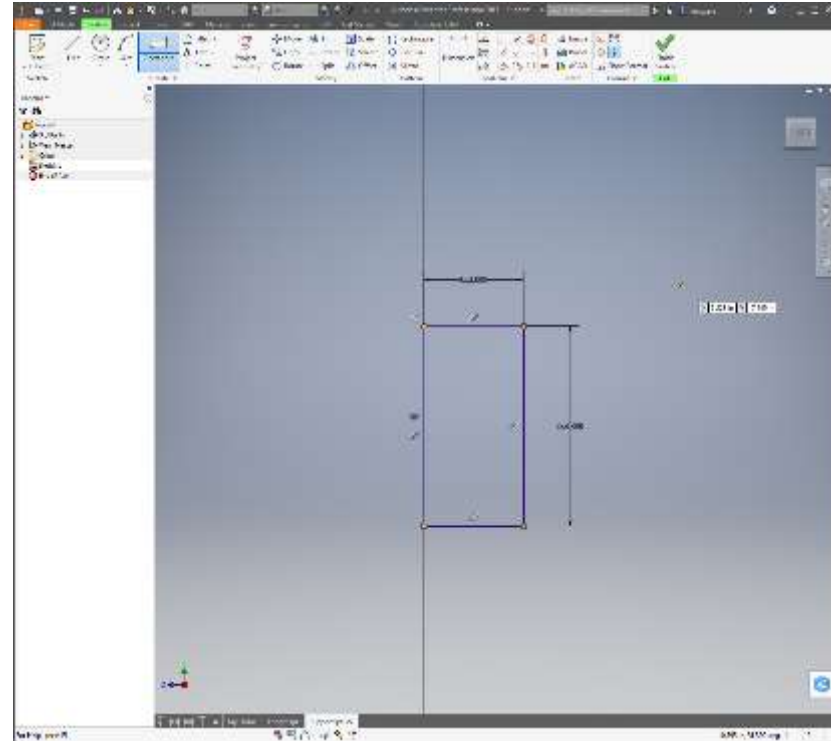
Extrude

Click Finish Sketch

Click on Extrude

Enter supportThick in the box under Distance
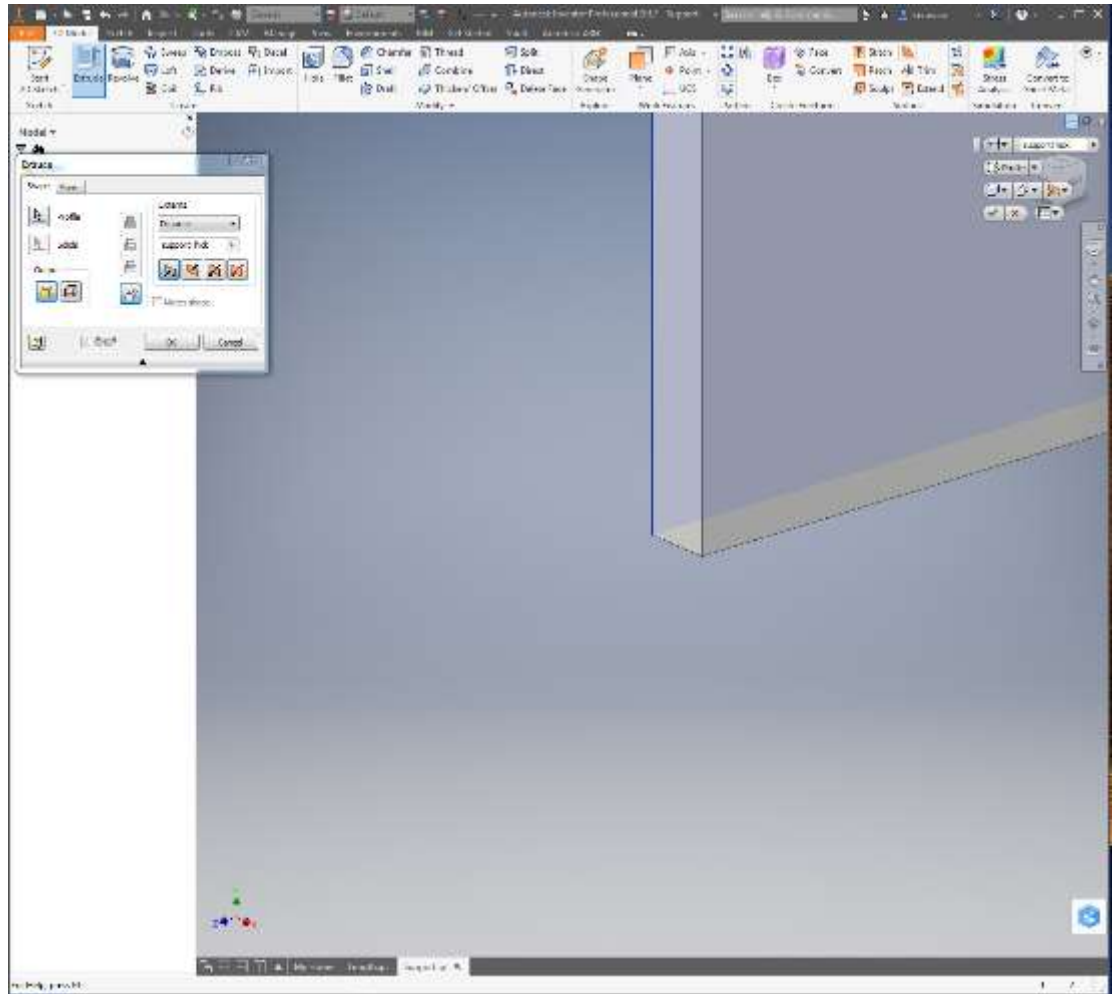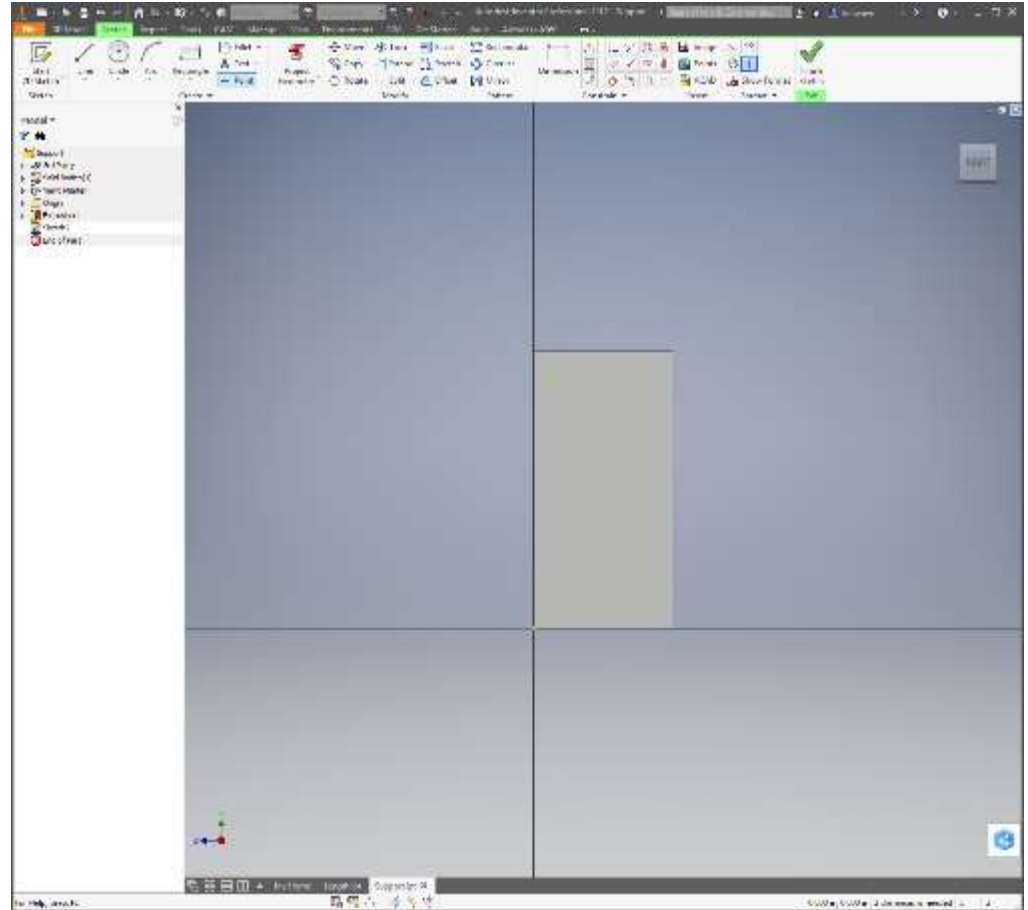
Click OK

Add Holes

Create a new sketch on the surface of the support

Click on the Point button the Toolbar

Click in the top left corner of the support to add a point

Position Point

Click on the Dimension Button

Click on the left side of the support

Click on the point

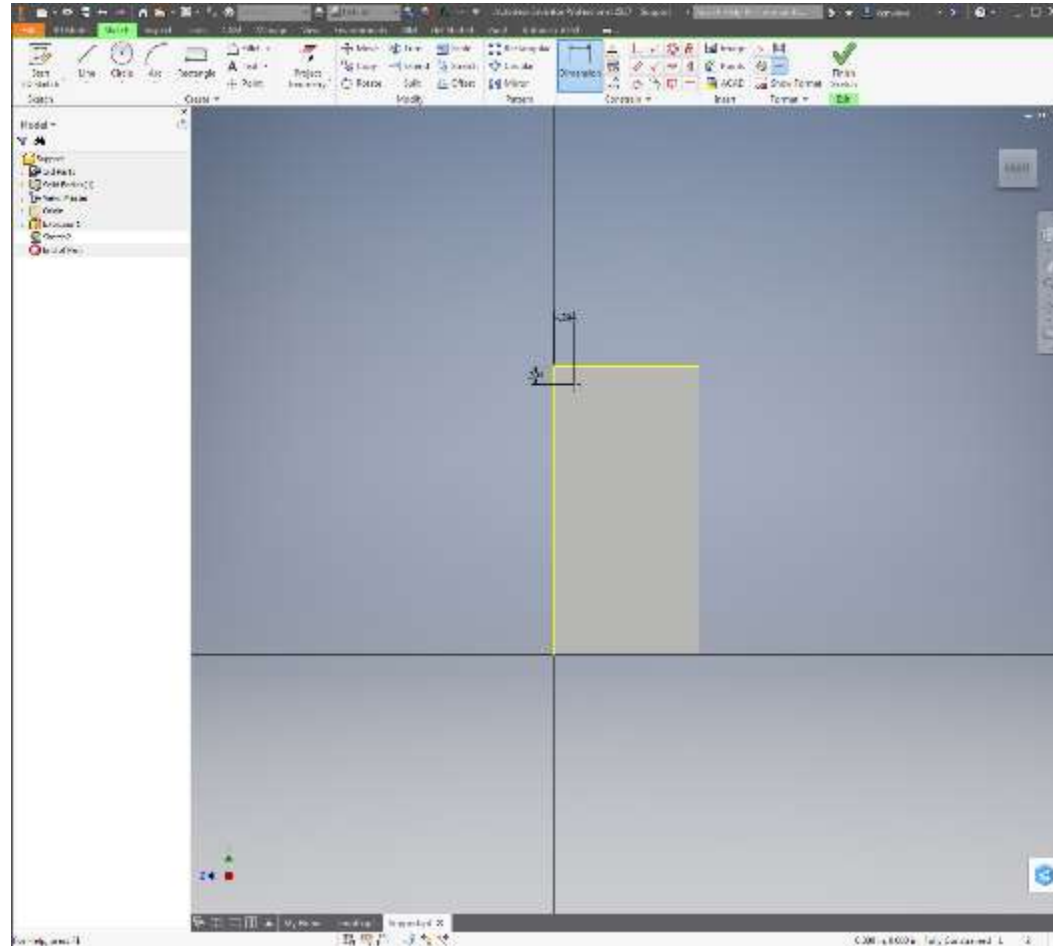Enter 10mm and Press Enter

Click on the top of the support

Click on the point

Enter 10mm and Press Enter

Repeat Points

Click on the Rectangular Button

Click on the Point

Click on the arrow below Direction1
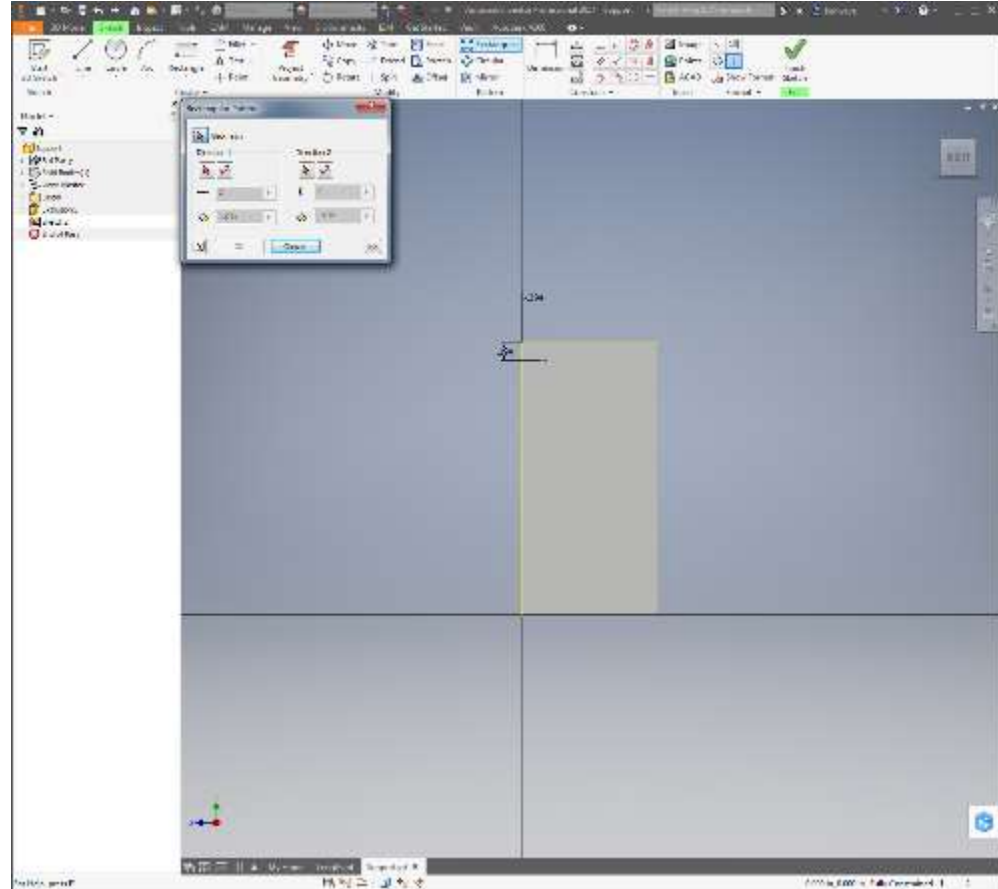
Click on the top of the support

Click on the double arrow below Direction1 to change the direction

Enter 3 into the box next to 3 dots

Enter (supportWidth-20mm)/2 into the next box

Repeat Points
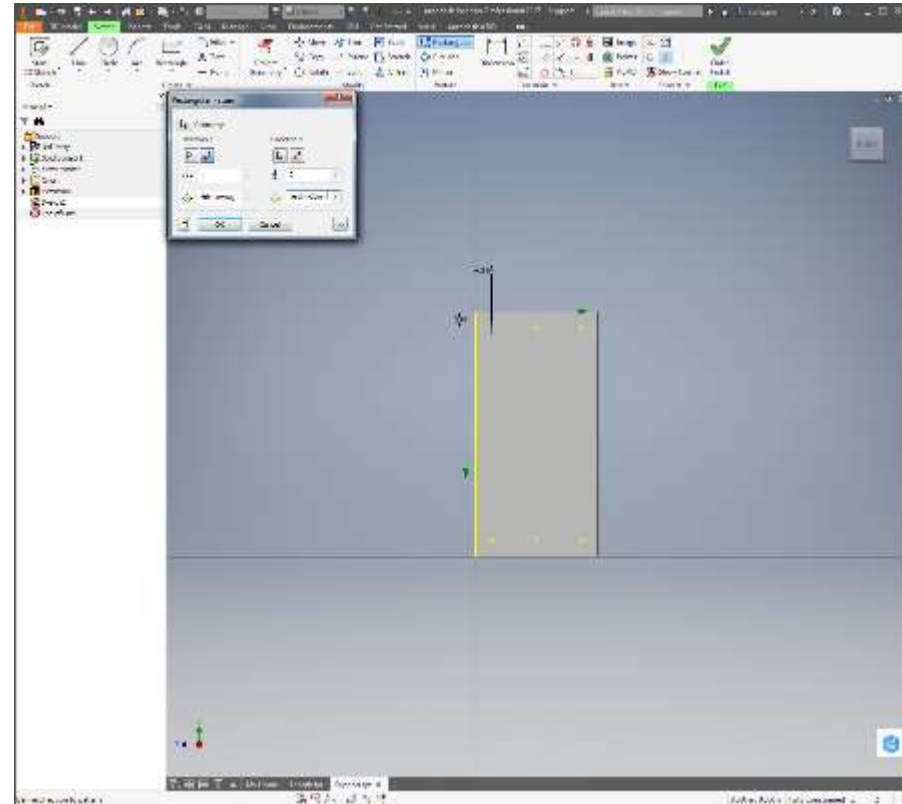
Click on the arrow below Direction3

Click on the left side of the support

Enter 2 into the box next to 3 dots

Enter baseHeight-20mm into the next box
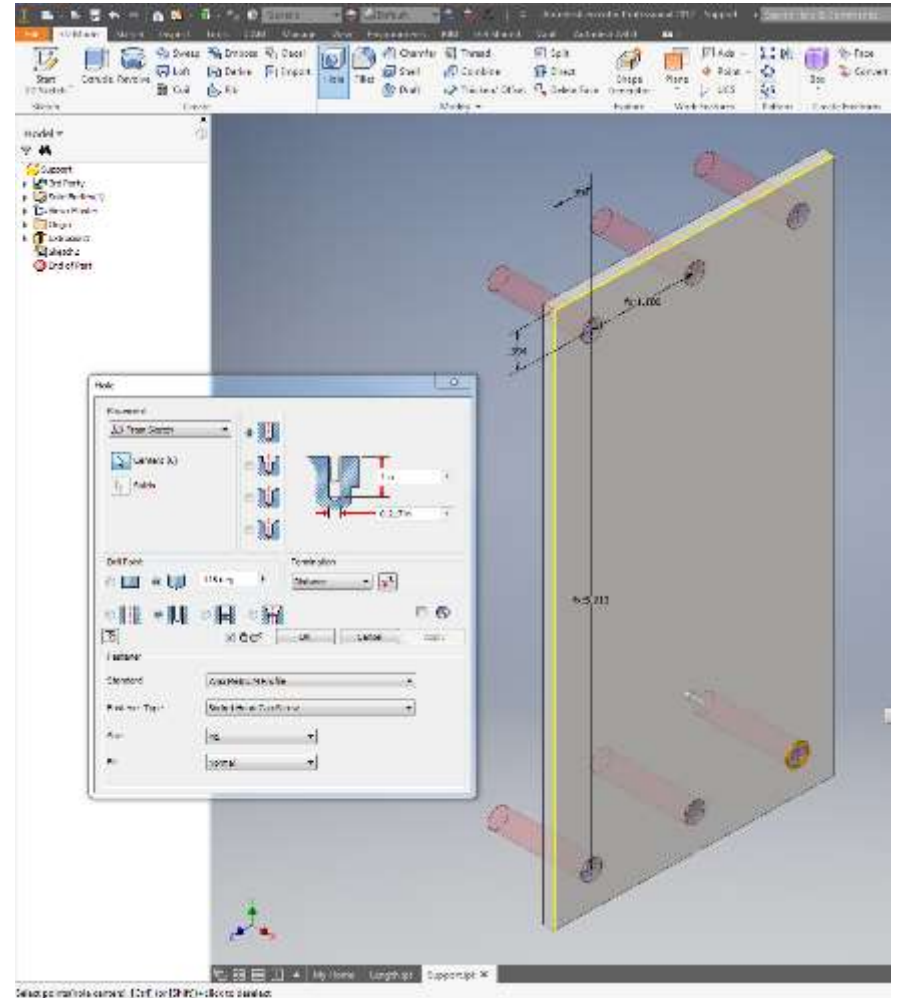
Click OK

Click Finish Sketch

Add Holes

Click on the Hole Button

Select Clearance (2nd radio button just about OK button)

Select Ansi Metric M, Socket Head Cap, M5, Normal in the dropdowns
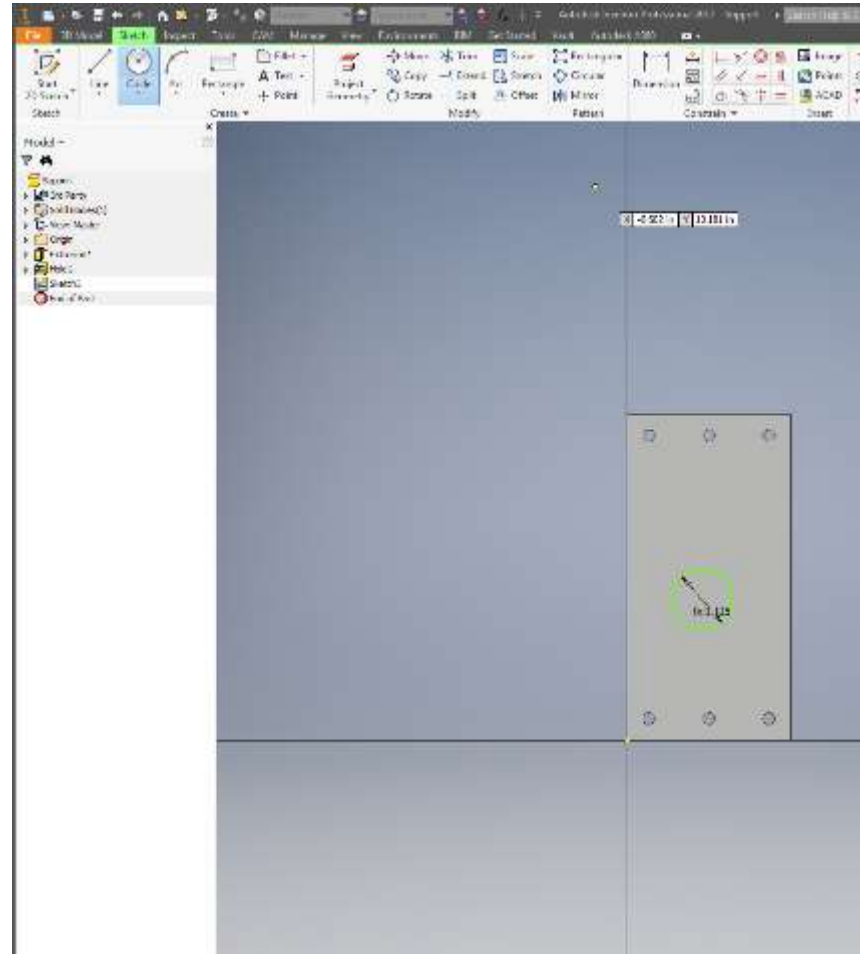
Click OK

Add Bearing Hole

Create a new sketch on the surface of the support

Click on the Circle button

Click on the sketch and move the mouse down

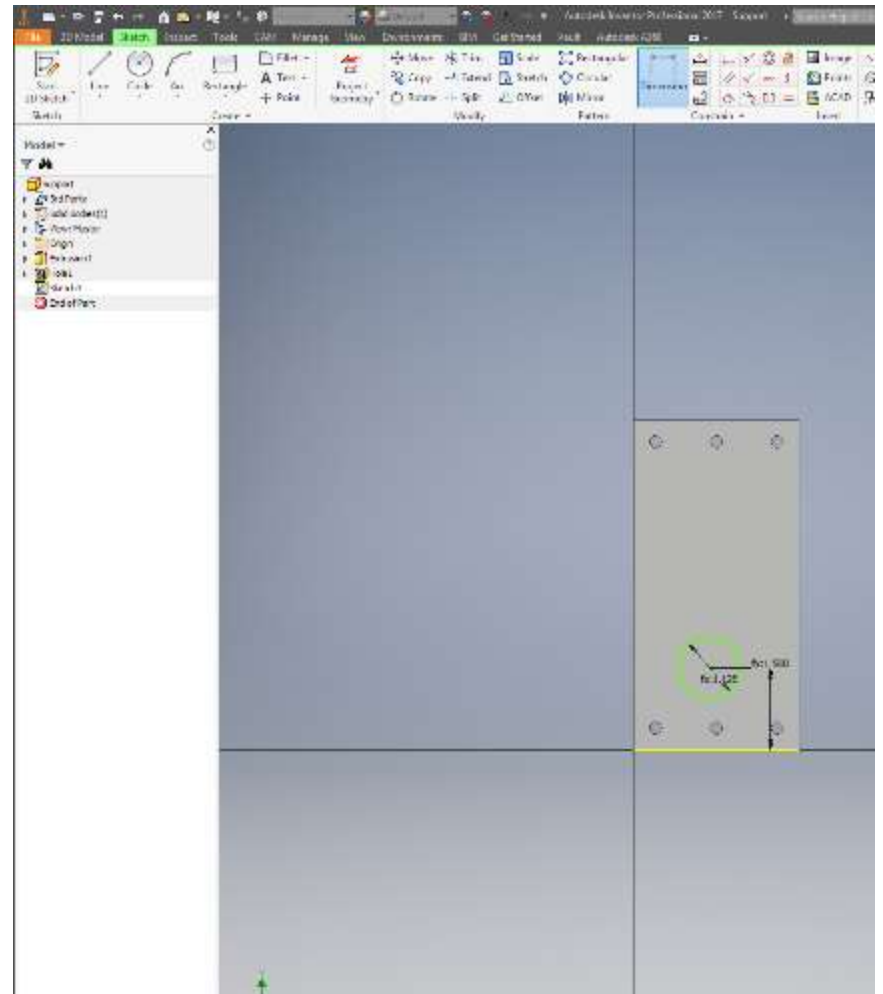Type bearingDiameter and press Enter

Position Bearing Vertically

Click on the Dimension Button

Click on the bottom of the support

Click on the center of the circle

Move mouse to the right and click

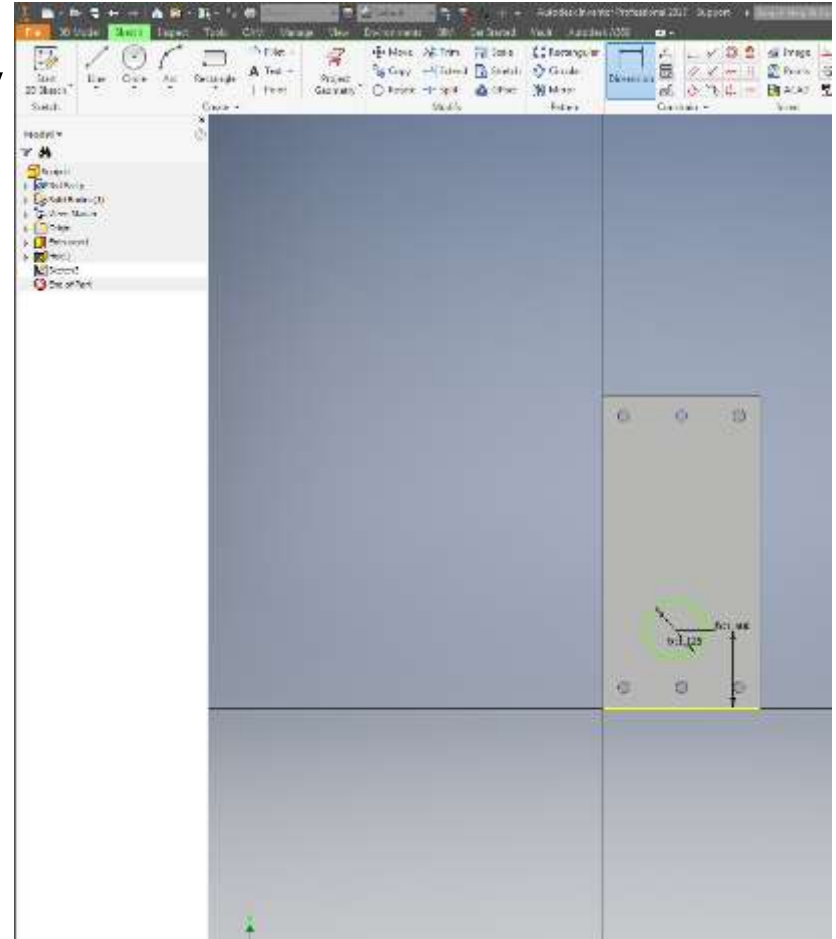Type wheelDiameter/2-groundClearance and press Enter

Position Bearing Horizontally

Click on the left side of the support

Click on the center of the circle

Move mouse to the down and click

Type supportWidth/2 and press
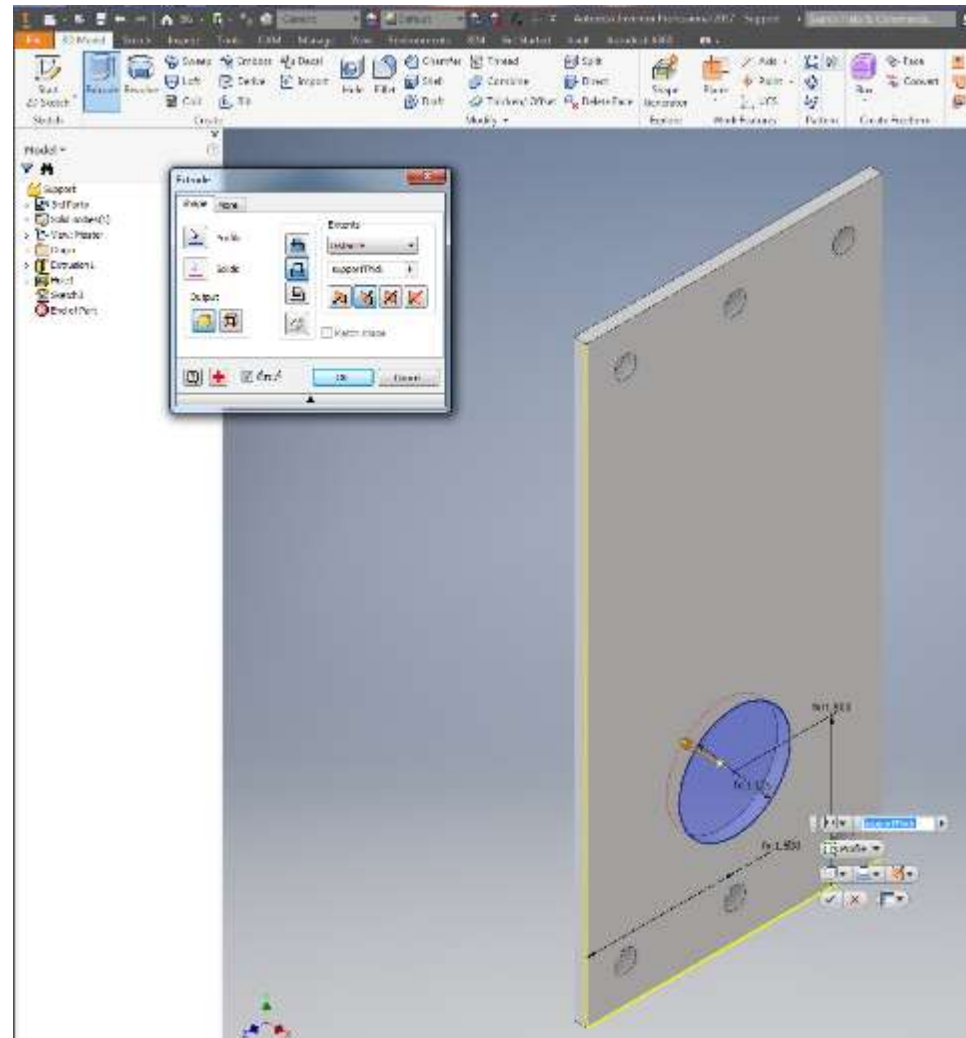Enter

Click Finish Sketch

Cut Hole With Extrude

Click Extrude Button

Select Cut in Dialog

Set distance to supportThick

Click OK

Create Side Assembly

Create a new assembly

Save the assembly as Side.iam

Link the Parameters worksheet
to the assembly

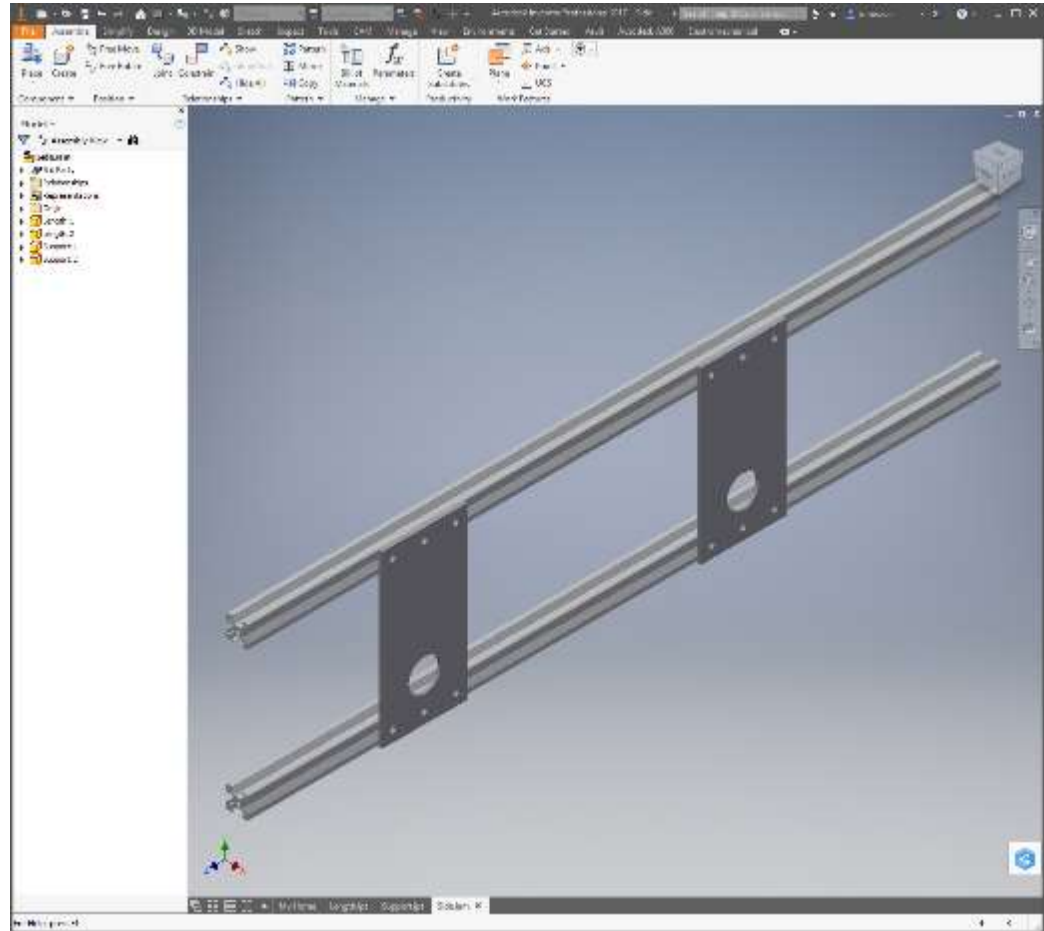Place the 2 Length.ipt and 2
Support.ipt

Constrain

Mate the Supports to the sides of the Length beams

Make the tops of the Supports Flush with the top of the top beam

Make the bottoms of the Supports Flush with the bottom of the bottom beam

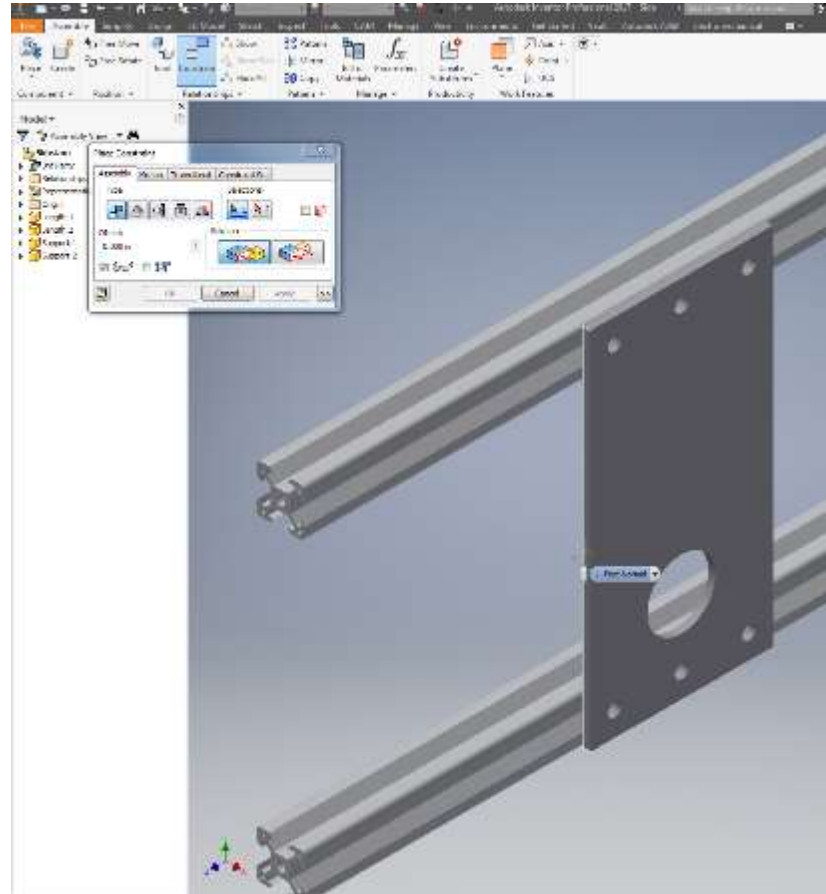Make one end of the top beam Flush with one end of the bottom beam

Constrain

With Flush Selected, click on the left edge of the left support

Click on the left end of the beam

Type supportOffset into the Offset box

Click OK

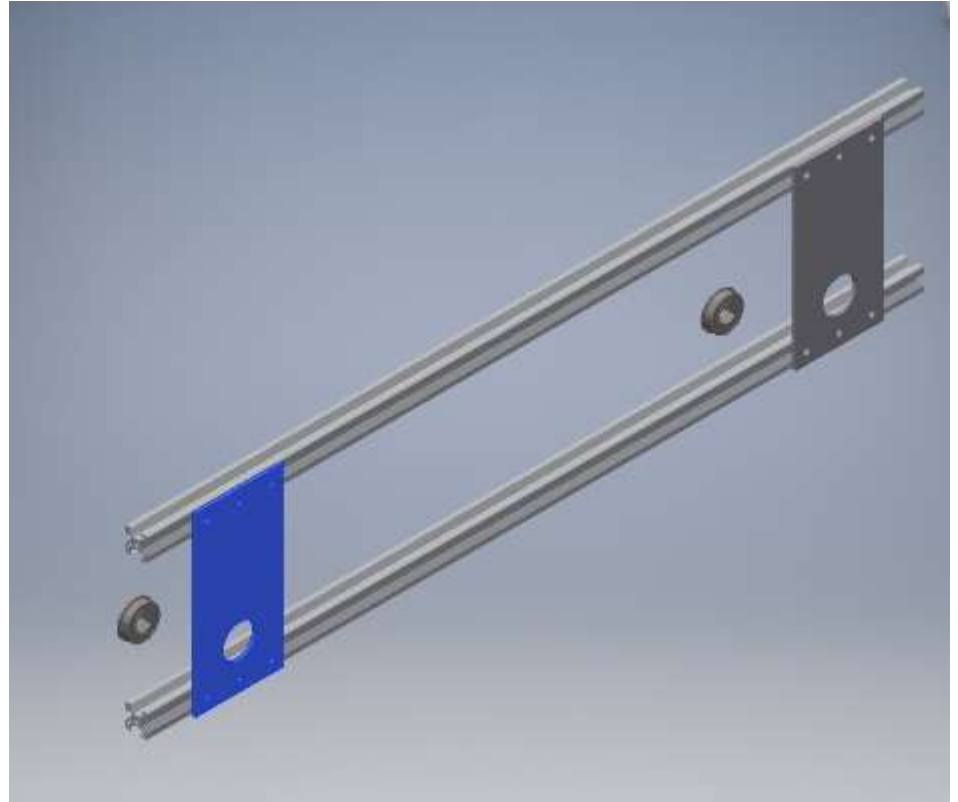Repeat the offset constraint on the right side

Add Bearings

Place 2 HexBearings5.ipt

Mate the flange of the bearing with surface of the Support

Constrain the center axis of the bearing with the center of the hole

Save Part

Assemble Frame

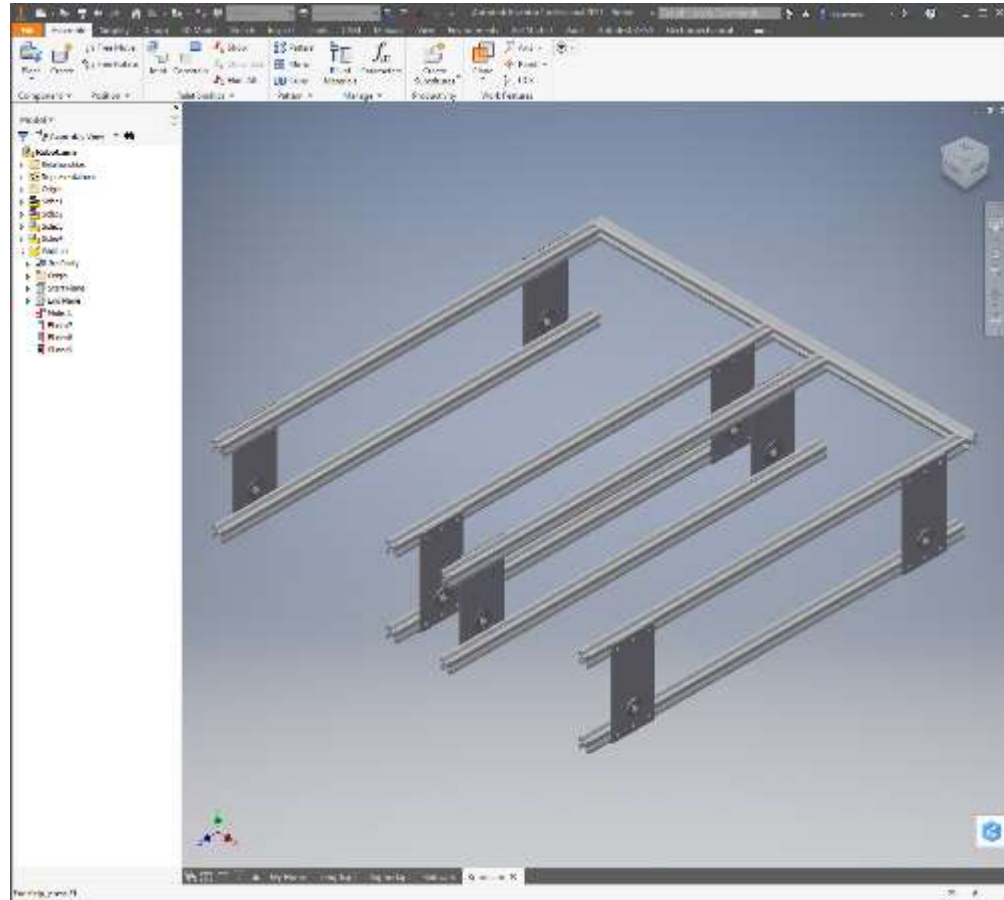Create a new assembly Robot.iam

Place the 4 Side.iam and 1 Width.ipt

Orient the Side assemblies as shown

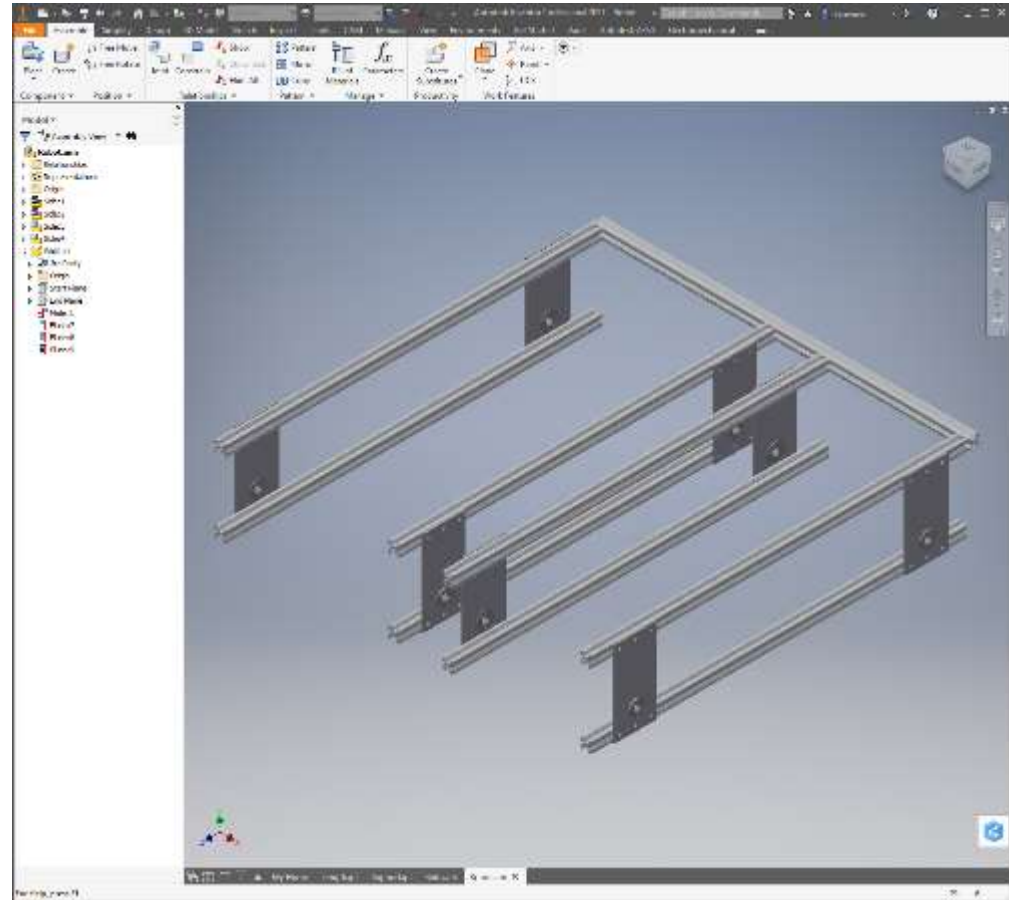Make the all the tops of the Side
assemblies Flush with each other

Make one end of the Side assemblies
all Flush

Mate one end of one Side assembly
with the Width beam

Assemble Frame

Make the outside Side assemblies
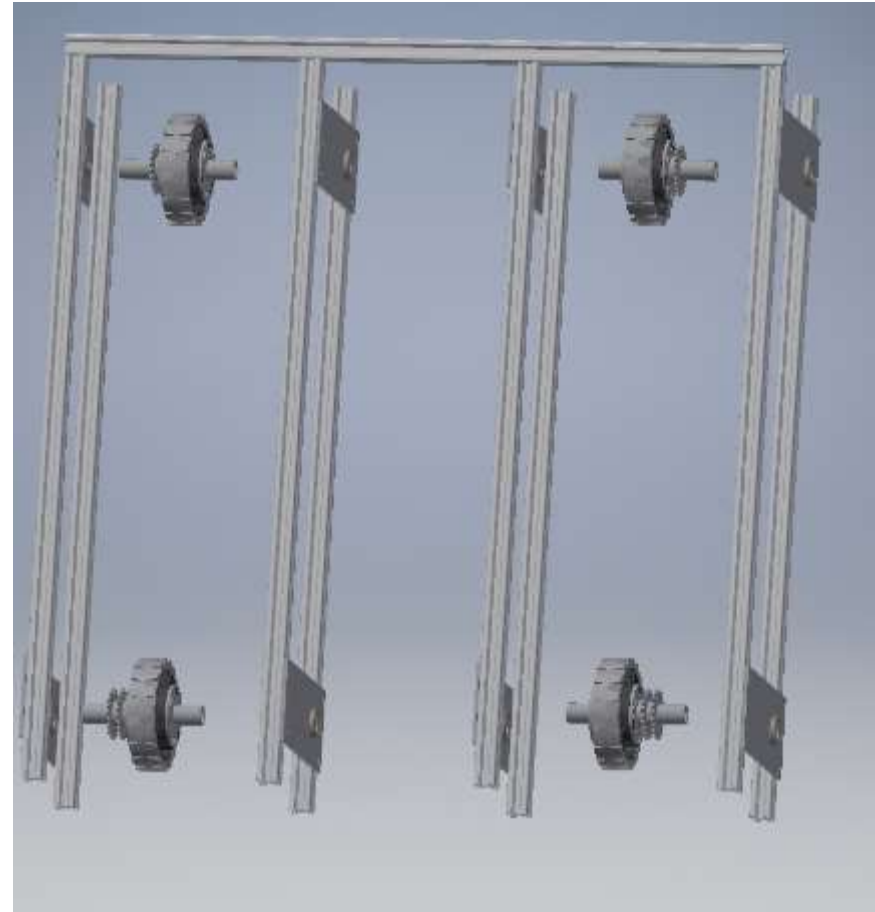Flush with ends of the Width beam

Add Wheels

Add FrontDrive.iam and Drive.iam assemblies to the Robot

Mate all 4 of the OutsideSpacers to the surface of the bearings

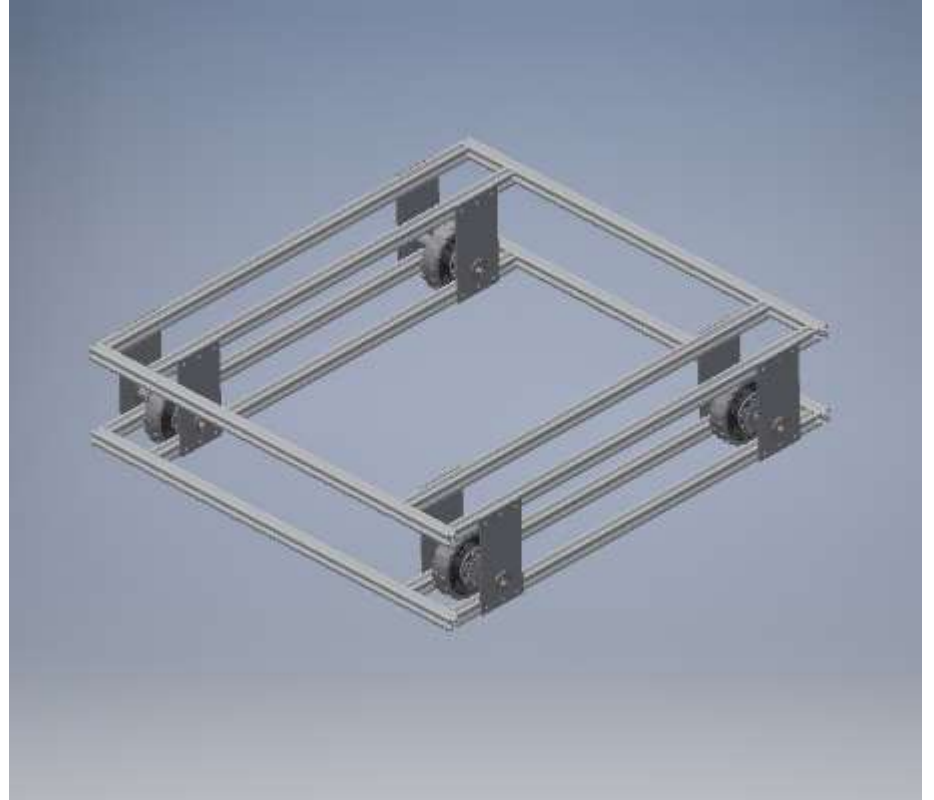Constrain the center axis of the wheel assemblies to the center axis of the bearings

Mate the InnerSpacers of the 2 Drive assemblies to the inner Side assemblies

Add Width Beam To Complete

Place 3 Width.ipt parts

Use Mate and Flush Constraints
attach the beams

Export for 3D Printing

Open InnerSpacer.ipt

Select Export > CAD Format

Select STL Files from Save As Type dropdown

If necessary, select model unit of measurement by Clicking Options...