

Computer Aided Design

First Robotics 4669
Galileo High School
Ken Fujimoto

Creating A Model

Model is an assembly which consists of parts

Two main types of files in Autodesk Inventor

.ipt - Inventor Part file

.iam- Inventor Assembly file

Part files may be created by users or download from manufacturers

Assembly files created by selecting and connecting parts

Assemblies may also consist of connected sub assemblies

Parametric Models

Parameters are values which can be used to describe a model

Length, width and height are common parameters

The power of parameters is that the values can be changed after the model is built and the model will automatically update

With the proper model things like the robot's width, height and length can be adjusted

Other things like arm length, location and width can also be adjusted

Setting Up The Project

Start Autodesk Inventor

Click on the Projects button

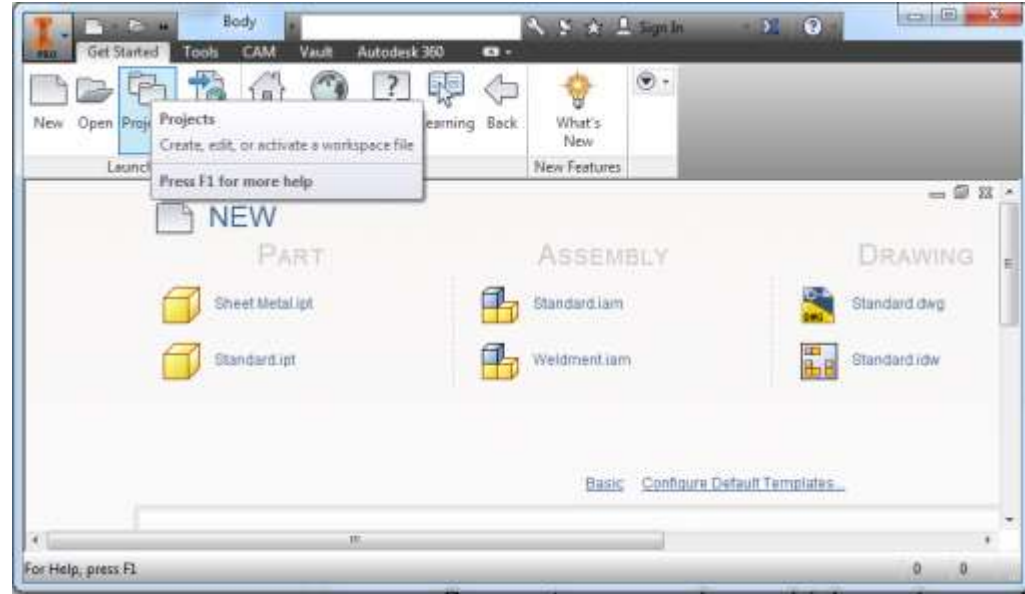
Click on the New button

Select New Single User Project

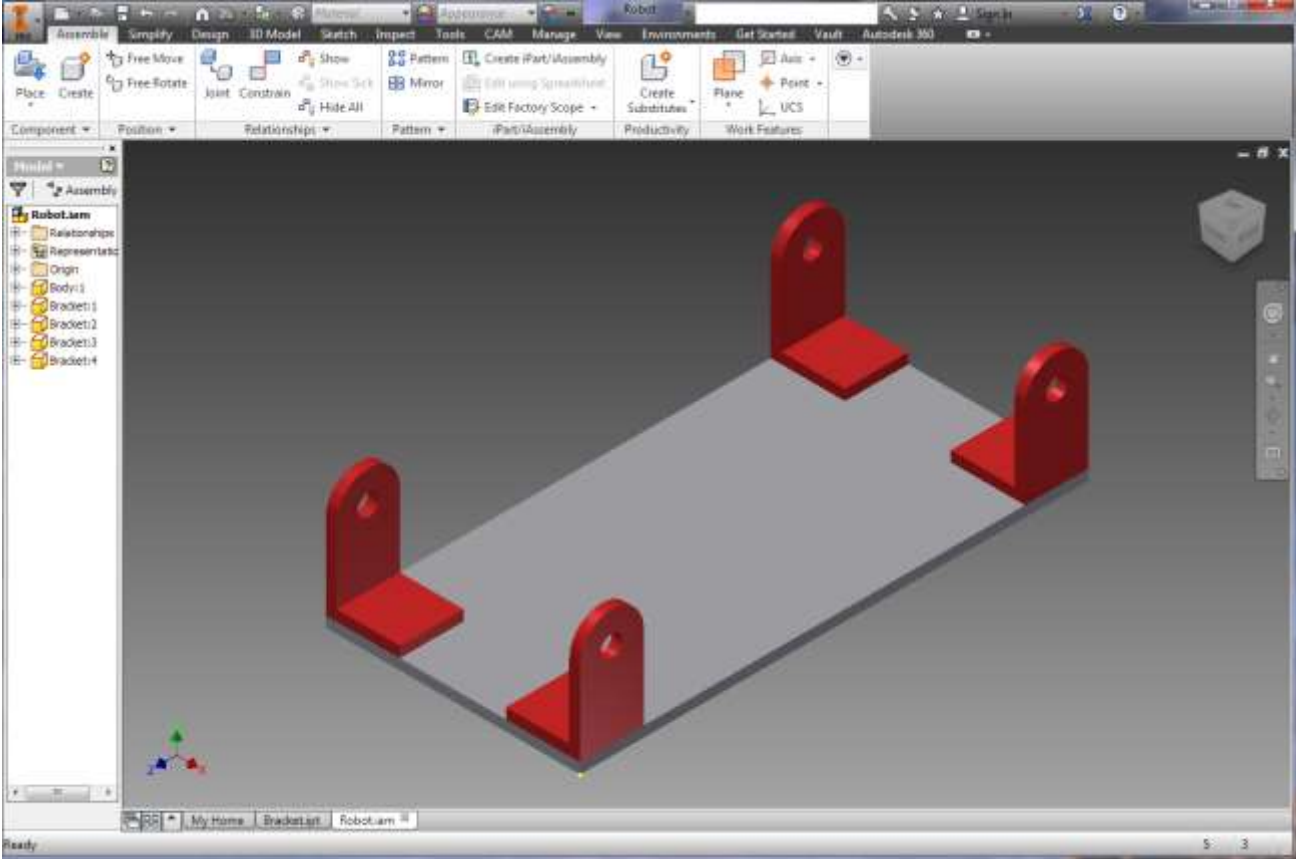
Click Next

Enter Project Name and Folder

Click Finish then Ok



Model To Be Created



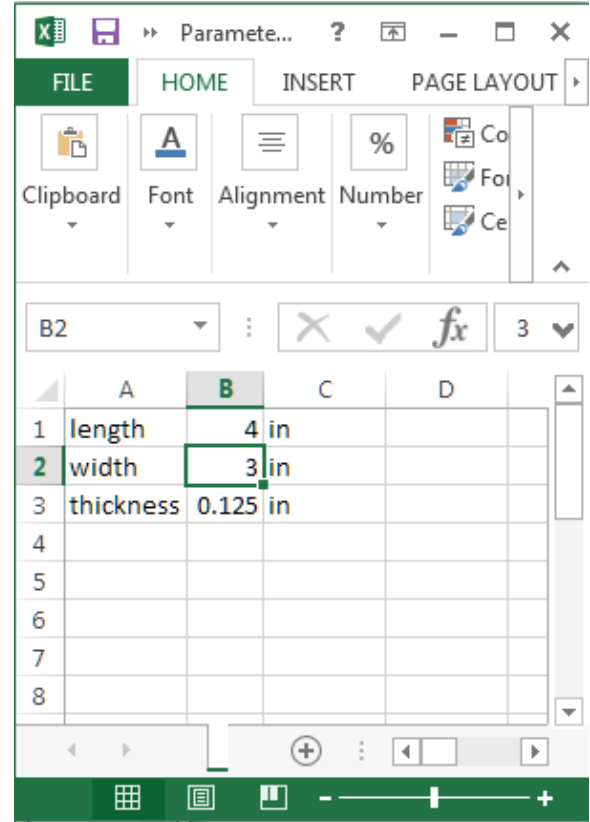
Setting Up The Parameter Worksheet

Start Microsoft Excel

Select New Blank Workbook

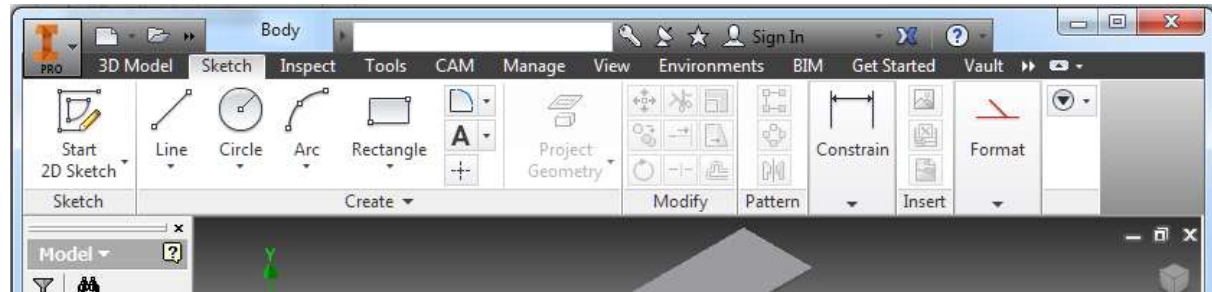
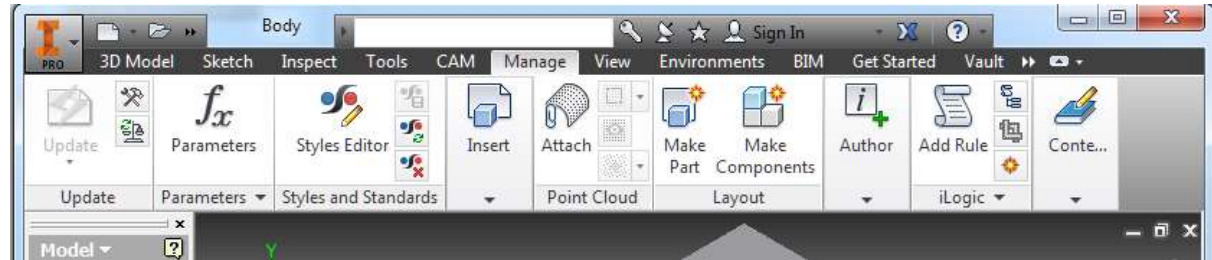
Enter the length, width and thickness as shown

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



Tool Bar Tabs

The tool buttons are organized into tabs which are named 3D Model, Sketch, Inspect, Tools, CAM, Manage, View etc.



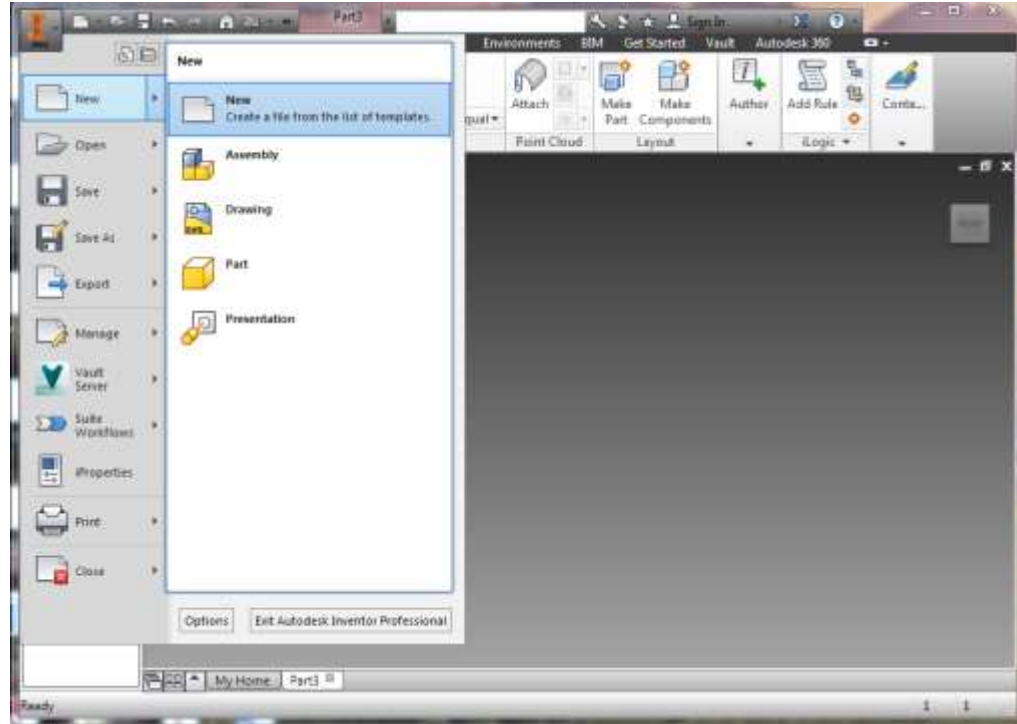
Creating A Part

In the I menu (top left) click New and select New

In the left window of the dialog click on English

In the middle double click on Standard (in).ipt

Click on the I in the top left to Save the part as “Body”



Connecting Parameters to the Part

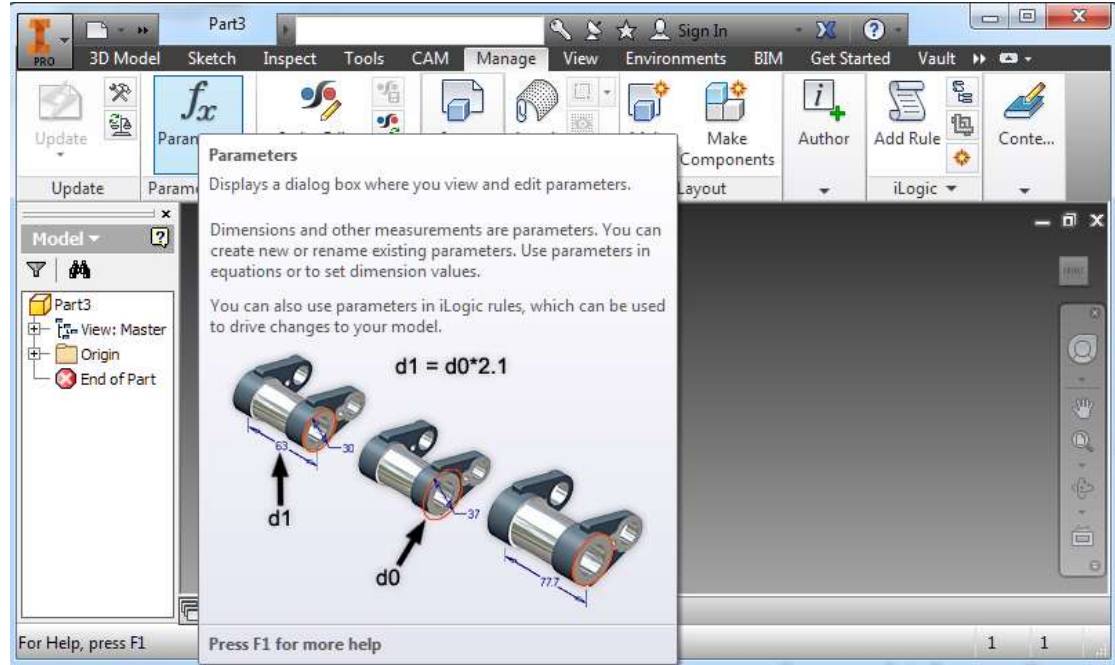
Click on the Manage Tab

Click on the Parameters
Button

Click on the Link Button in the
lower left

Select the Parameters
Workbook in the Project
Directory

Click Done



Creating Volumes

Parts are made by adding and subtracting 3D volumes

Volumes are made by first creating a 2D sketch and then extruding it into 3D

The 2D sketches must first be placed and oriented in 3D space

Next 2D lines and shapes are added to the sketch

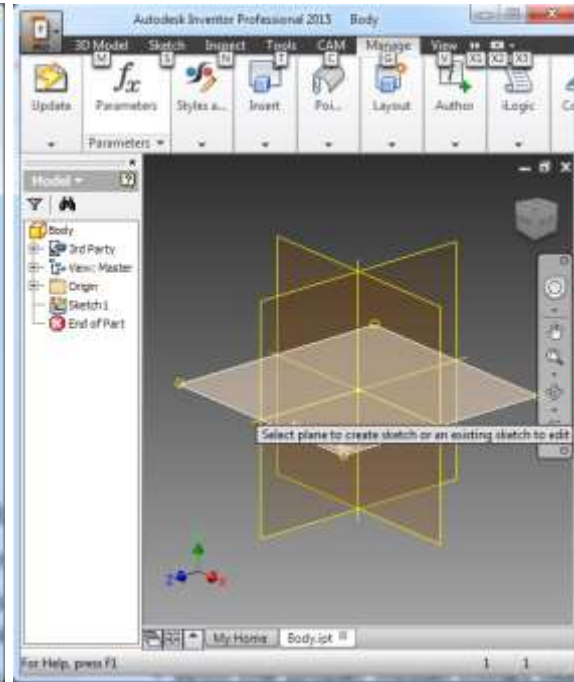
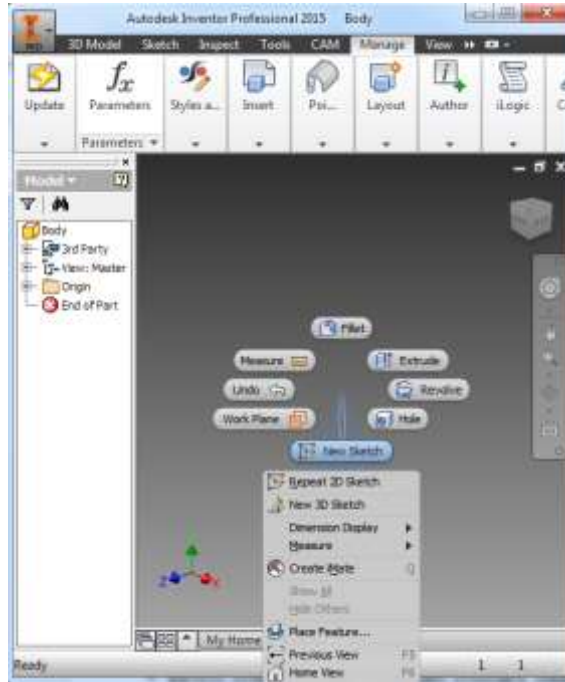
Finally the 2D shapes are extruded into 3D volumes

Creating A Sketch

Right click in the lower right part view and select New Sketch

Click the plane for the sketch.

Choose the XZ plane



Draw Rectangle

Click the Rectangle Button

Click on the yellow origin dot

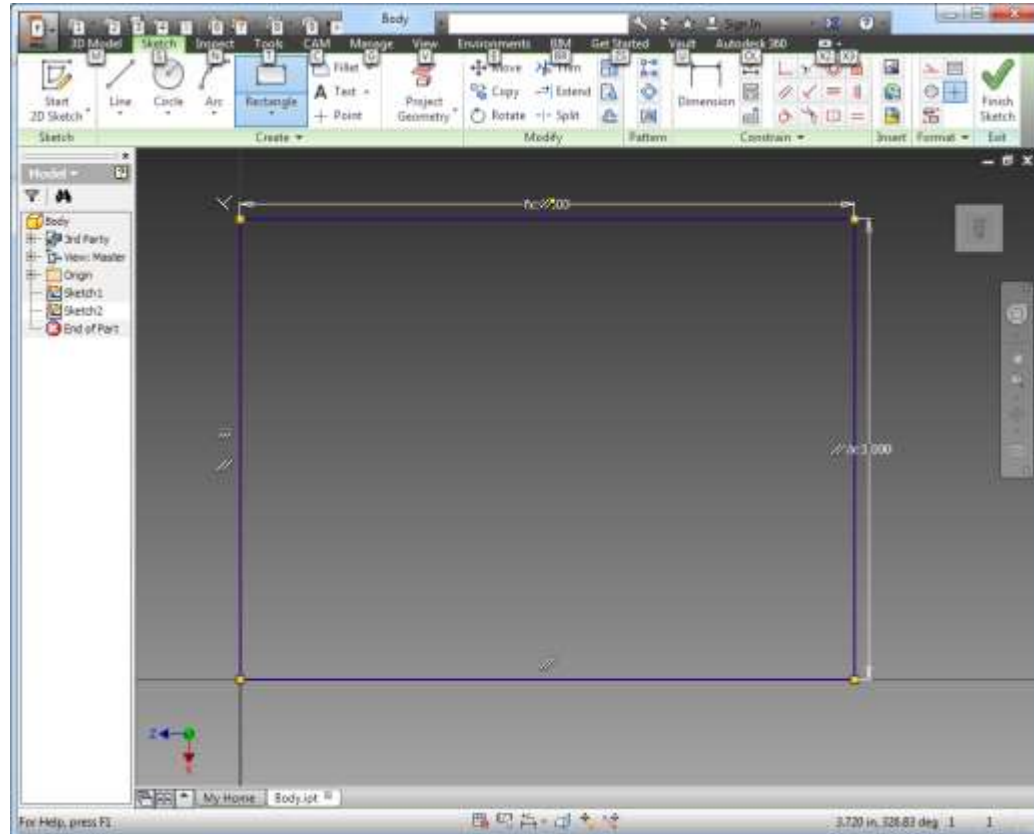
Drag mouse up and right

Type 'width'

Press the Tab key

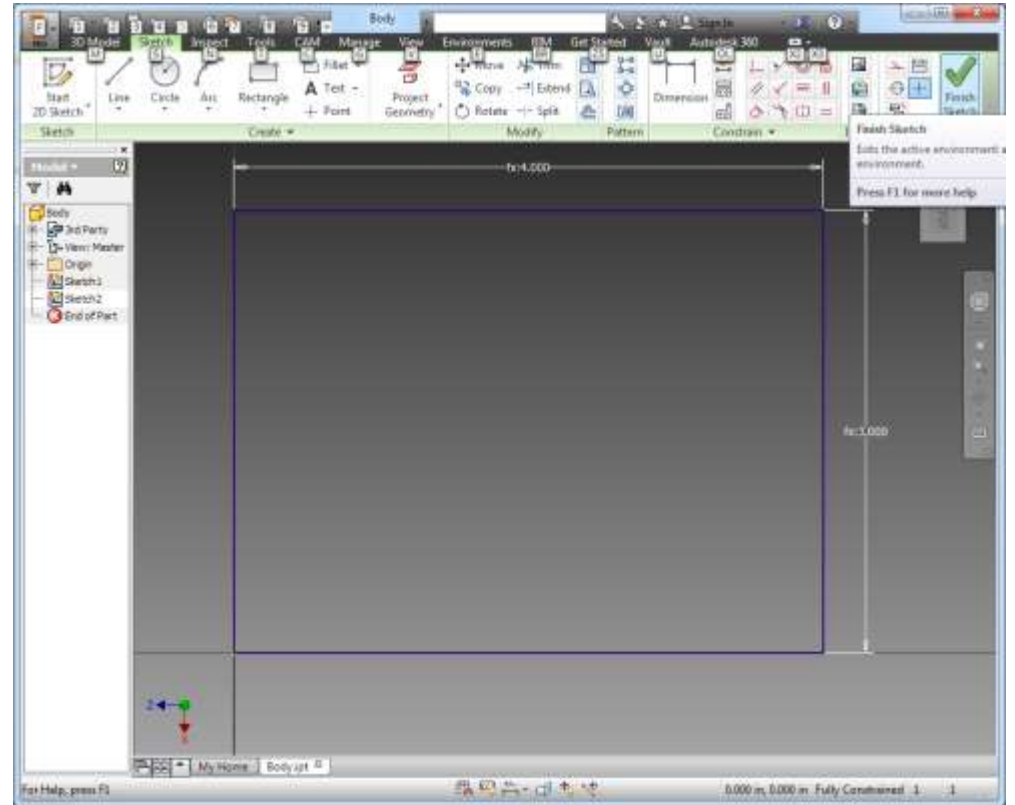
Type 'length'

Press Enter key



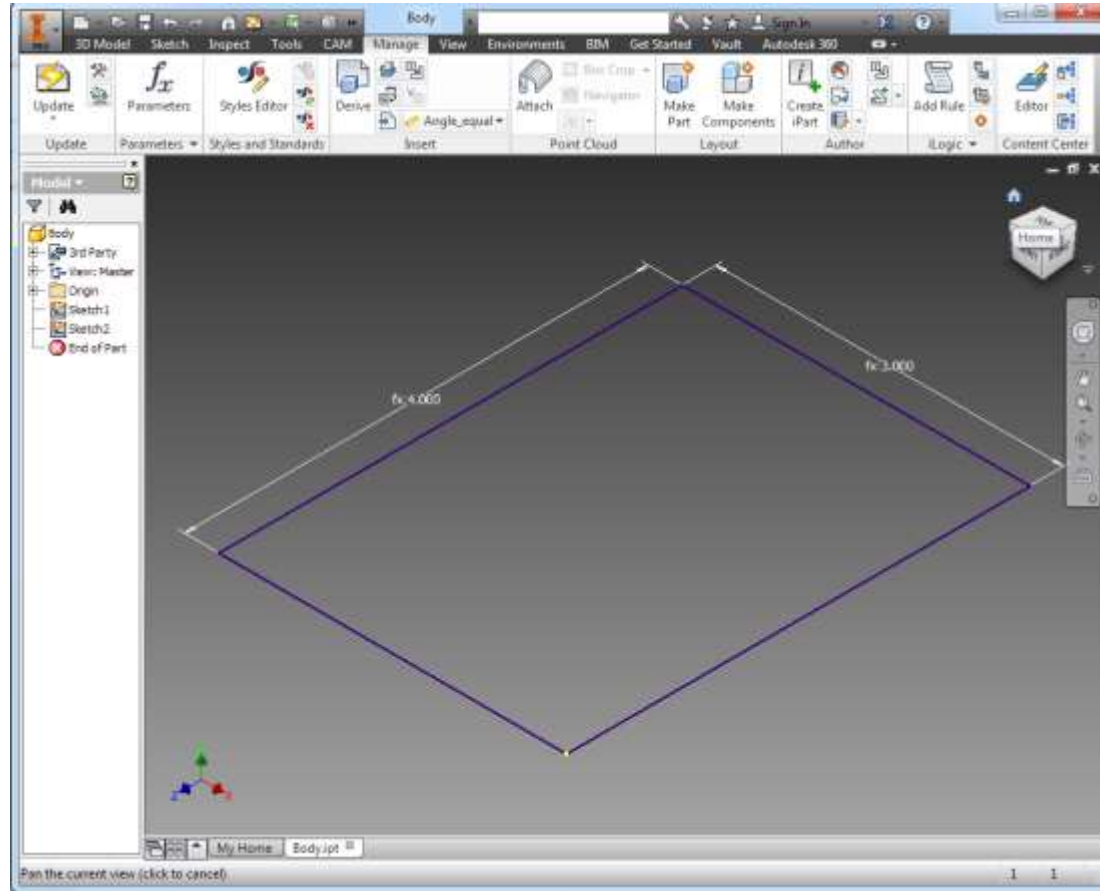
Finish Sketch

Click Finish Sketch Button



Center Drawing

Click the little house icon above the cube in the upper right part of the part window to center the drawing

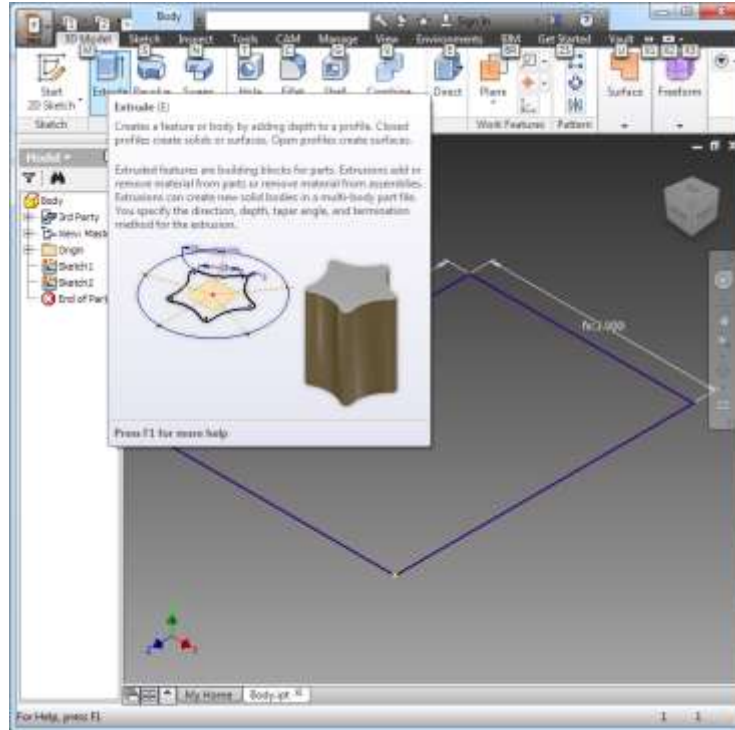


Extrude

Click on the Extrude button under the 3D Model tab

In the Extrude dialog type in 'thickness' under the Distance dropdown

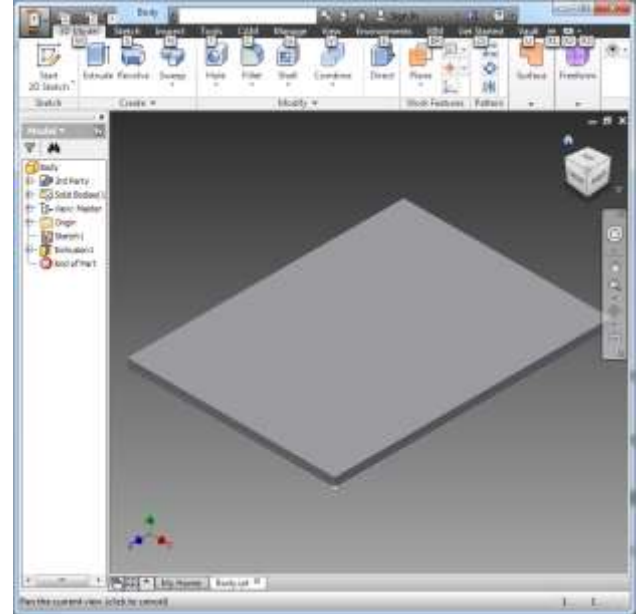
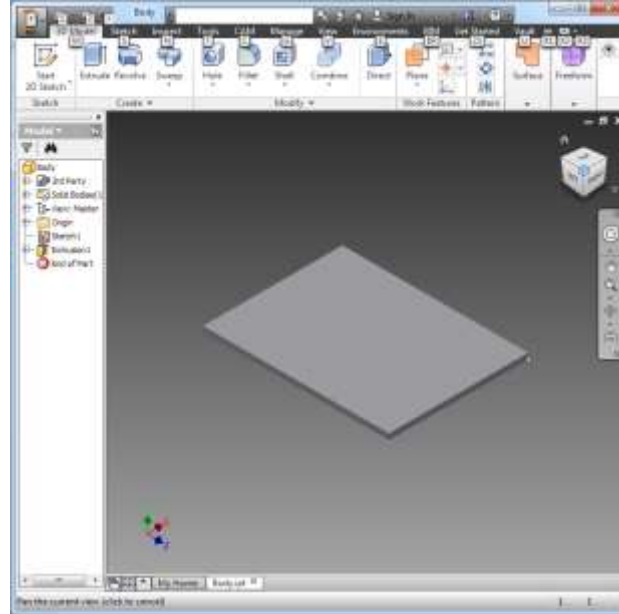
Click OK in the dialog



Navigation Cube

To view the part from various directions, you can click on the faces, corners or edges of the cube to rotate and scale the part

Clicking on the house icon returns you to the home view

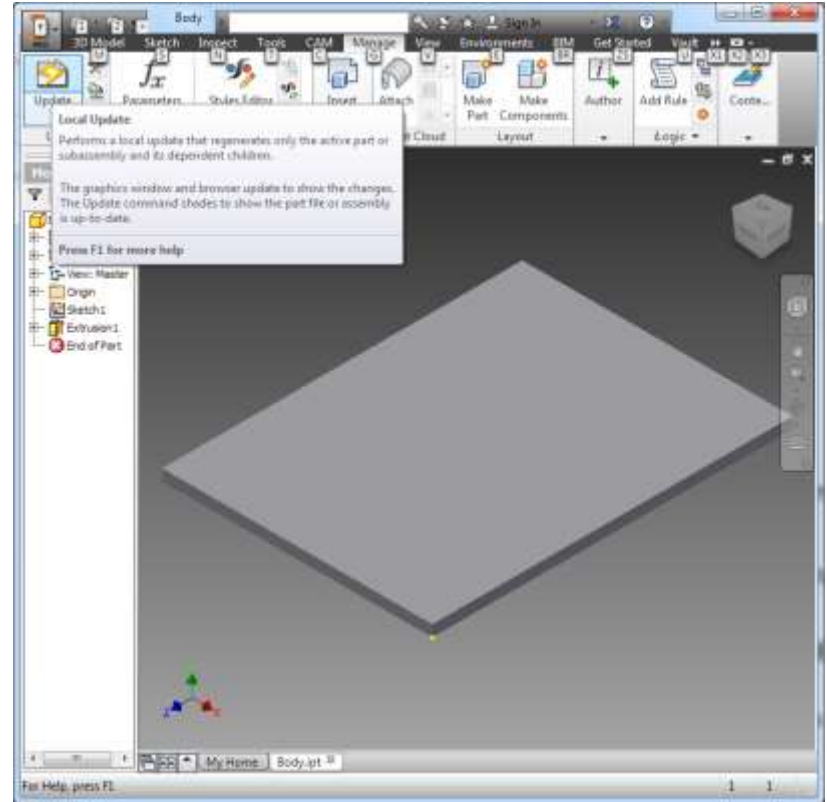


Changing Parameters

Change the length in the spreadsheet to 7 in

Save the spreadsheet

Go to Manage tab and click on the Update button to see the model change



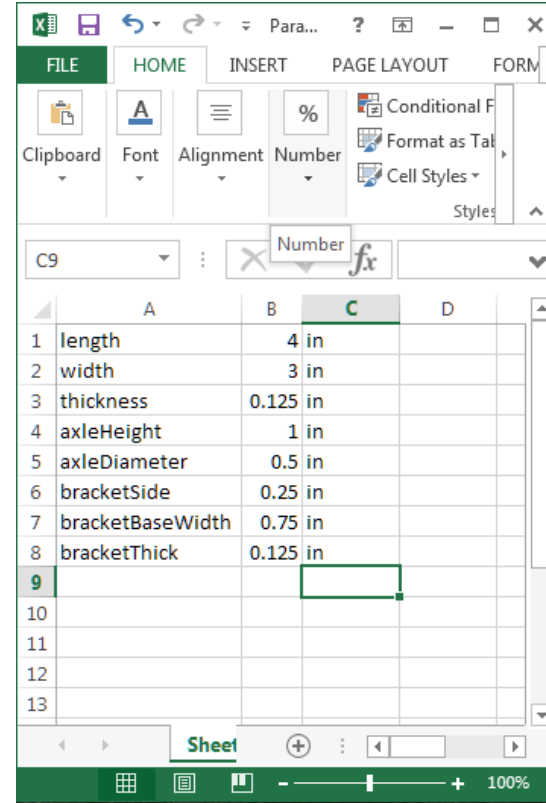
Create Bracket Parameters

To create a bracket to connect an axle to the body, the parameters need to be defined

Add the bracket parameters axleHeight, axleDiameter, bracketSide, bracketBaseWidth, and bracketThick to the spreadsheet

Save the spreadsheet

You should never type numbers into parts files. All dimensions should come from the Parameter spreadsheet or be defined as equations of parameters



The screenshot shows an Excel spreadsheet with the following data:

	A	B	C	D
1	length	4 in		
2	width	3 in		
3	thickness	0.125 in		
4	axleHeight	1 in		
5	axleDiameter	0.5 in		
6	bracketSide	0.25 in		
7	bracketBaseWidth	0.75 in		
8	bracketThick	0.125 in		
9				
10				
11				
12				
13				

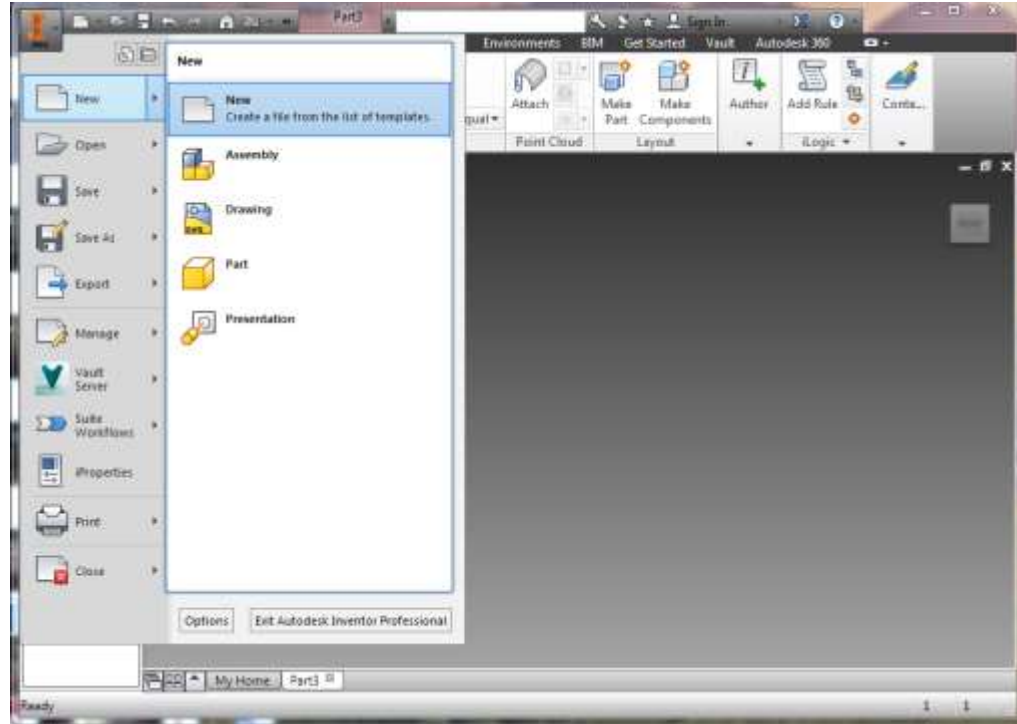
Creating The Bracket Part

In the I menu (top left) click New and select New

In the left window of the dialog click on English

In the middle double click on Standard (in).ipt

Click on the I in the top left to Save the part as “Bracket”



Connecting Parameters to the Bracket Part

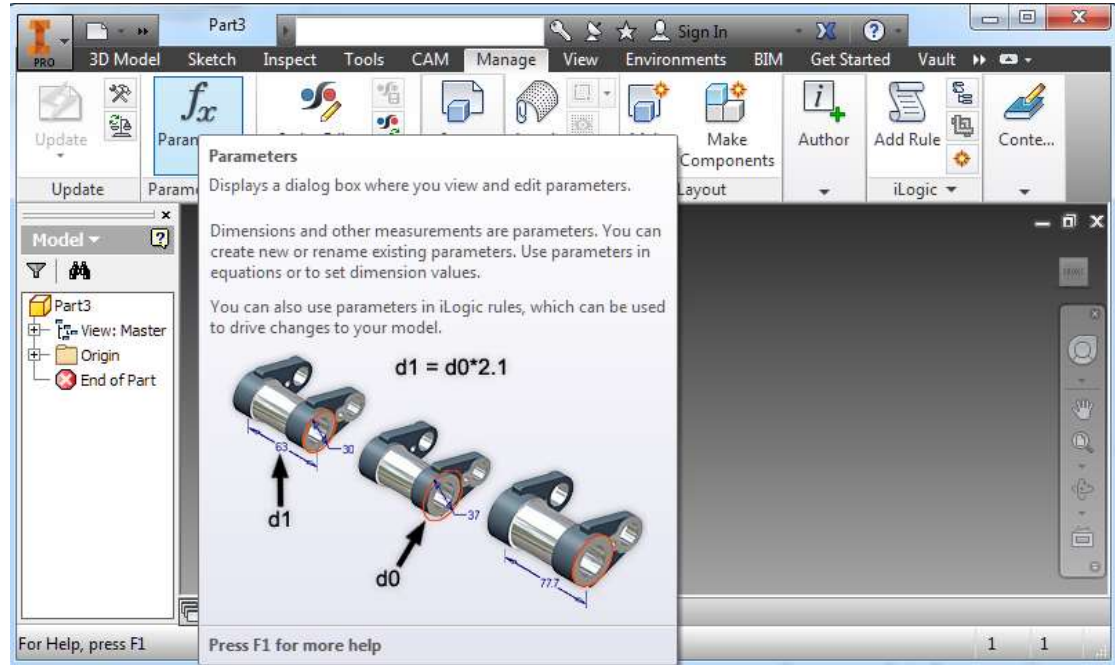
Click on the Manage Tab

Click on the Parameters
Button

Click on the Link Button in the
lower left

Select the Parameters
Workbook in the Project
Directory

Click Done

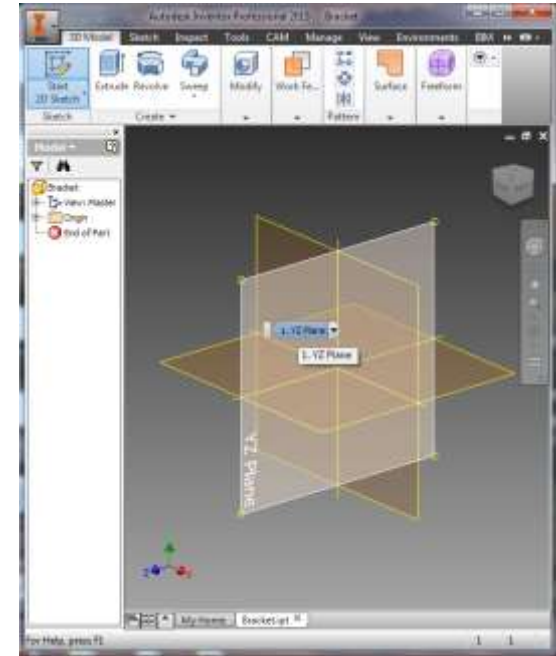
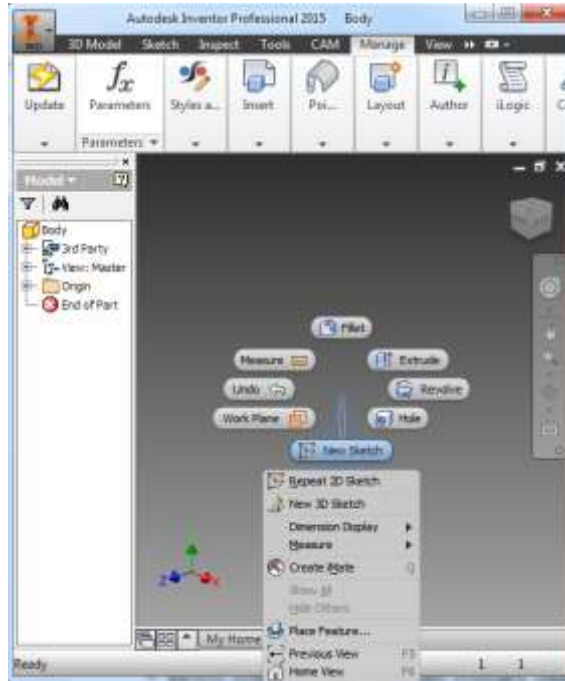


Creating The Bracket Sketch

Right click in the lower right part view and select New Sketch

Click the plane for the sketch.

Choose the YZ plane



Draw Circle

Click the Circle Button

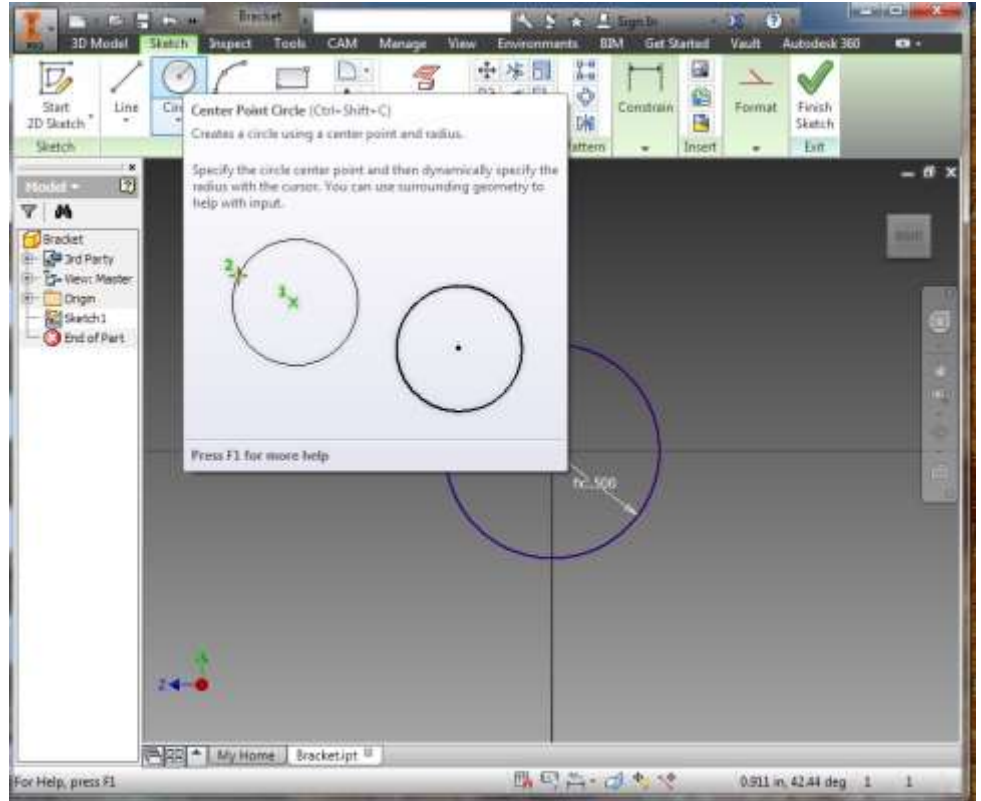
Click on the yellow origin dot

Drag mouse right

Type 'axleDiameter+2*axleSide'

Press Enter key

Click Finish Sketch

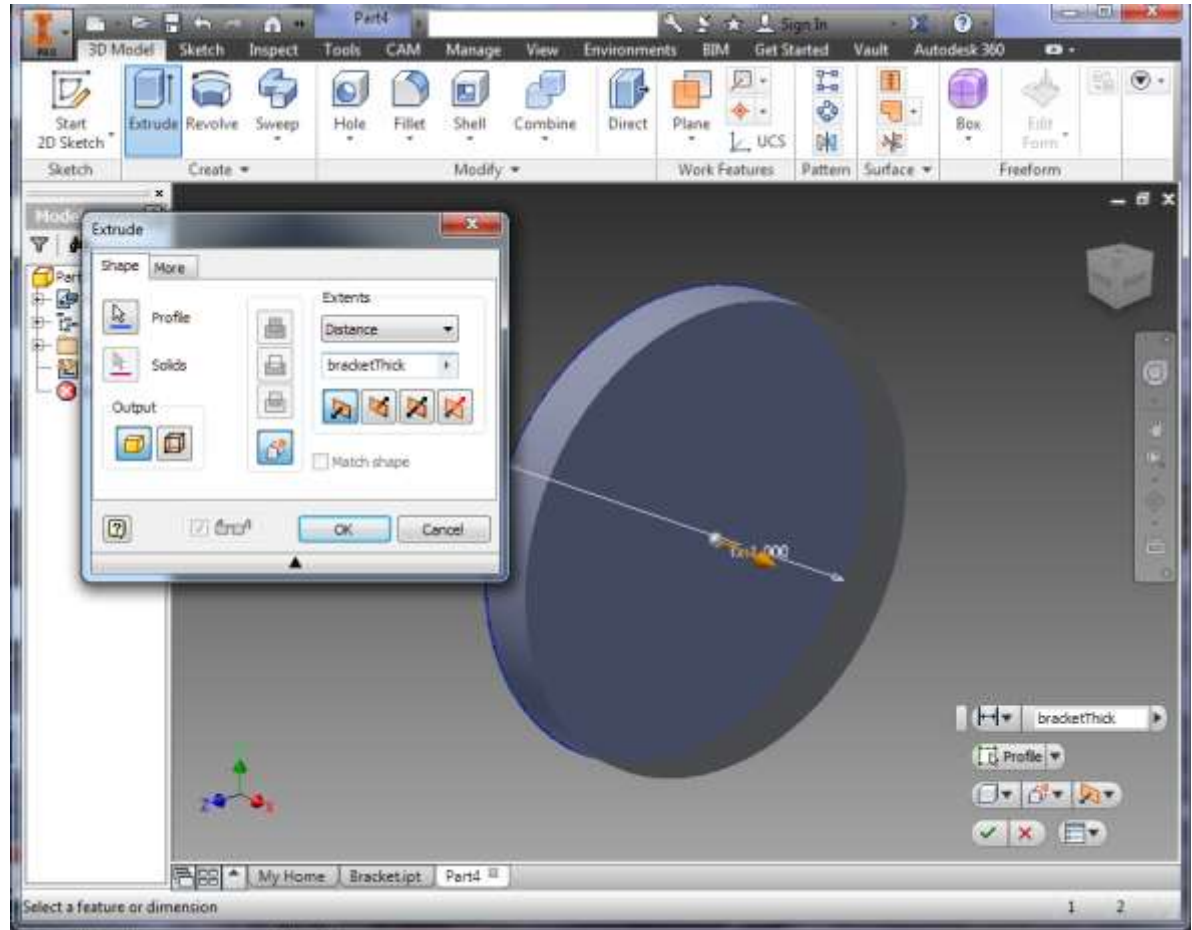


Extrude

Click on the Extrude button under the 3D Model tab

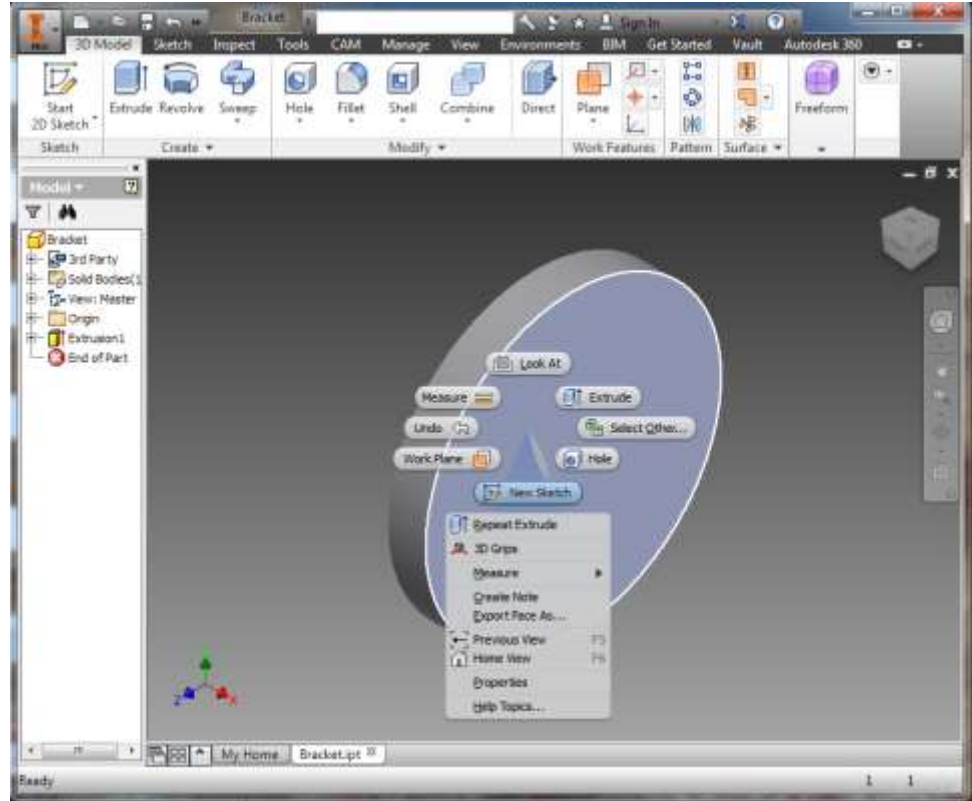
In the Extrude dialog type in 'bracketThick' under the Distance dropdown

Click OK in the dialog



Create Sketch

Right Click on the flat face of the disc and select New Sketch



Draw Rectangle

Select the Rectangle Tool and click on the sketch below and to the left of the center of the disc

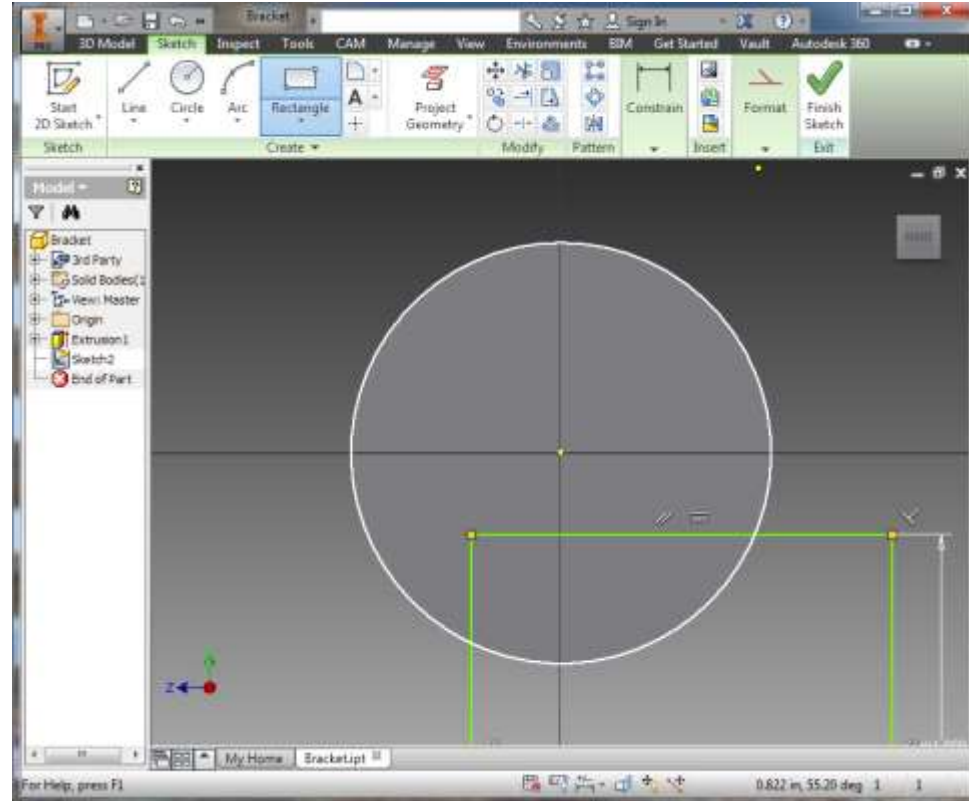
Drag mouse down and right

Type 'axleDiameter+2*axleSide'

Press the Tab key

Type 'axleHeight'

Press Enter key



Constrain Horizontal Location

Select Dimension Tool

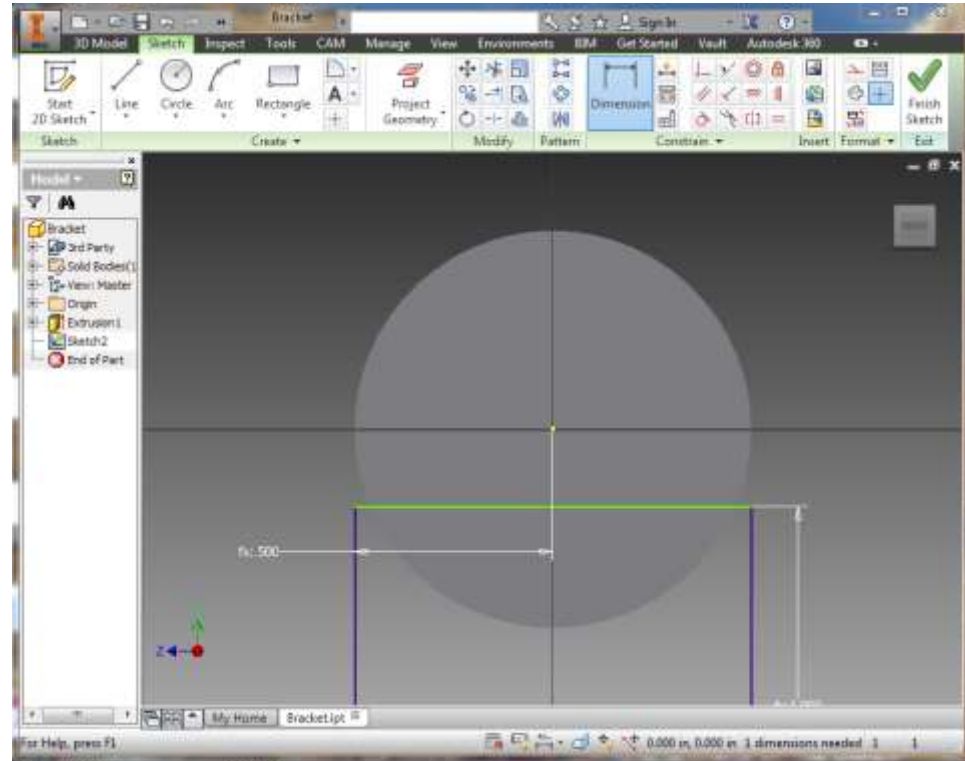
Click on left side of rectangle

Click on yellow origin

Drag mouse down and click

Type 'axleDiameter/2+bracketSide' into dialog

Press Enter



Constrain Vertical Location

Select Dimension Tool

Click on top of rectangle

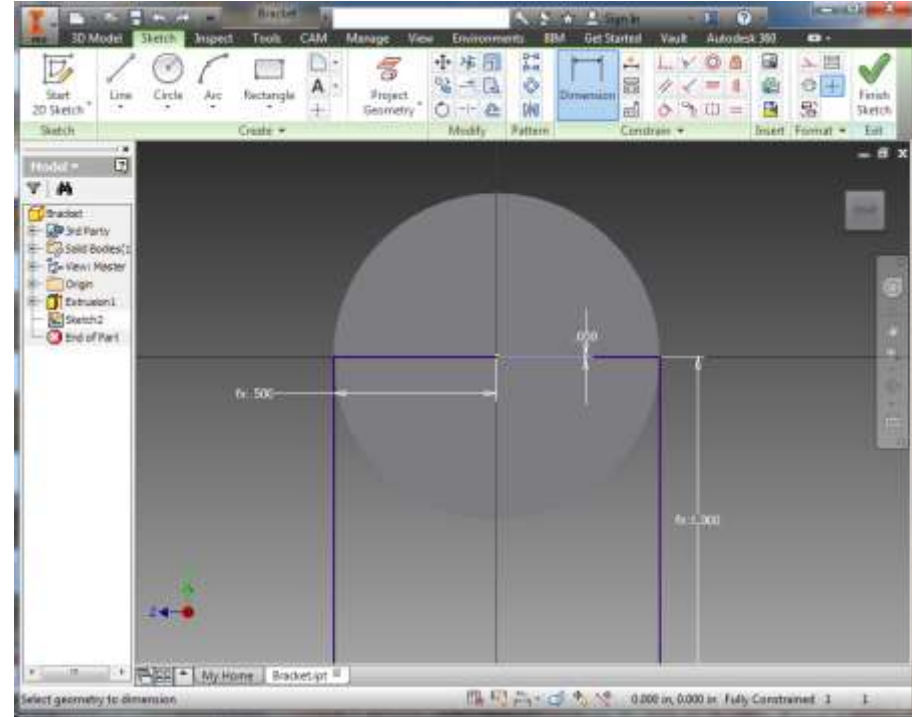
Click on yellow origin

Drag mouse right and click

Type 0 into dialog

Press Enter

Click Finish Sketch



Extrude

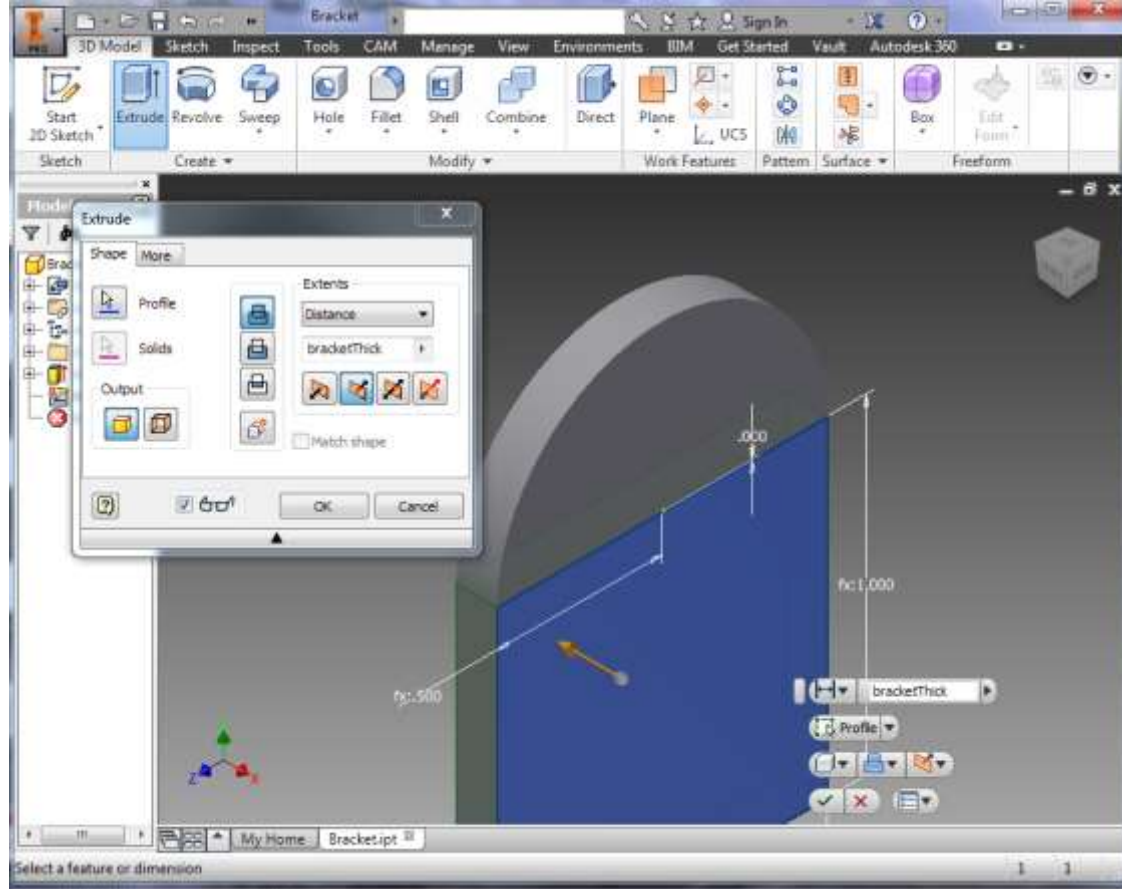
Click on the Extrude button

In the Extrude dialog type in 'bracketThick' under the Distance dropdown

Select the second button for the extrusion direction and the top button for the join

Match the selections in the picture

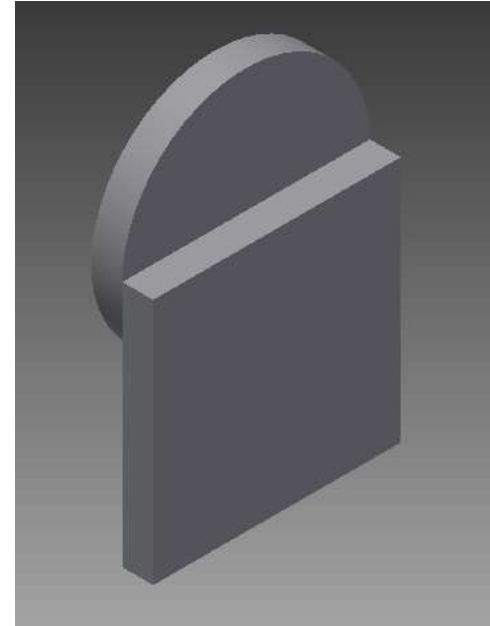
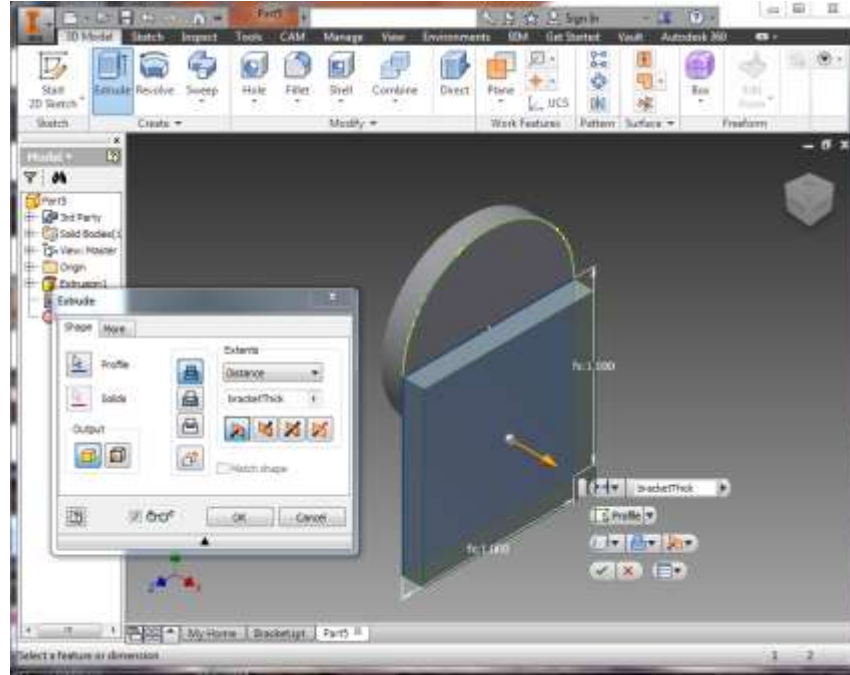
Click OK in the dialog



Join Outward Extrusion

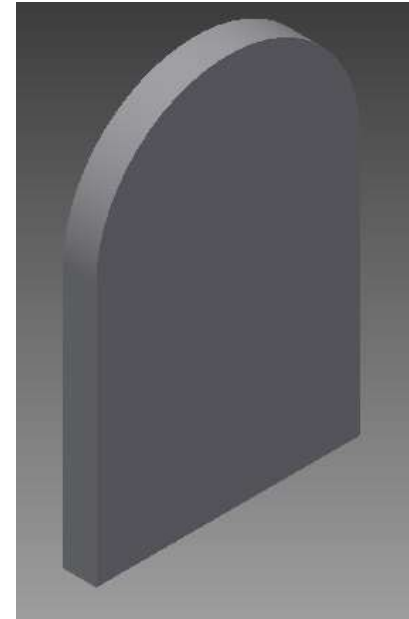
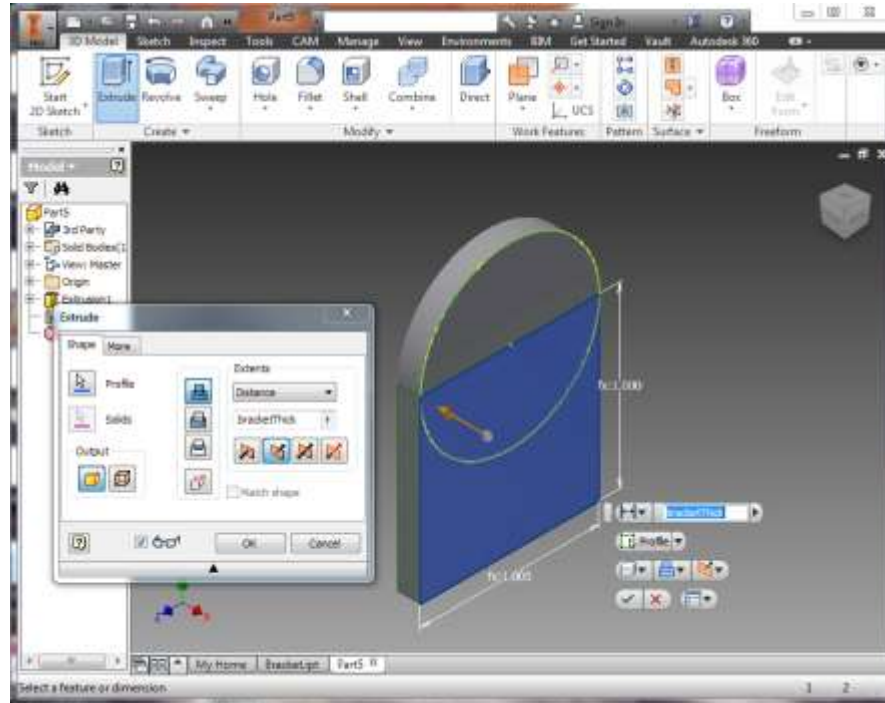
Selecting different Extrusion options results in different shapes

Select Join and the Outward Direction results in a this shape



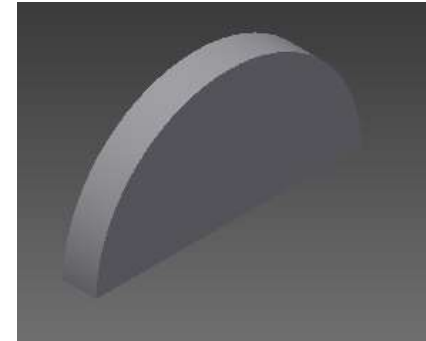
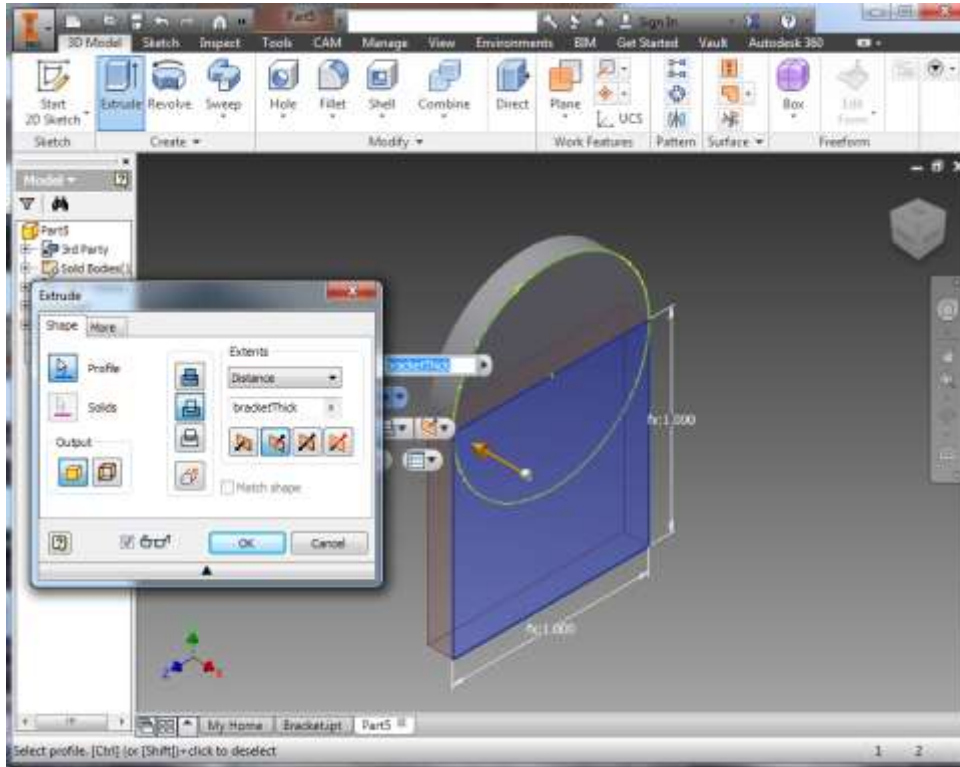
Join Inward Extrusion

Select Join and the
Inward Direction
results in a this
shape



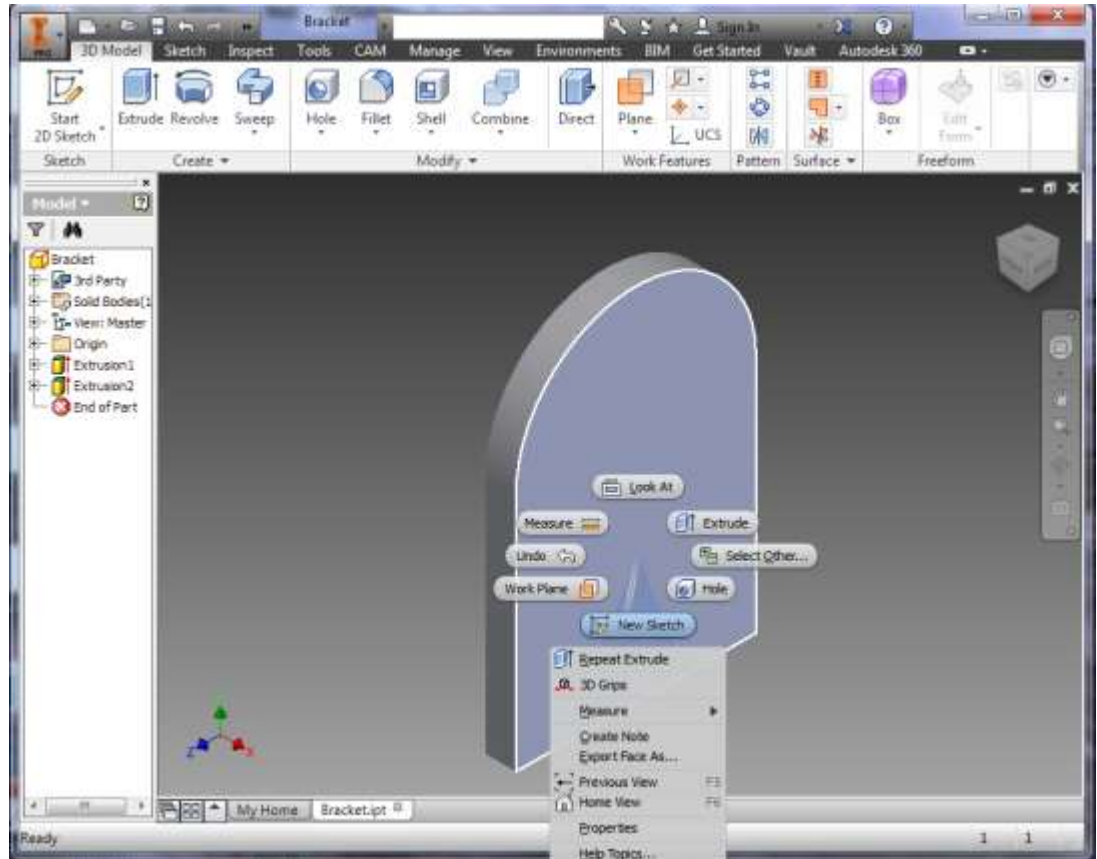
Cut Inward Extrusion

Select Cut and the Inward Direction results in a this shape



Create Sketch

Right Click on the flat face of the part and select New Sketch



Draw Rectangle

Select the Rectangle Tool and click on the sketch below and to the left of the center of the disc

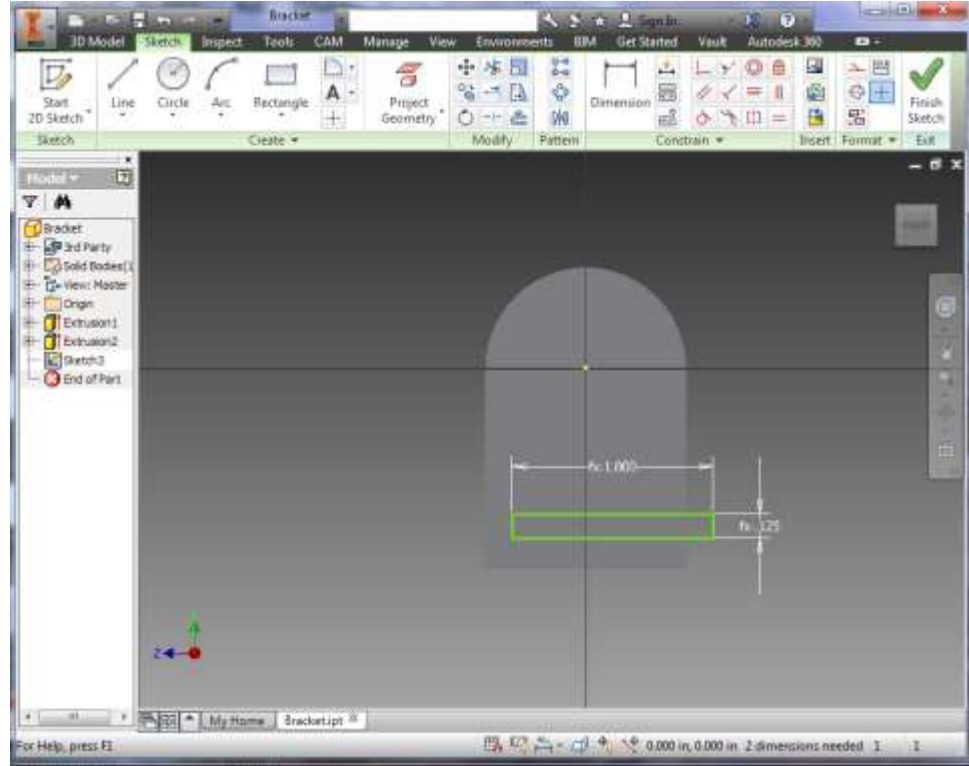
Drag mouse down and right

Type ' $\text{axleDiameter}+2*\text{axleSide}$ '

Press the Tab key

Type ' bracketThick '

Press Enter key



Constrain Horizontal Location

Select Dimension Tool

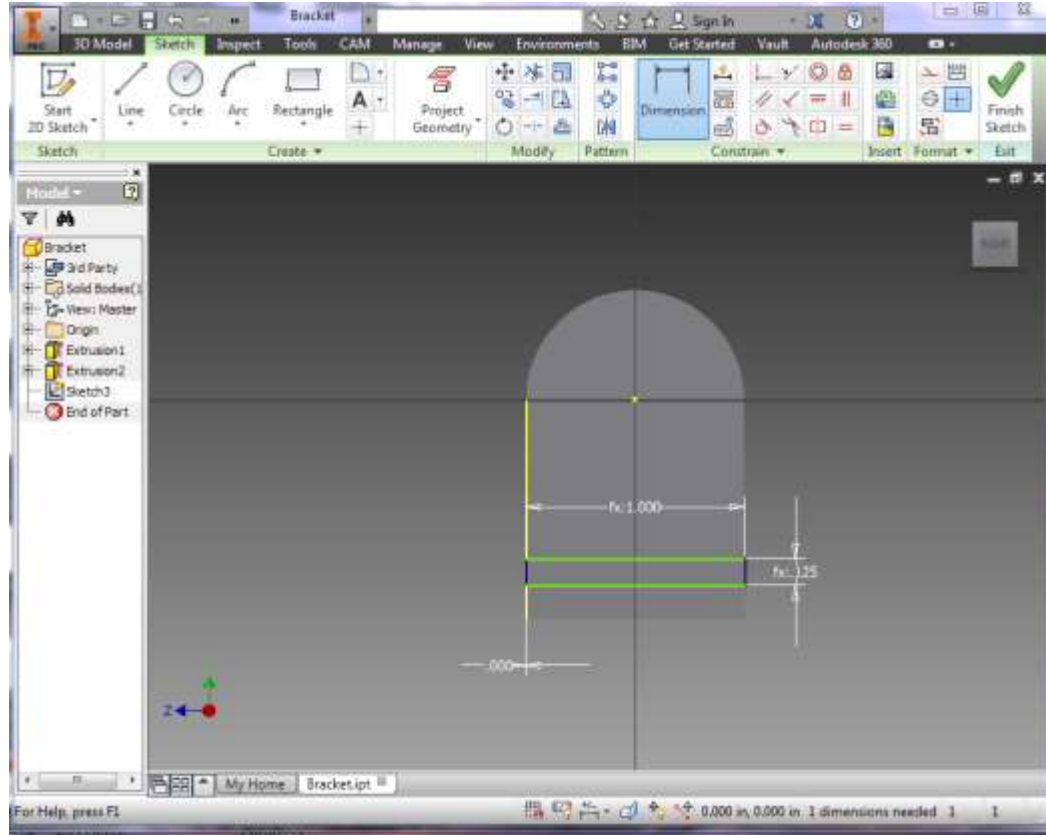
Click on left side of rectangle

Click on left side of part

Drag mouse down and click

Type '0' into dialog

Press Enter



Constrain Vertical Location

Select Dimension Tool

Click on bottom of part

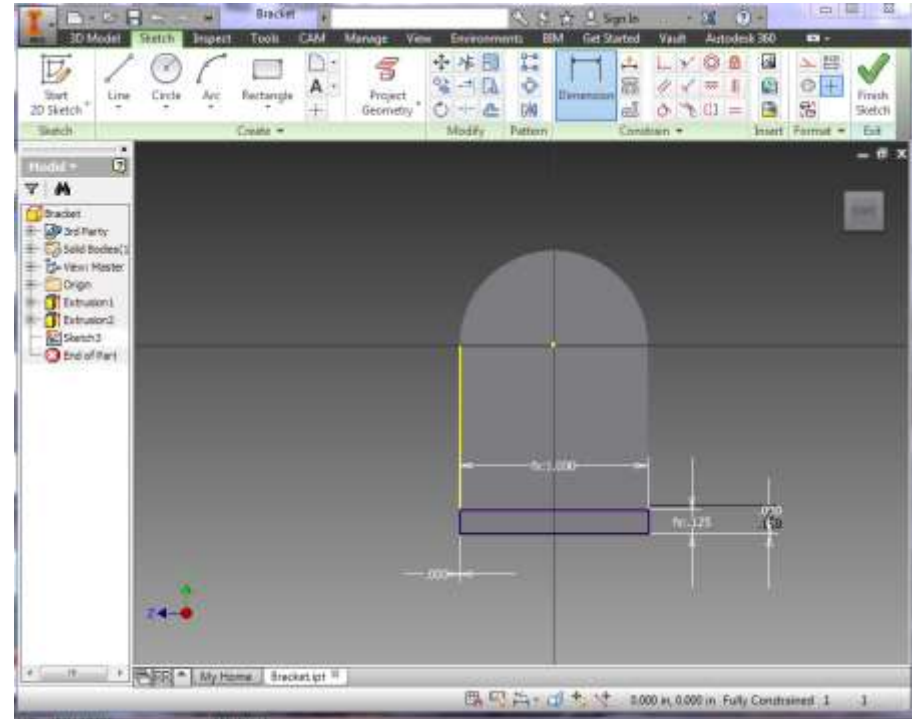
Click on bottom of rectangle

Drag mouse right and click

Type '0' into dialog

Press Enter

Press Finish Sketch



Extrude

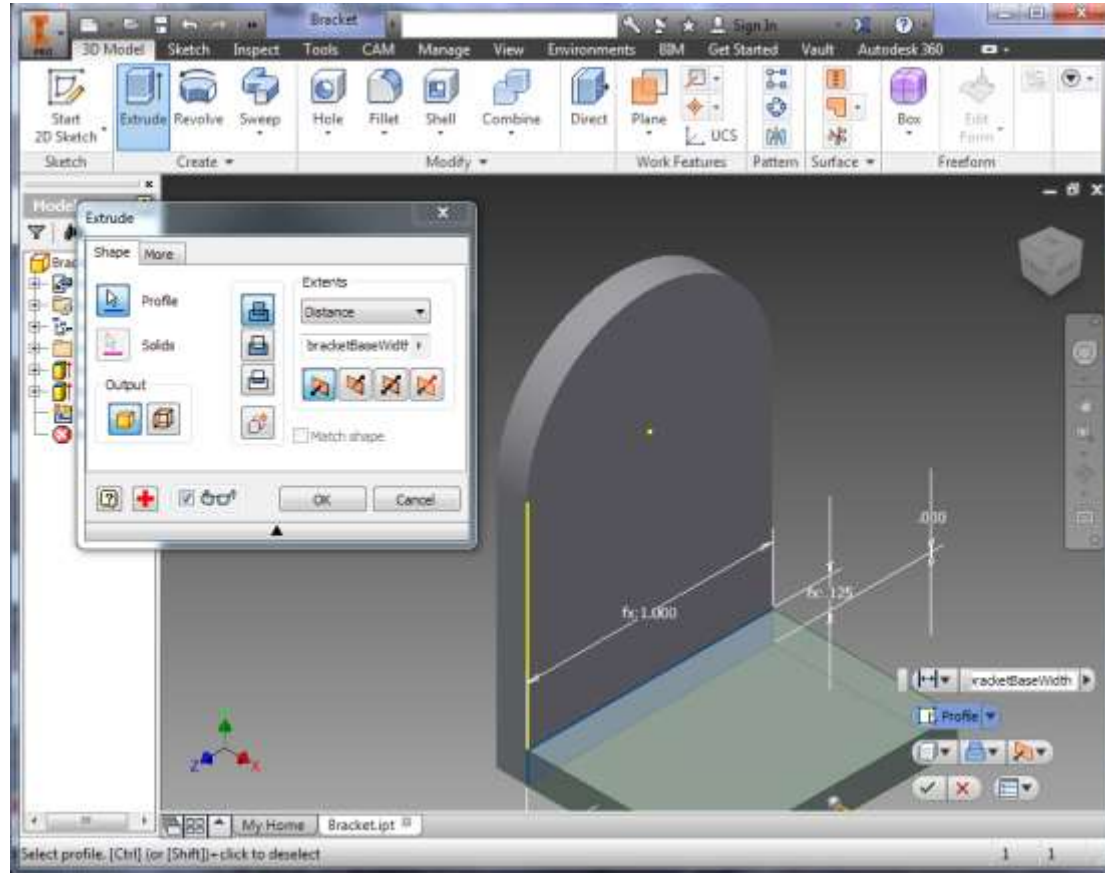
Click on the Extrude button

Click on the rectangle in the sketch

In the Extrude dialog type in 'bracketBaseWidth' under the Distance dropdown

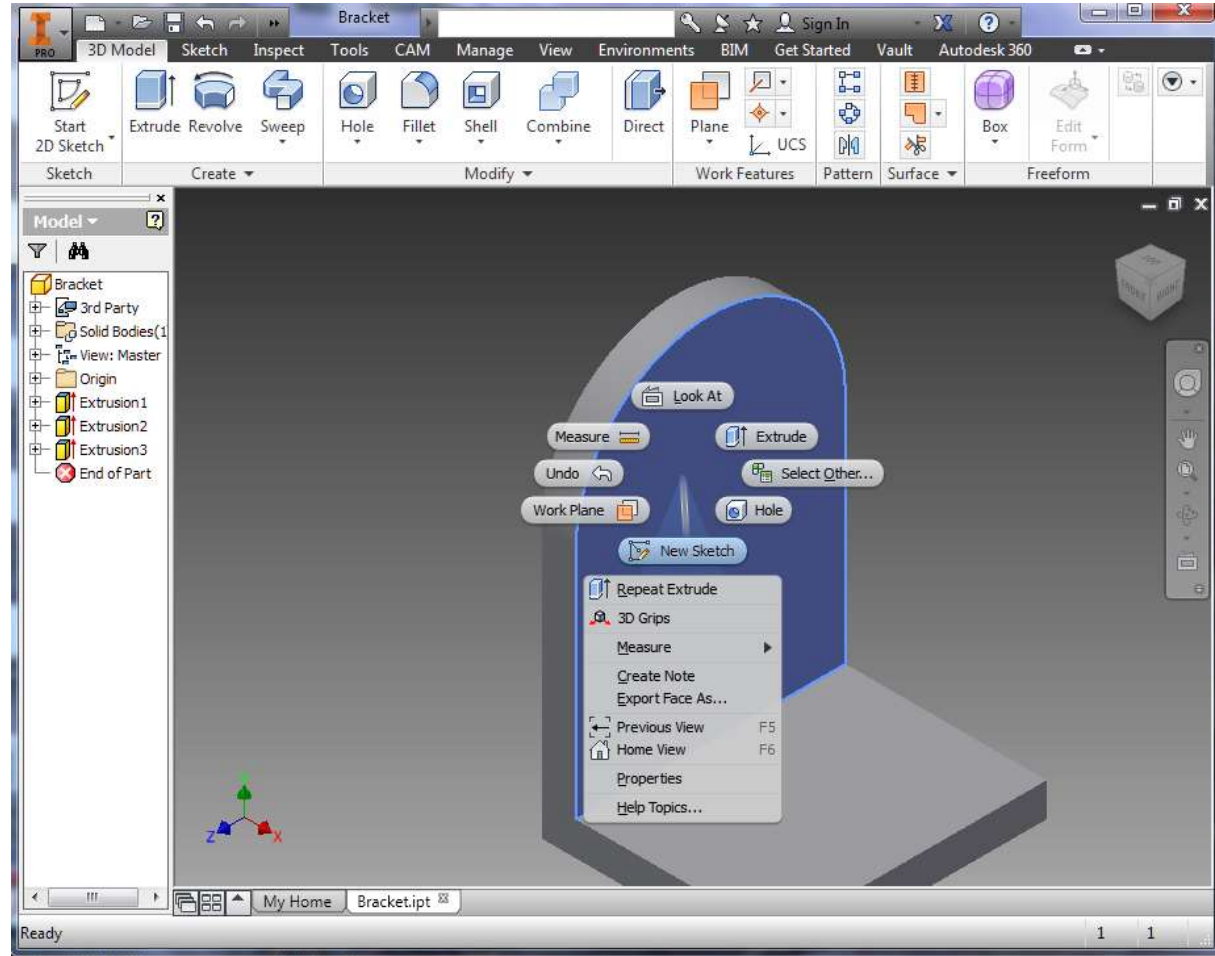
Match the selections in the picture

Click OK in the dialog



Create Sketch

Right Click on the flat face of the part and select New Sketch



Draw Circle

Click the Circle Button

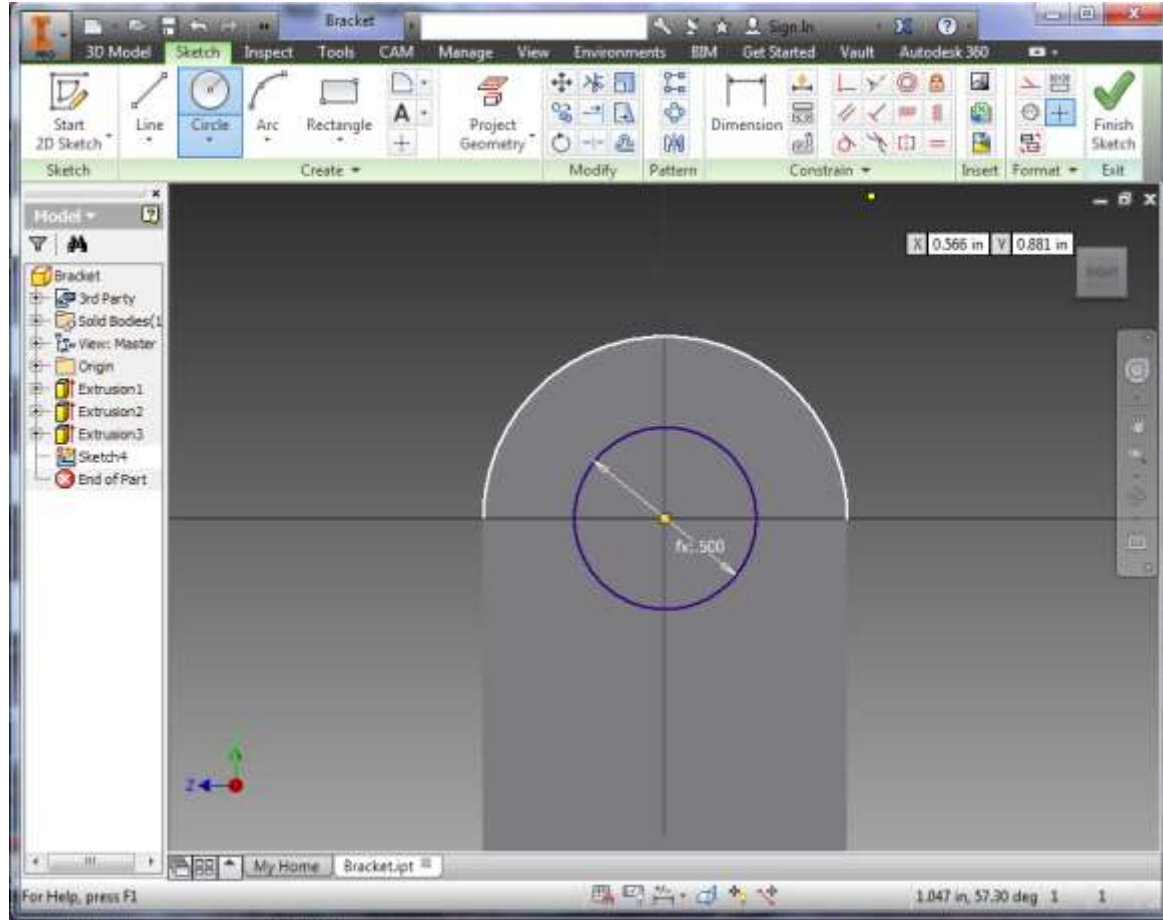
Click on the yellow origin dot

Drag mouse right

Type 'axleDiameter'

Press Enter key

Click Finish Sketch



Extrude

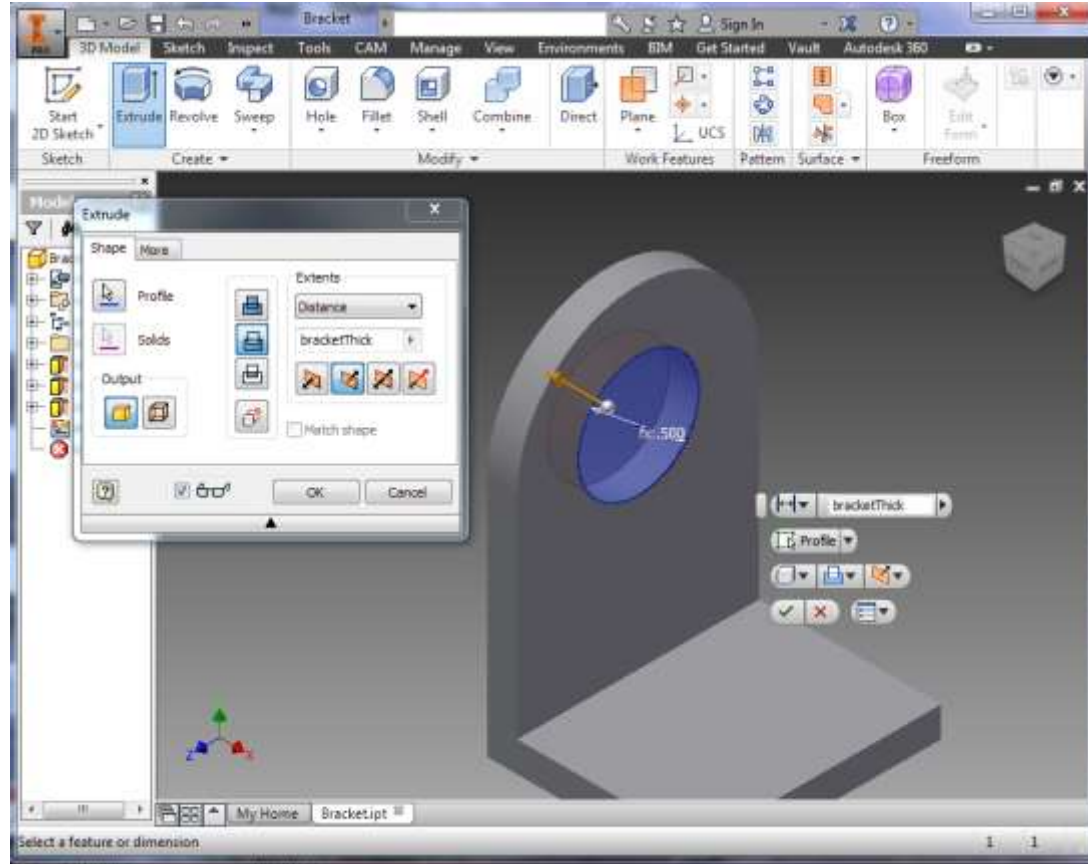
Click on the Extrude button

Click on the circle in the sketch

In the Extrude dialog type in 'bracketThick' under the Distance dropdown

Match the selections in the picture

Click OK in the dialog



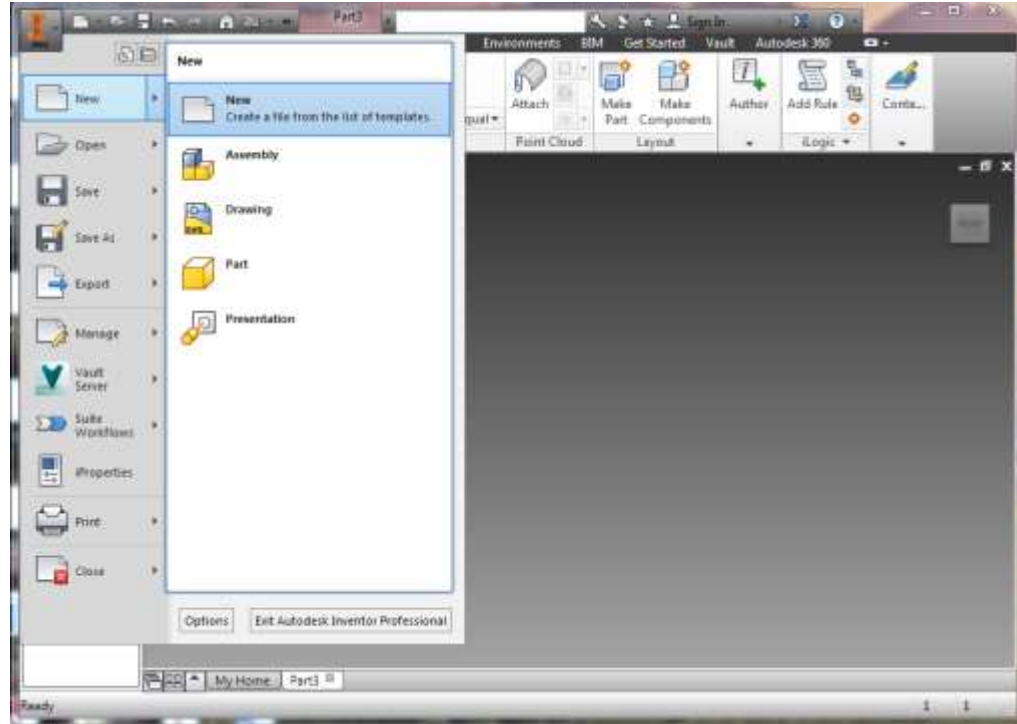
Creating An Assembly

In the I menu (top left) click New and select New

In the left window of the dialog click on English

In the middle double click on Standard (in).iam

Click on the I in the top left to Save the assembly as “Robot”



Placing the Body Part

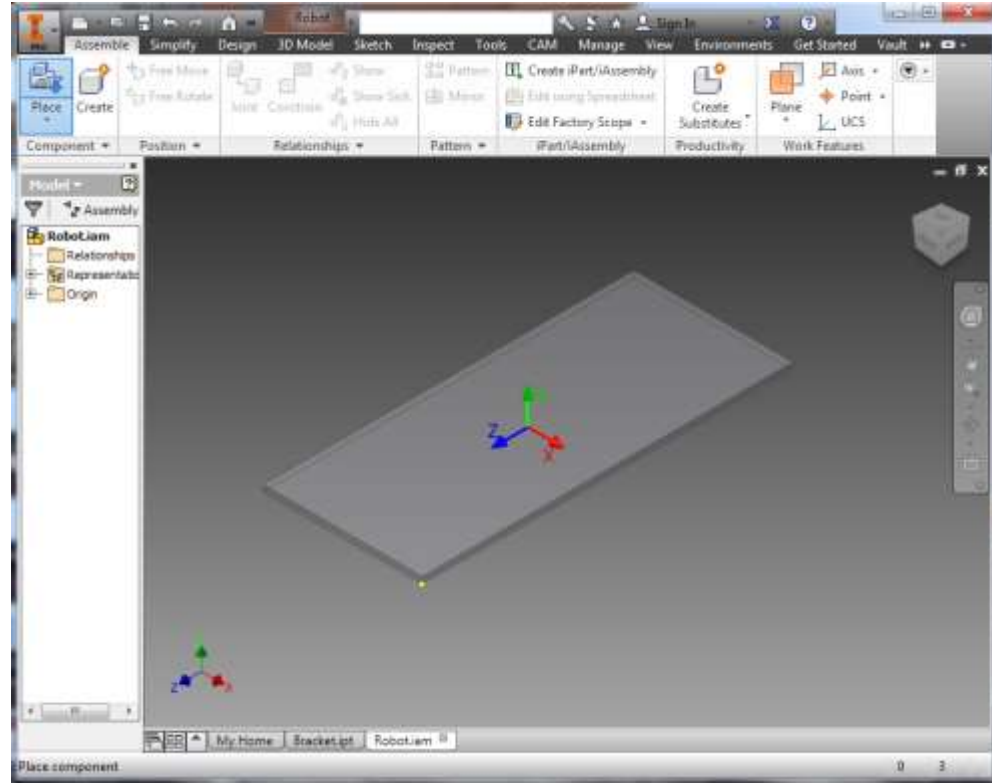
Clicking on the house icon to get the home view

Click on the Place Button

Select the Body.ipt file from the dialog

Click Open

Click on the center of the assembly window



Placing the Bracket

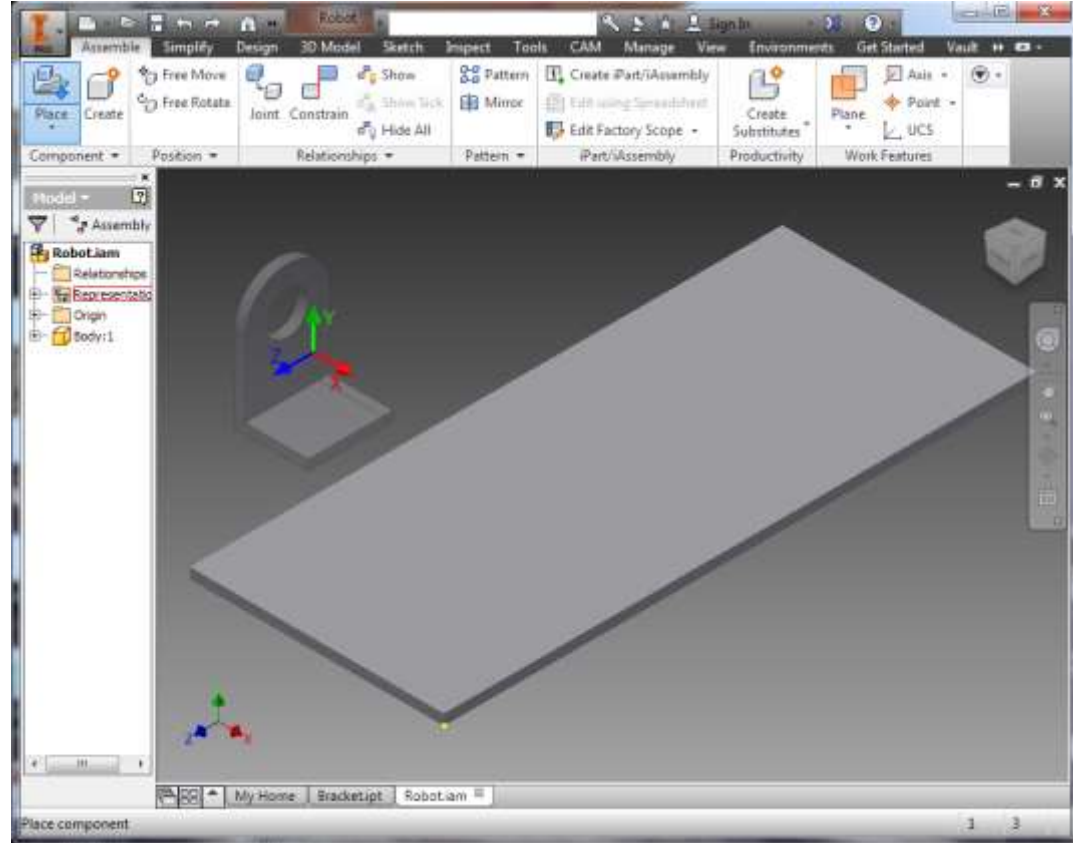
Click on the Place Button

Select the Bracket.ipt file from the dialog

Click Open

Position the bracket as shown in the picture

Click on the assembly window



Moving Around

Zoom - rotate the mouse wheel to zoom in and out based on cursor position in the view window

Rotate - while holding shift key + mouse wheel click, move mouse to rotate part around the origin

Shift - while holding ctrl key + mouse wheel click, move mouse to shift the part in the view

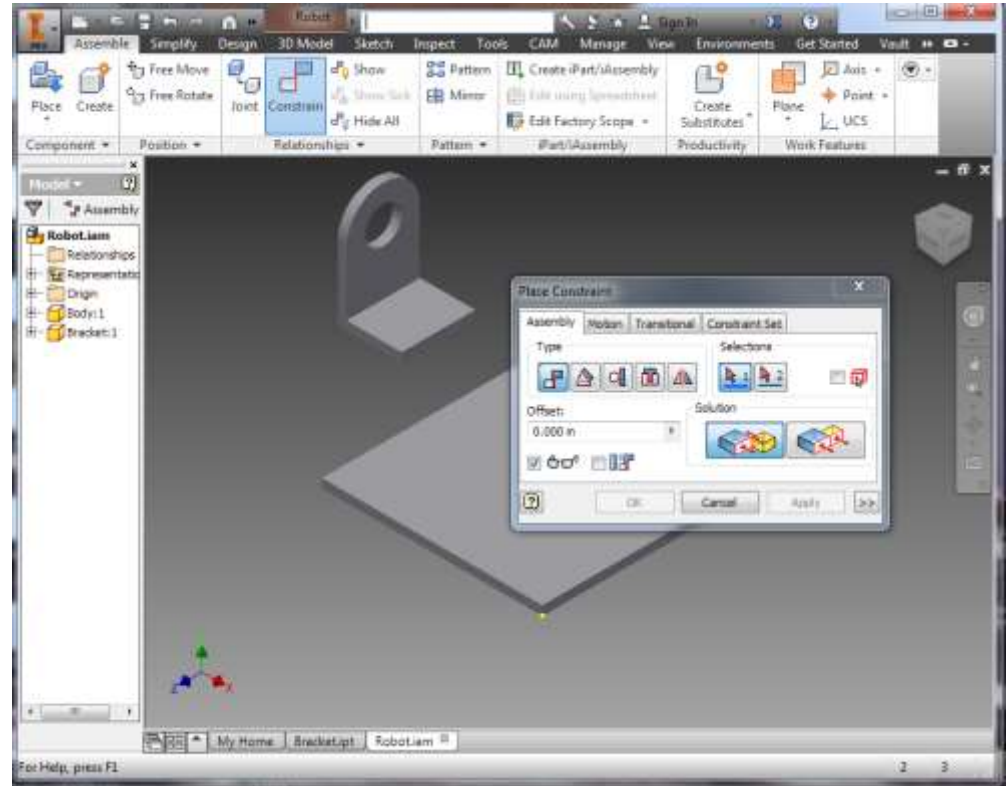
Constraint Types

To join parts into assemblies, use constraints

There are two types of constraints-
Mate and Flush

Mate constraints make two
selected planes join face to face

Flush constraints make two
selected planes join side by side



Mate Bottom of Bracket to Top of Body

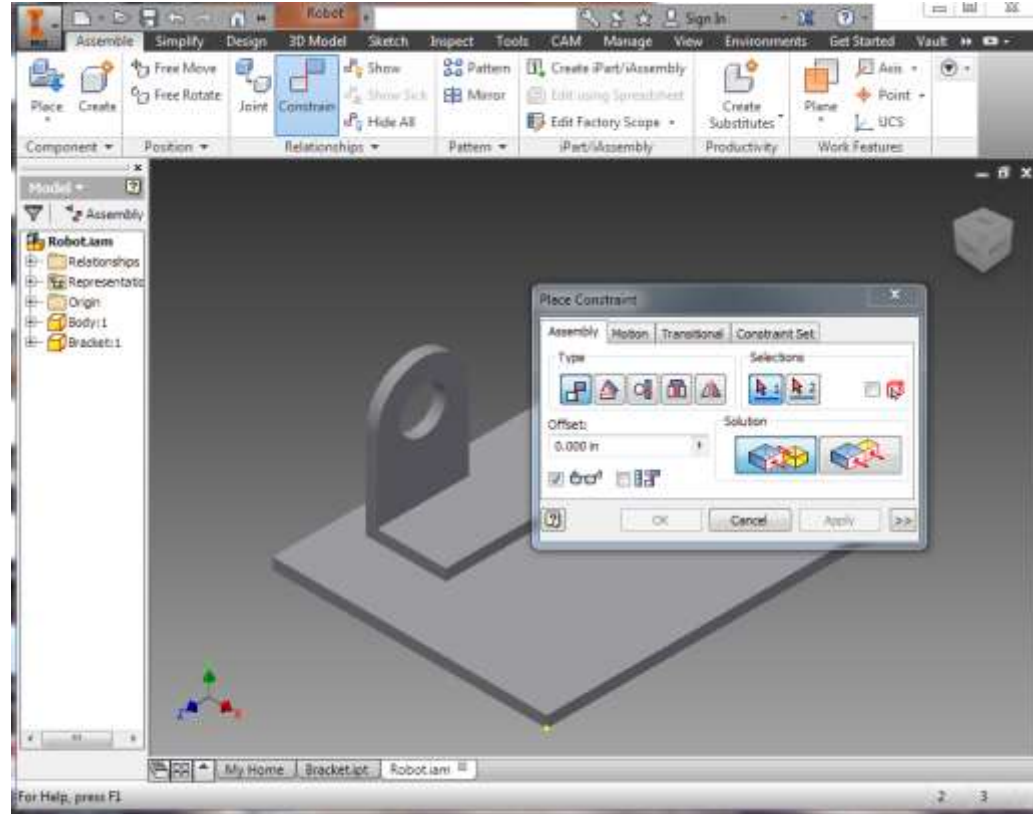
Click on Constrain

Make sure the Mate option (leftmost) is selected in the Solution section

Click on the top of the Body

Rotate parts and click on the bottom of the bracket

Click Apply



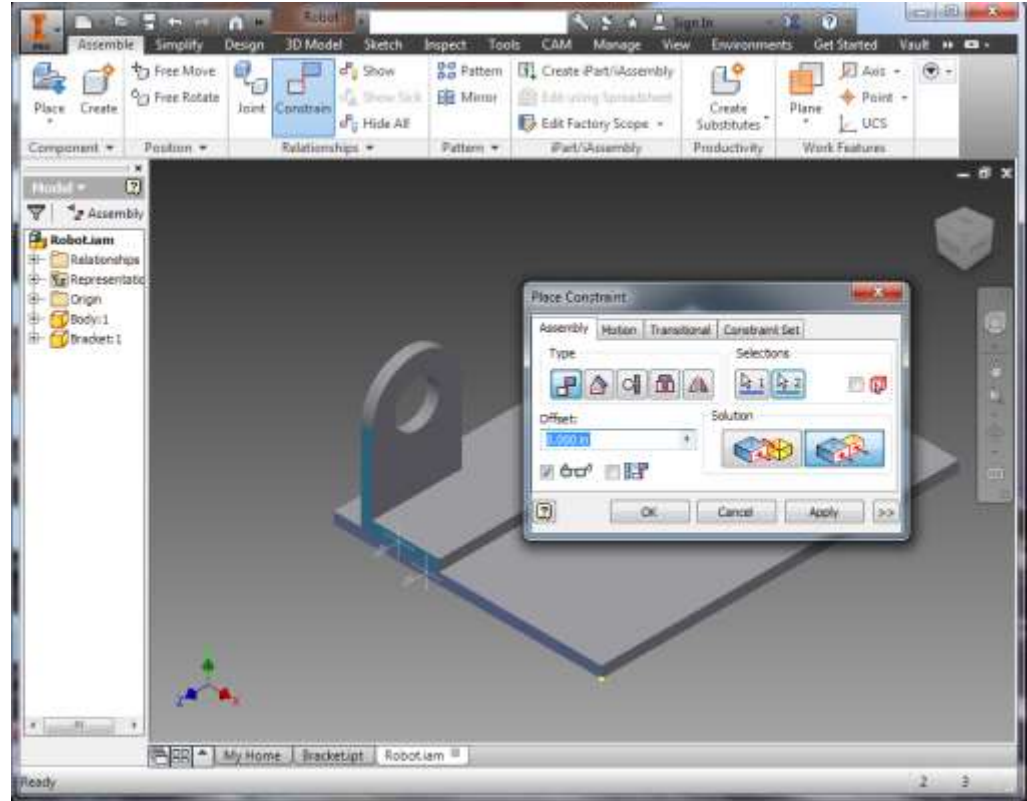
Flush Constrain Bracket Edge To Body Edge

Select the Flush option (rightmost) is selected in the Solution section

Click on the frontmost side (most + plane in Z dir) of the Body

Click on the frontmost side (most + plane in Z dir) of the Bracket

Click Apply



Flush Constrain Bracket Back To Body Left Edge

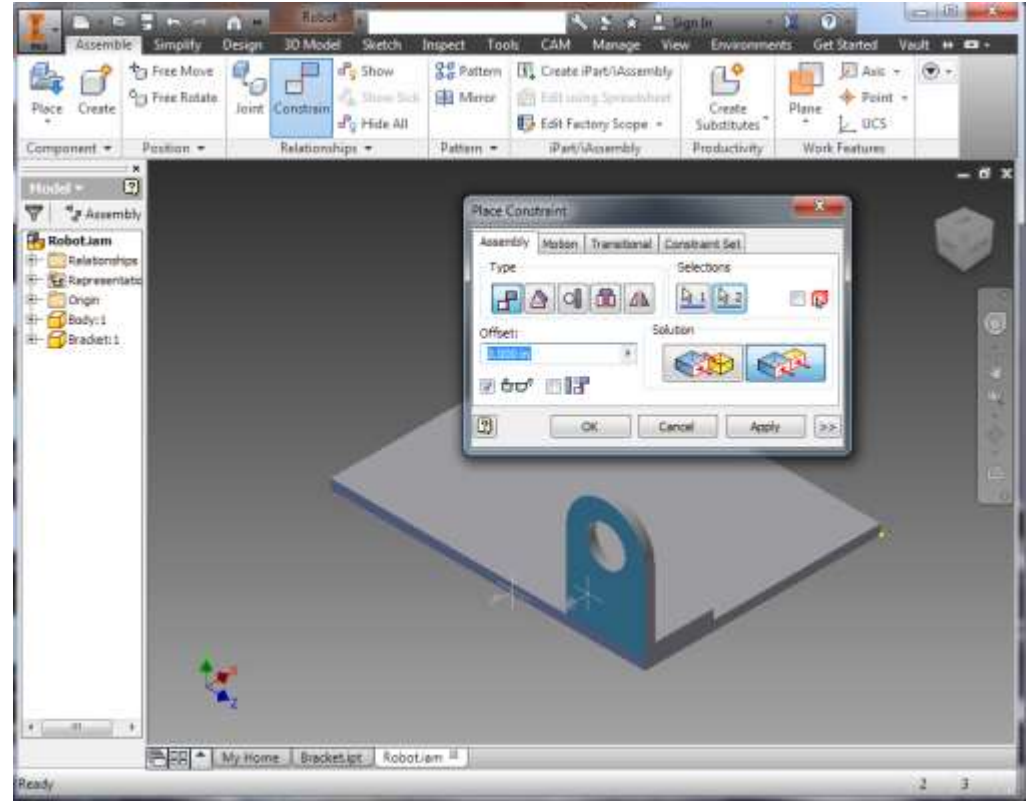
Rotate parts around to see Bracket Back

Click on the leftmost side (most - plane in X dir) of the Body

Click on the Bracket Back (most - plane in X dir) of the Bracket

Click Apply

Click Cancel



Add Multiple Brackets

Click Place Button

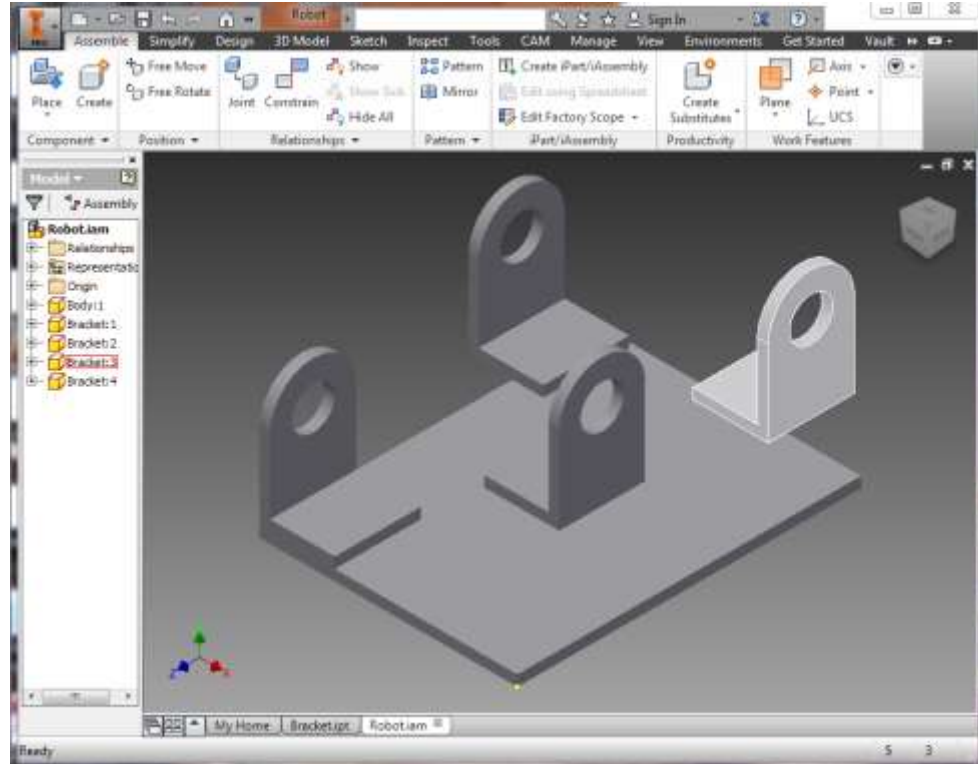
Select Bracket

Click to place Bracket on same side as first bracket

Right click and select Rotate Y 90

Right click and select Rotate Y 90 again

Click to place two more brackets opposite the first bracket



Mate Constrain Bracket Bottoms to Top of Body

Click on Constrain

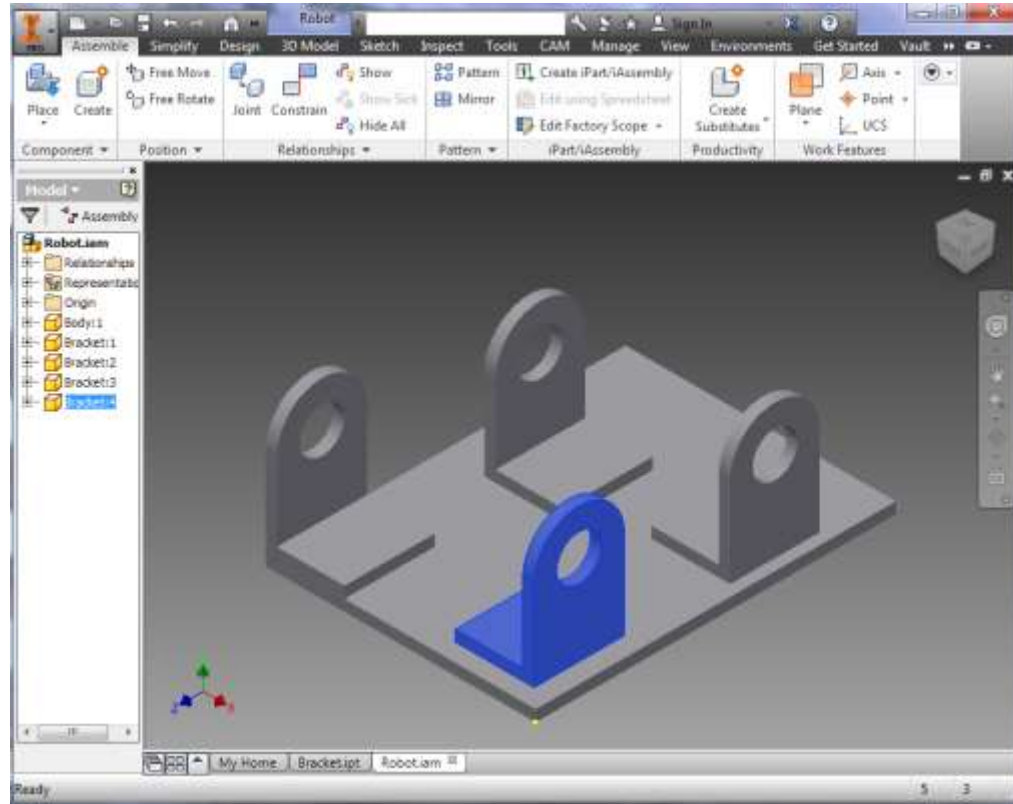
Make sure the Mate option (leftmost) is selected in the Solution section

Click on the top of the Body

Rotate parts and click on the bottom of a bracket

Click Apply

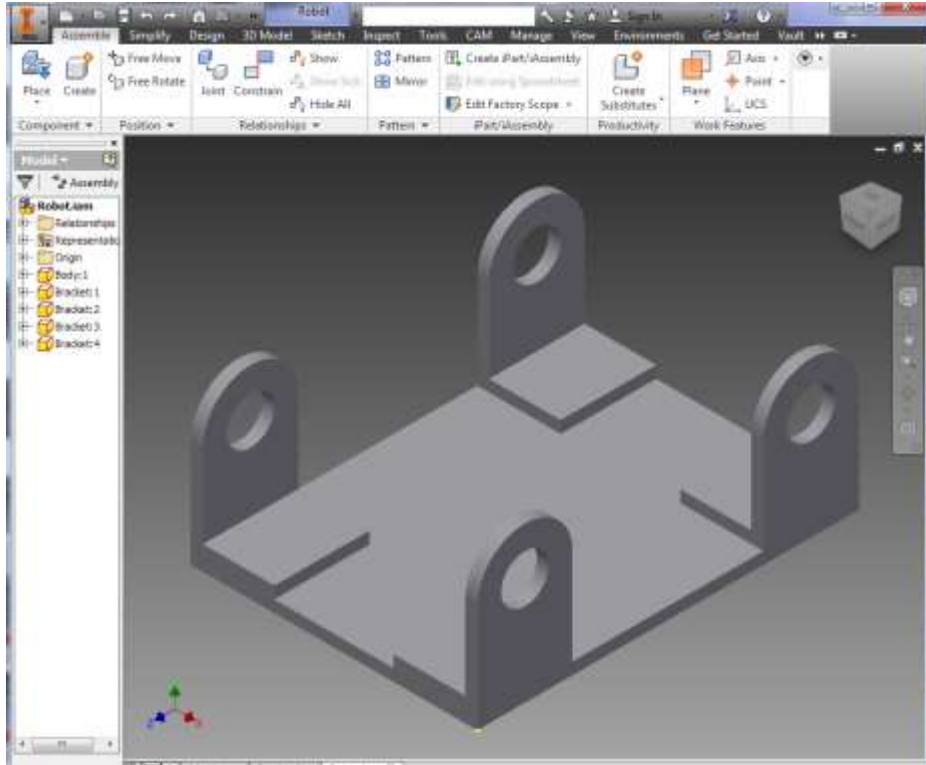
Repeat Steps for all Brackets



Flush Constrain Bracket To Corners of the Body

Select the Flush option (rightmost) is selected in the Solution section

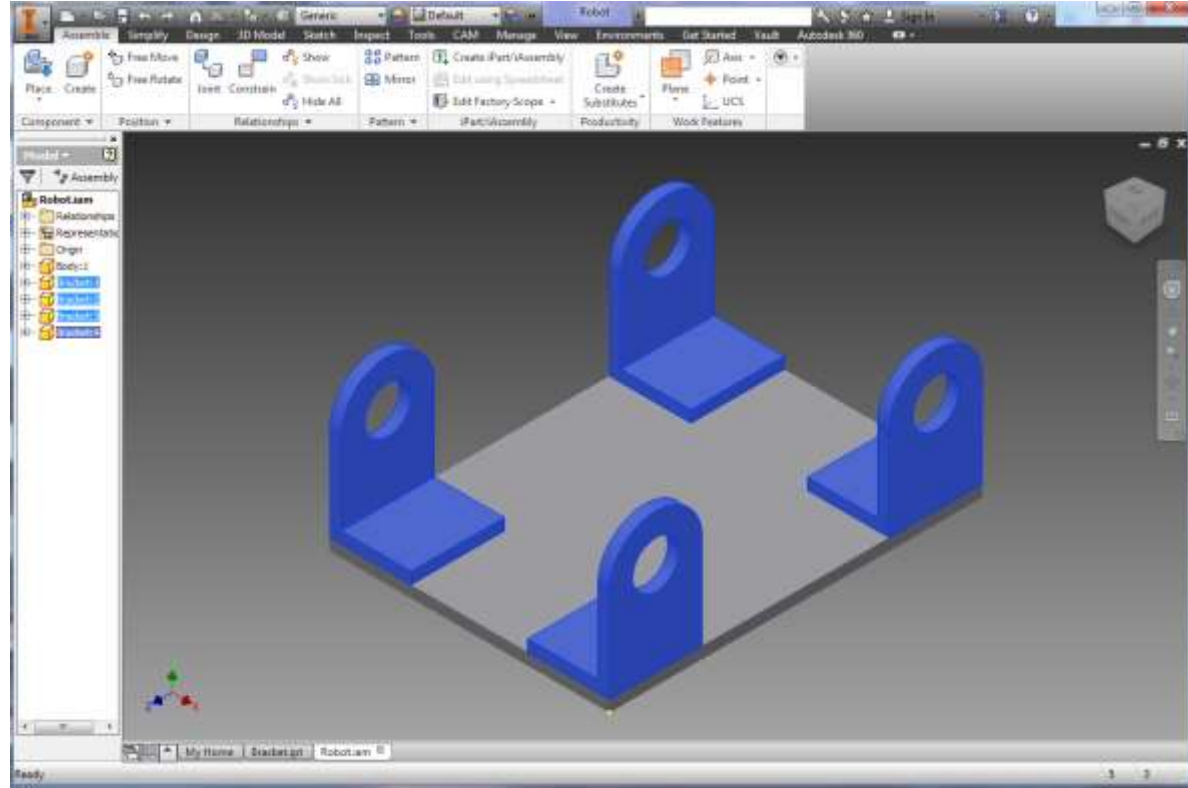
Click on the planes to constrain the brackets to the corners of the Body



Select All Of The Brackets

Hold down the ctrl key

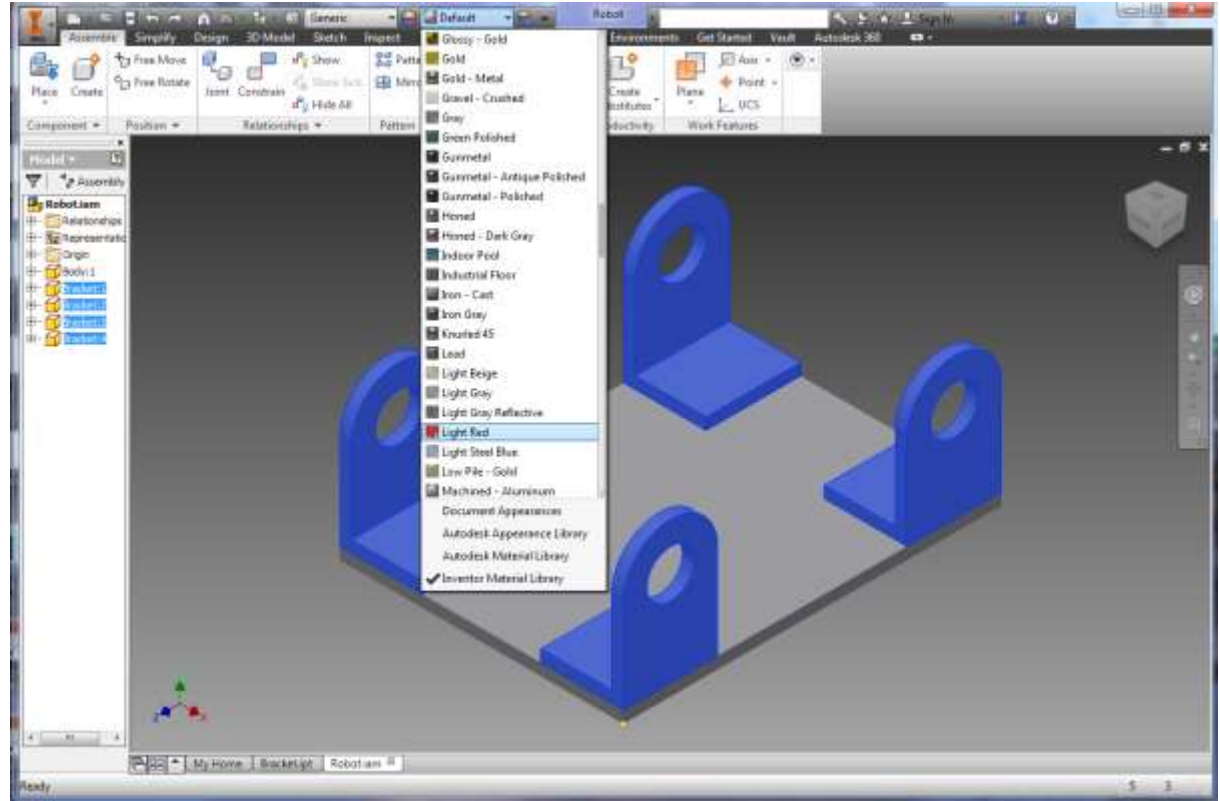
One by one click on all of the Brackets in the Model window on the left



Change Bracket Color

Click on the Material dropdown on the top bar

Select Light Red



Change Parameters

Change length on spreadsheet to 6 in

Change width to 3 in

Change axleDiameter to .25 in

Save spreadsheet

Click on Update Button in top toolbar

