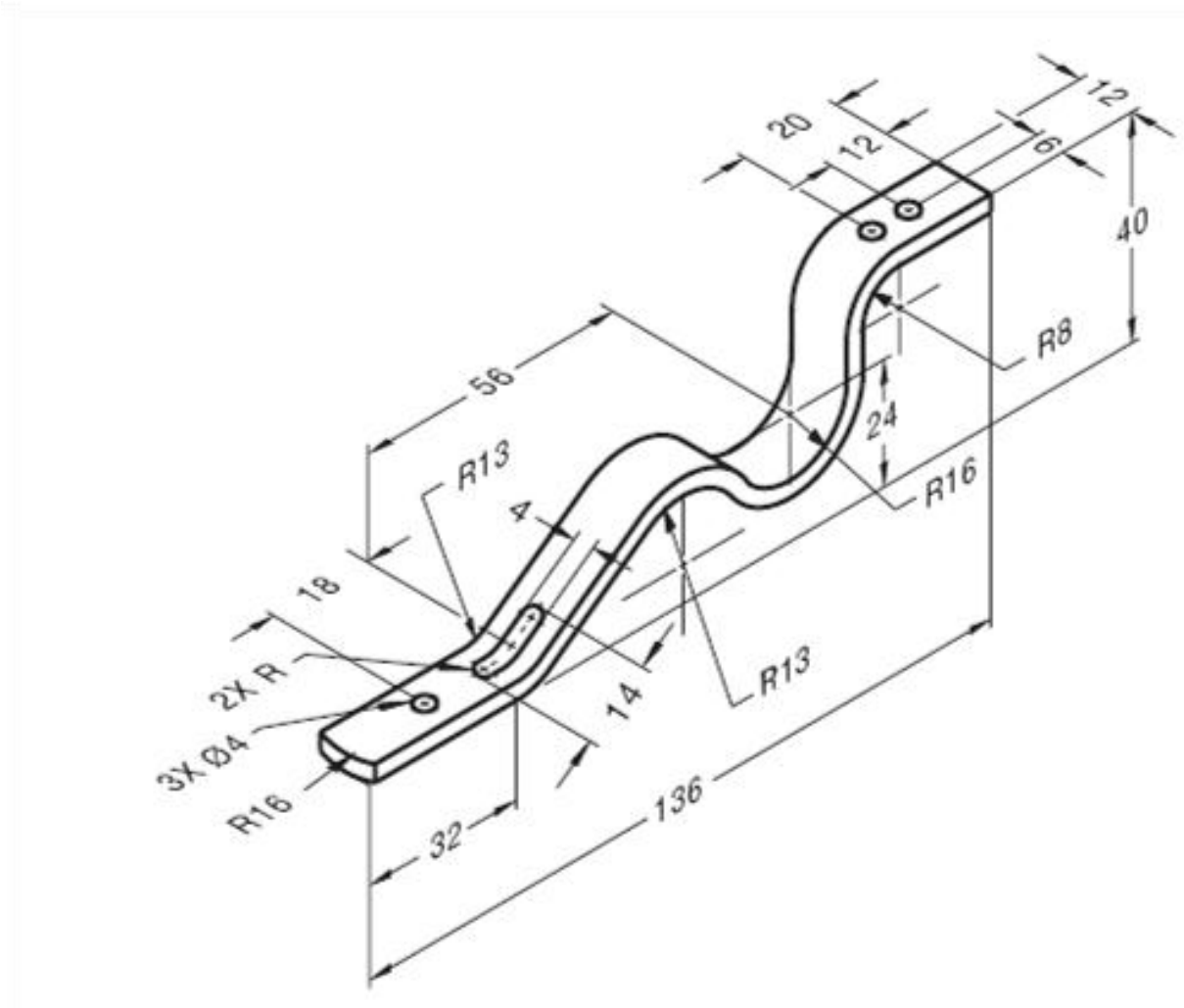
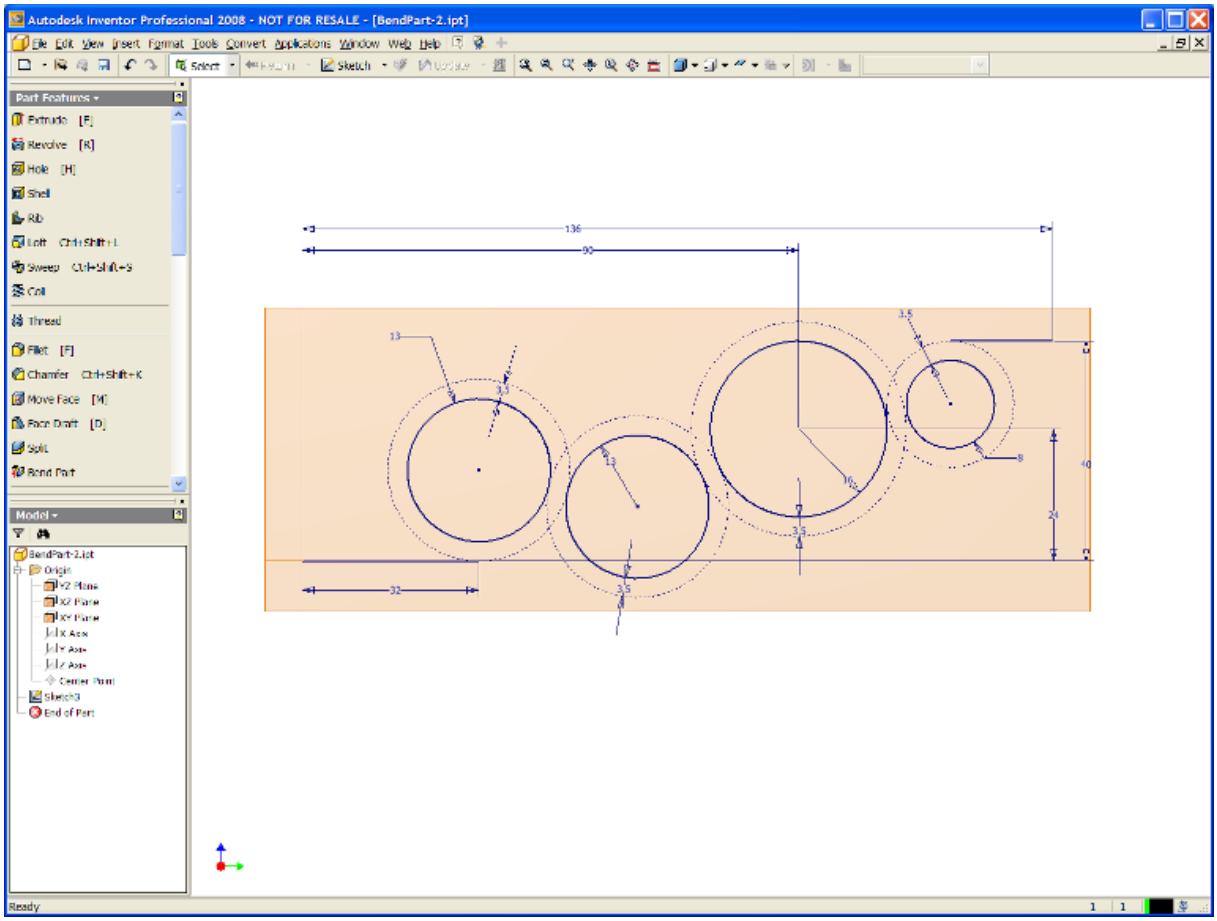
Given:



Desired: create the inventor part model using the bend part feature.

1) Create a Layout sketch: Based on given distances / locations

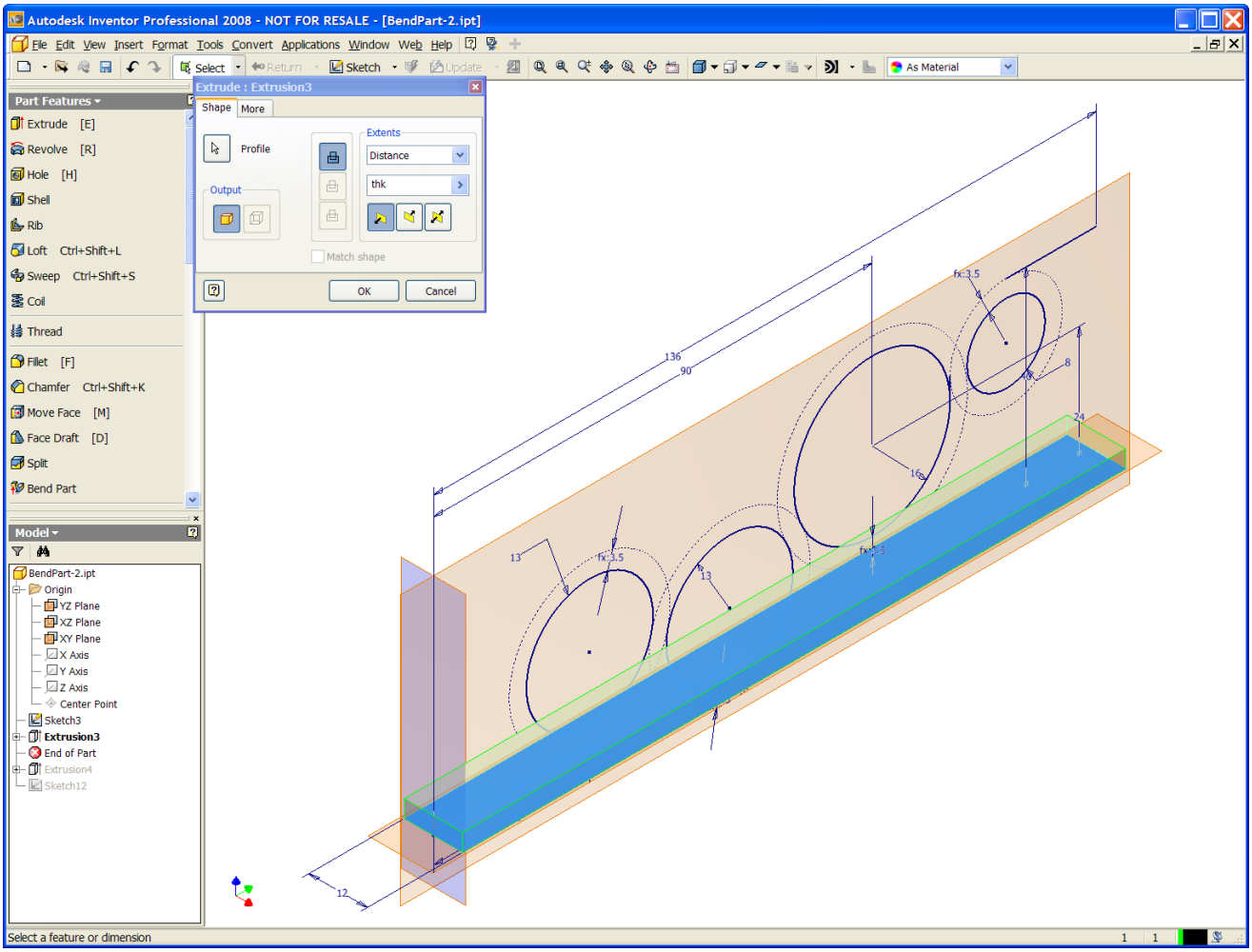
   a. Defined Variable for Material Thickness: Thk = 3.5



Note: Didn't seem to need the indicated straight segment between the left two bends.

2) the base feature

a. The Feature

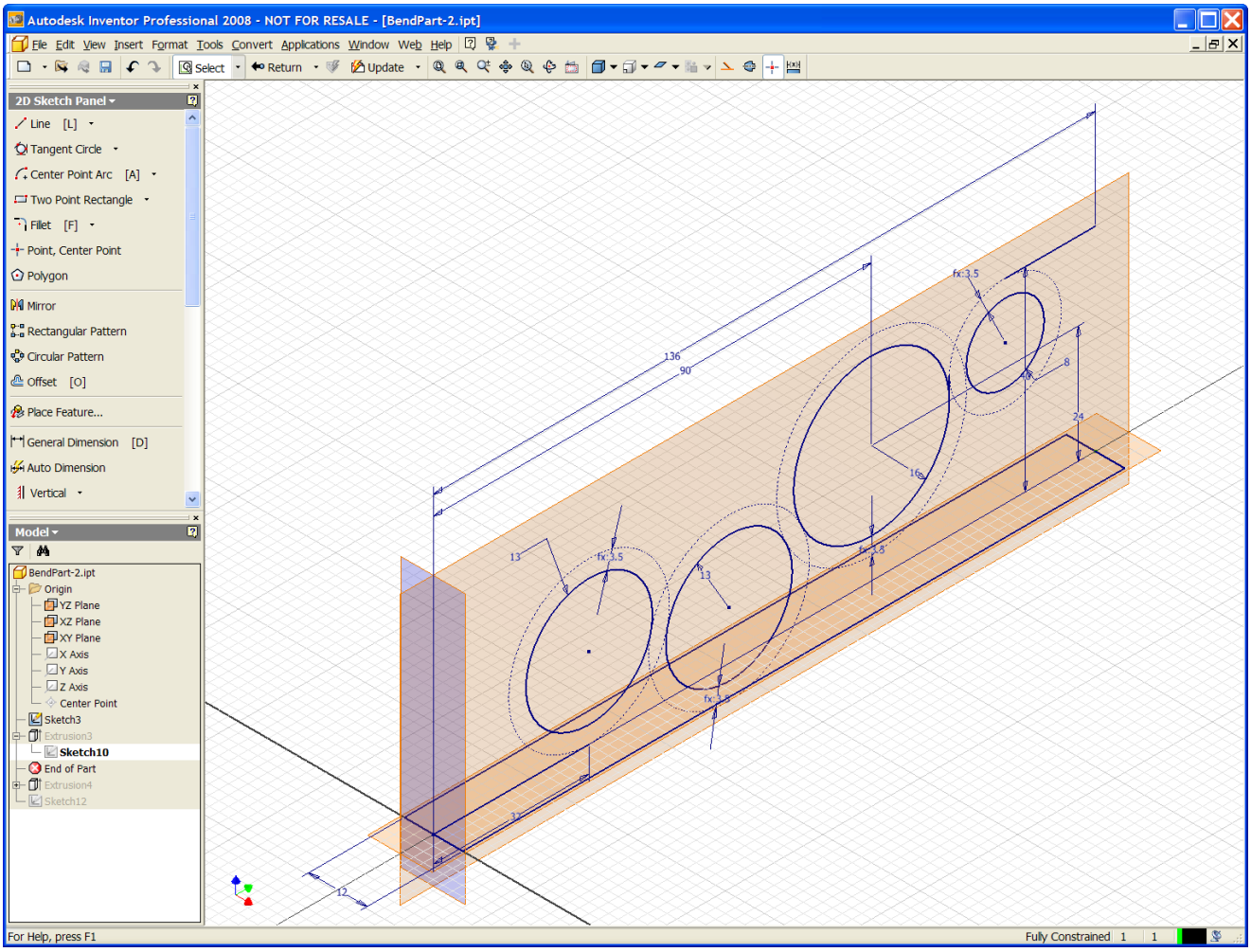


b. The Sketch

i. Project Geometry:

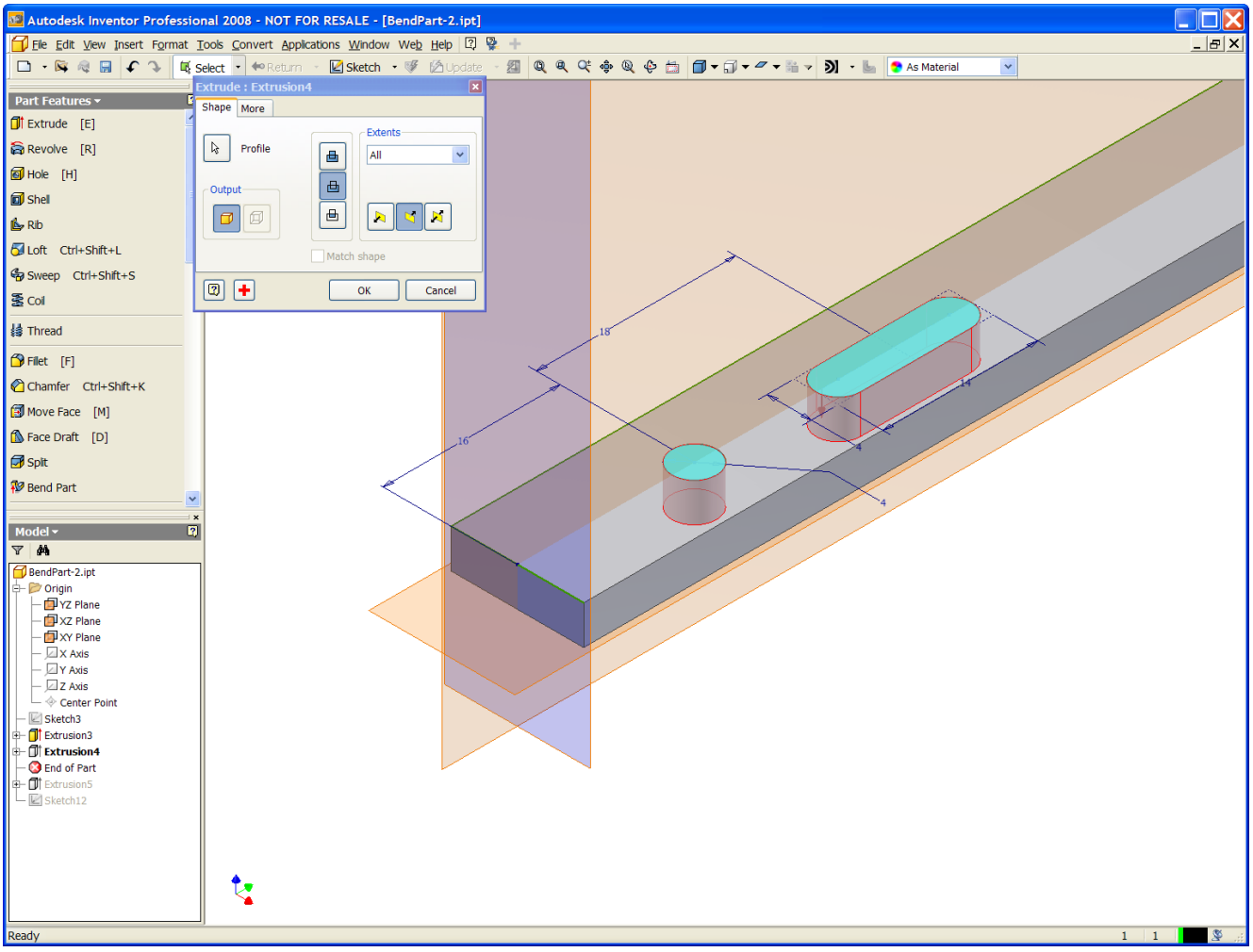
1. Center point Constrained to midpoint of rectangle

2. Endpoint (dim =136) Constrained to far end end of rectangleNe

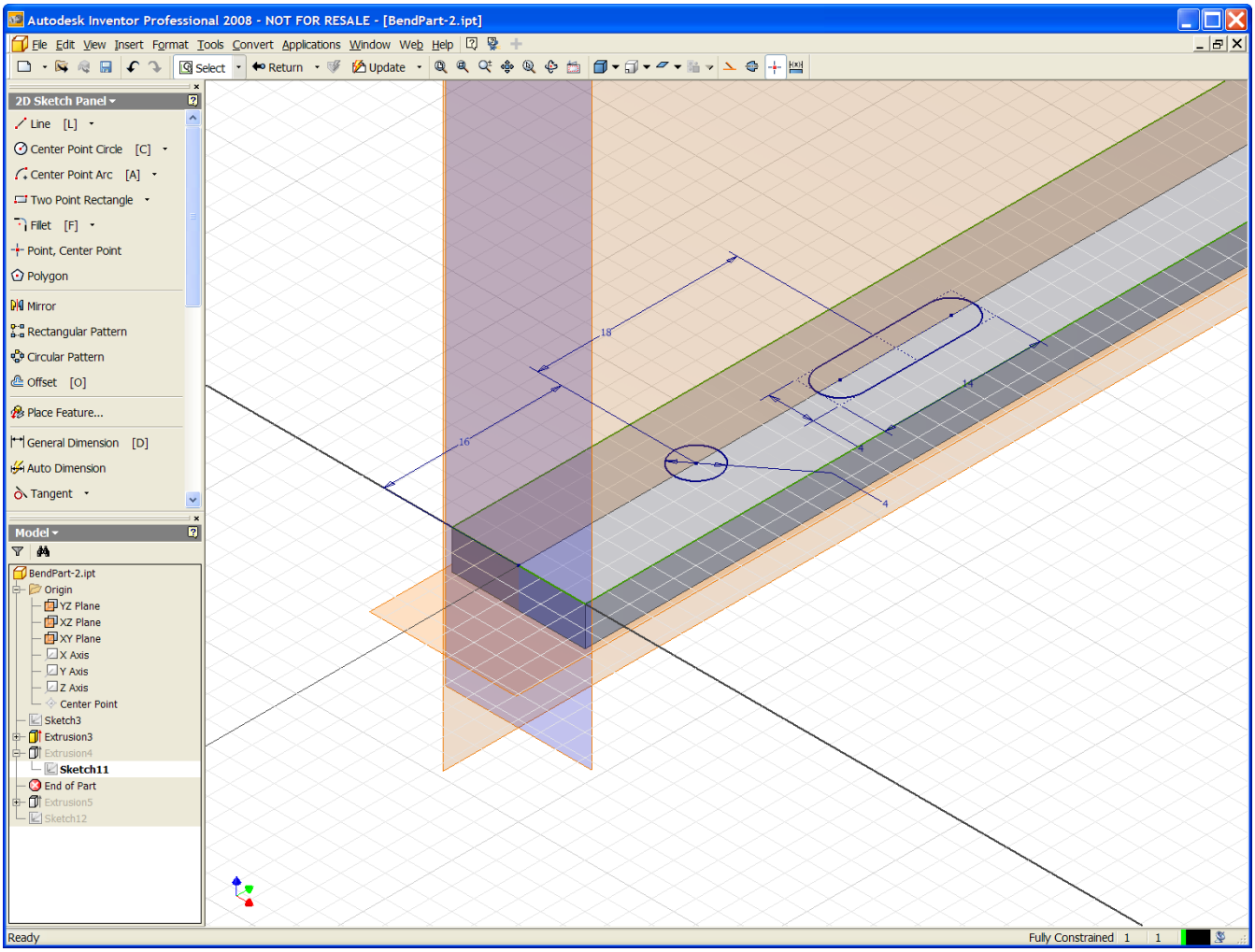


3) Next Feature: Cuts on the left side

a. The feature

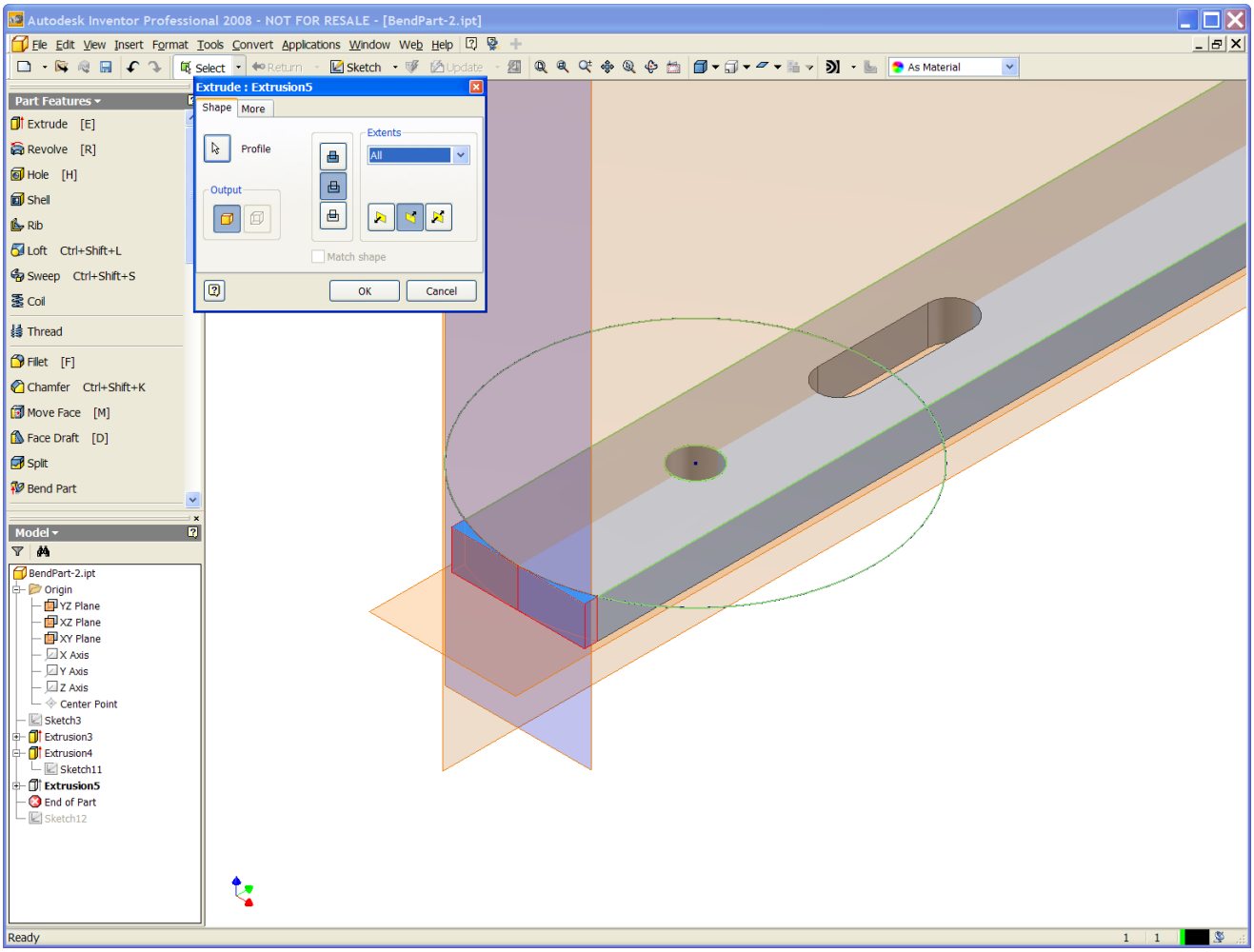


b. The Sketch

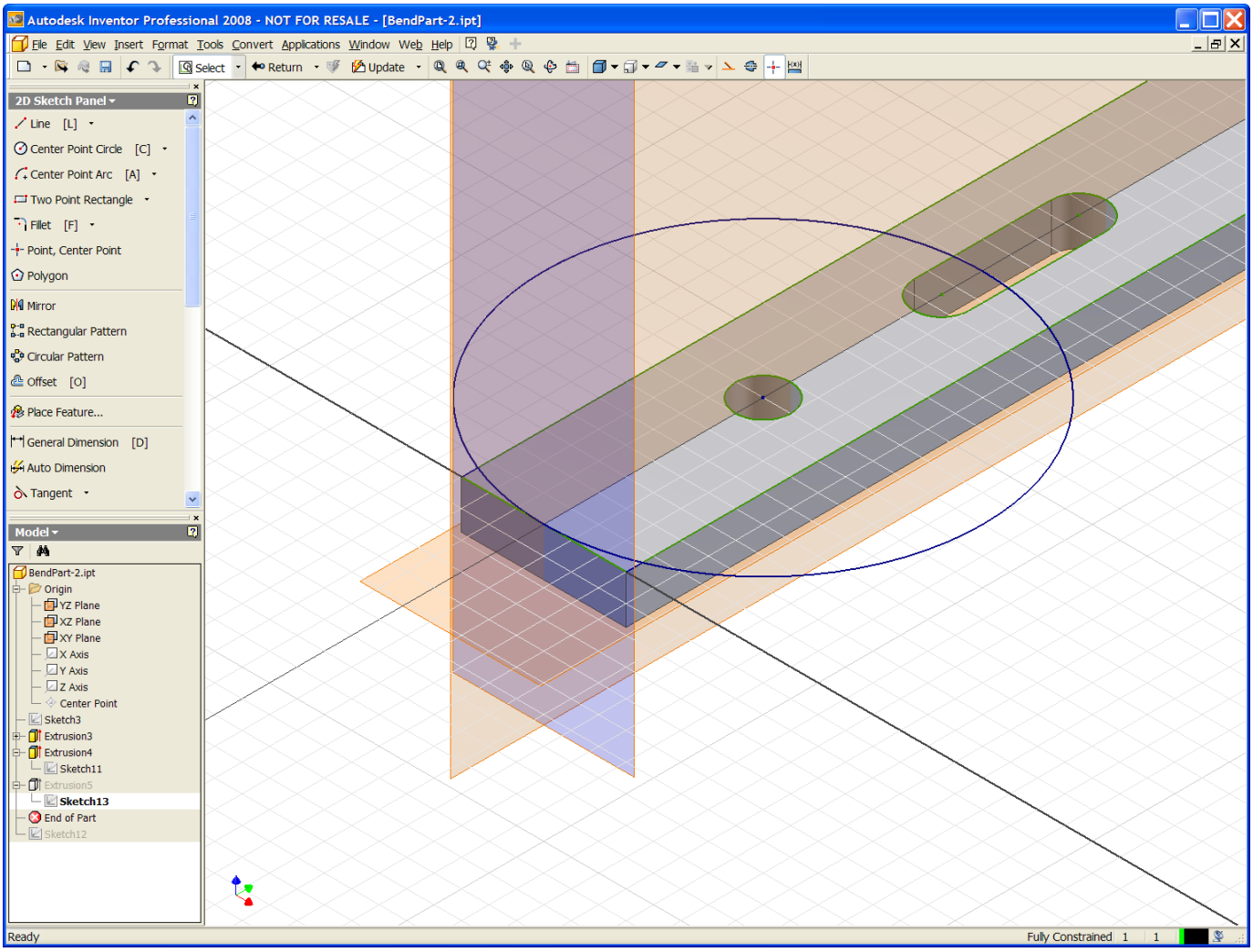

Autodesk Inventor Professional 2008 - NOT FOR RESALE - [BendPart-2.ipt]
File Edit View Insert Format Tools Convert Applications Window Web Help
Select Return Update
2D Sketch Panel
Line [L]
Center Point Circle [C]
Center Point Arc [A]
Two Point Rectangle
Fillet [F]
Point, Center Point
Polygon
Mirror
Rectangular Pattern
Circular Pattern
Offset [O]
Place Feature...
General Dimension [D]
Auto Dimension
Tangent
Model
BendPart-2.ipt
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Sketch3
Extrusion3
Extrusion4
Sketch11
End of Part
Extrusion5
Sketch12
18
16
14
4
4
Ready
Fully Constrained 1 1

4) Next Feature

a. The feature



b. The Sketch


Autodesk Inventor Professional 2008 - NOT FOR RESALE - [BendPart-2.ipt]
File Edit View Insert Format Tools Convert Applications Window Web Help
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Center Point Arc [A]
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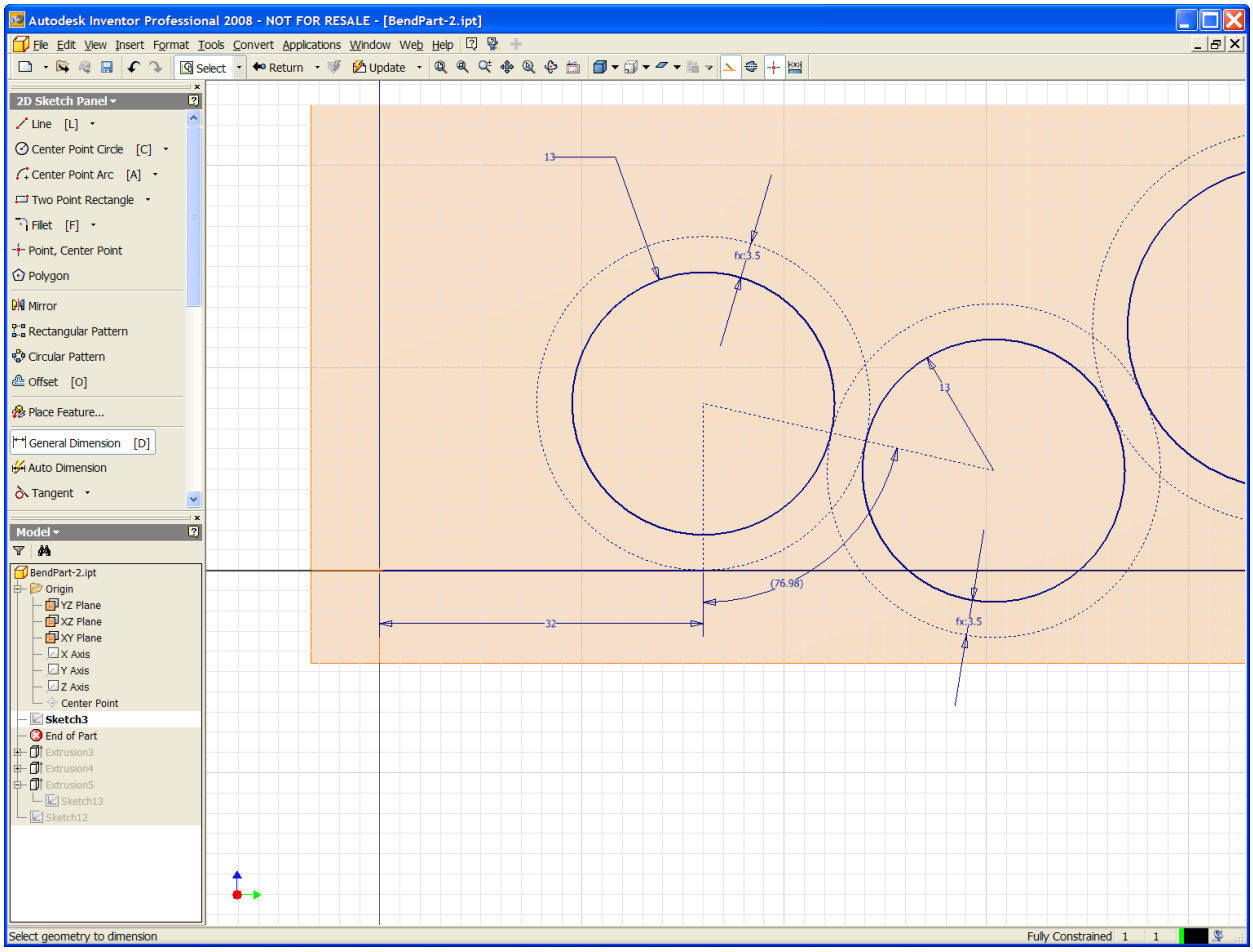
Auto Dimension
Tangent
Model
BendPart-2.ipt
Origin
YZ Plane
XZ Plane
XY Plane
X Axis
Y Axis
Z Axis
Center Point
Sketch3
Extrusion3
Extrusion4
Sketch11
Extrusion5
Sketch13
End of Part
Sketch12
Ready
Fully Constrained 1 1

5) Next Feature: The first bend

a. Determine the bend geometry on the layout sketch

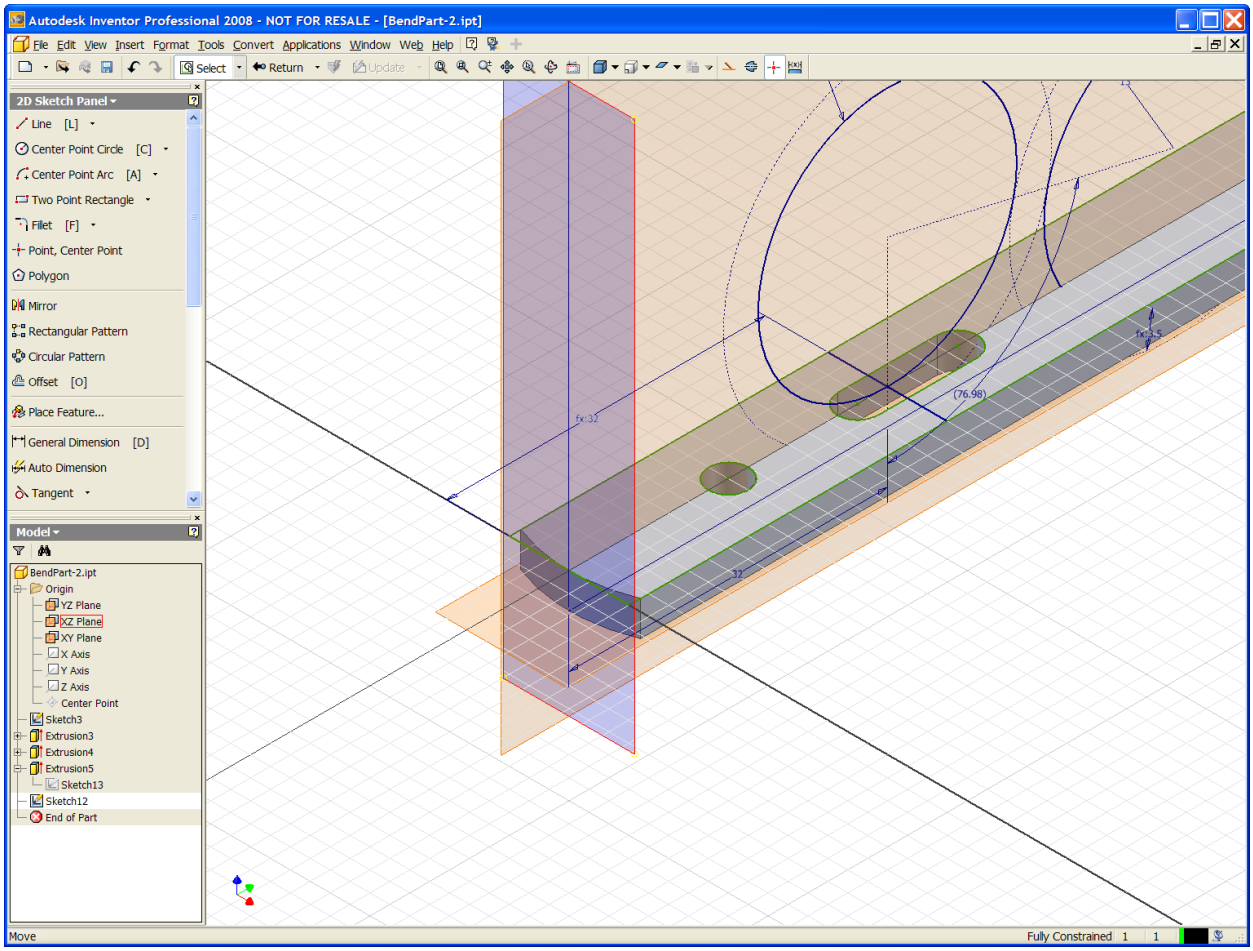
i. Location: 32 mm from the end

ii. Bend Angle: (76.98): Added the reference geometry /dimension to the layout.



b. The sketch: Sketch the bend line

i. Location dimension (fx:32): selected the 32mm in the layout sketch

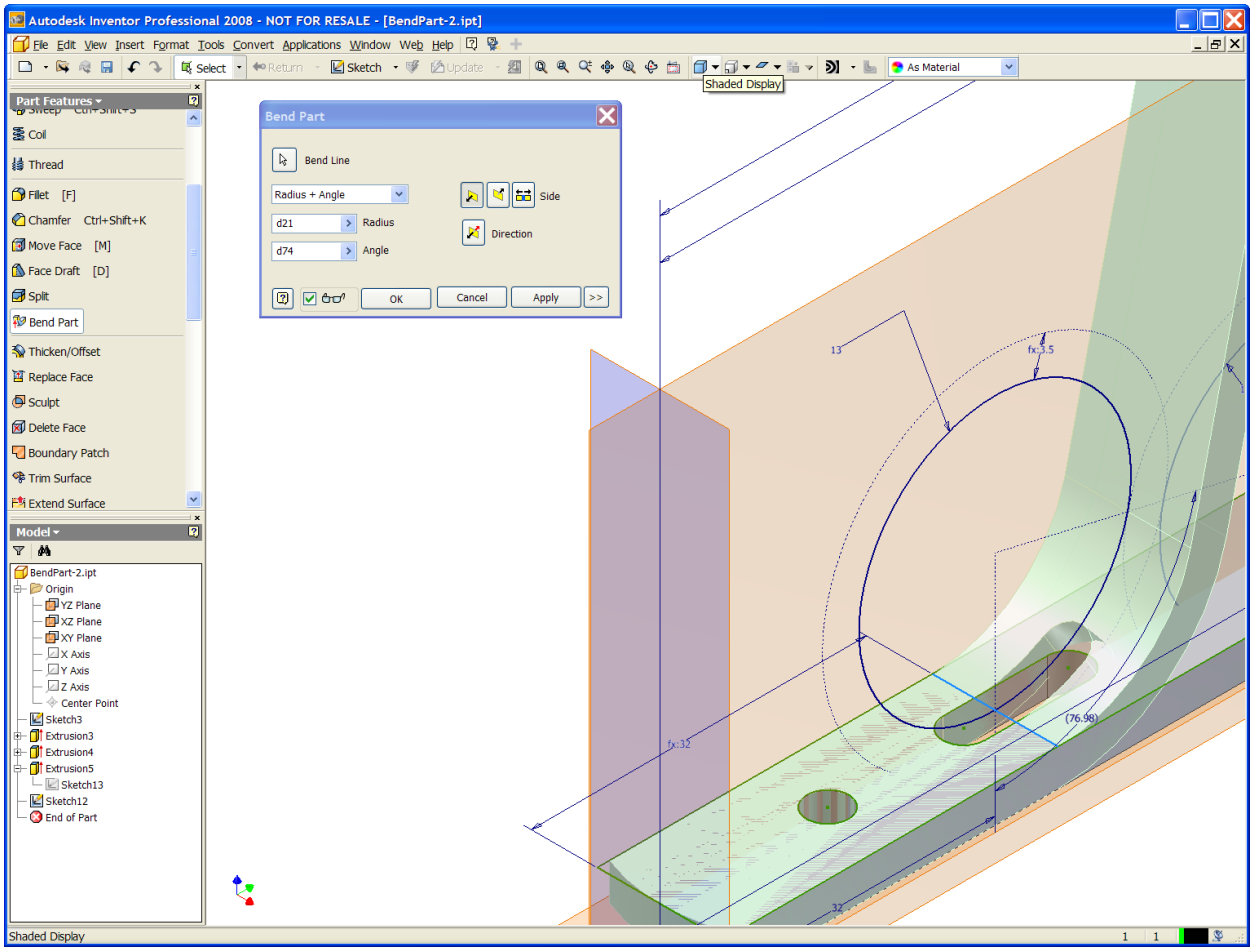


c. The feature

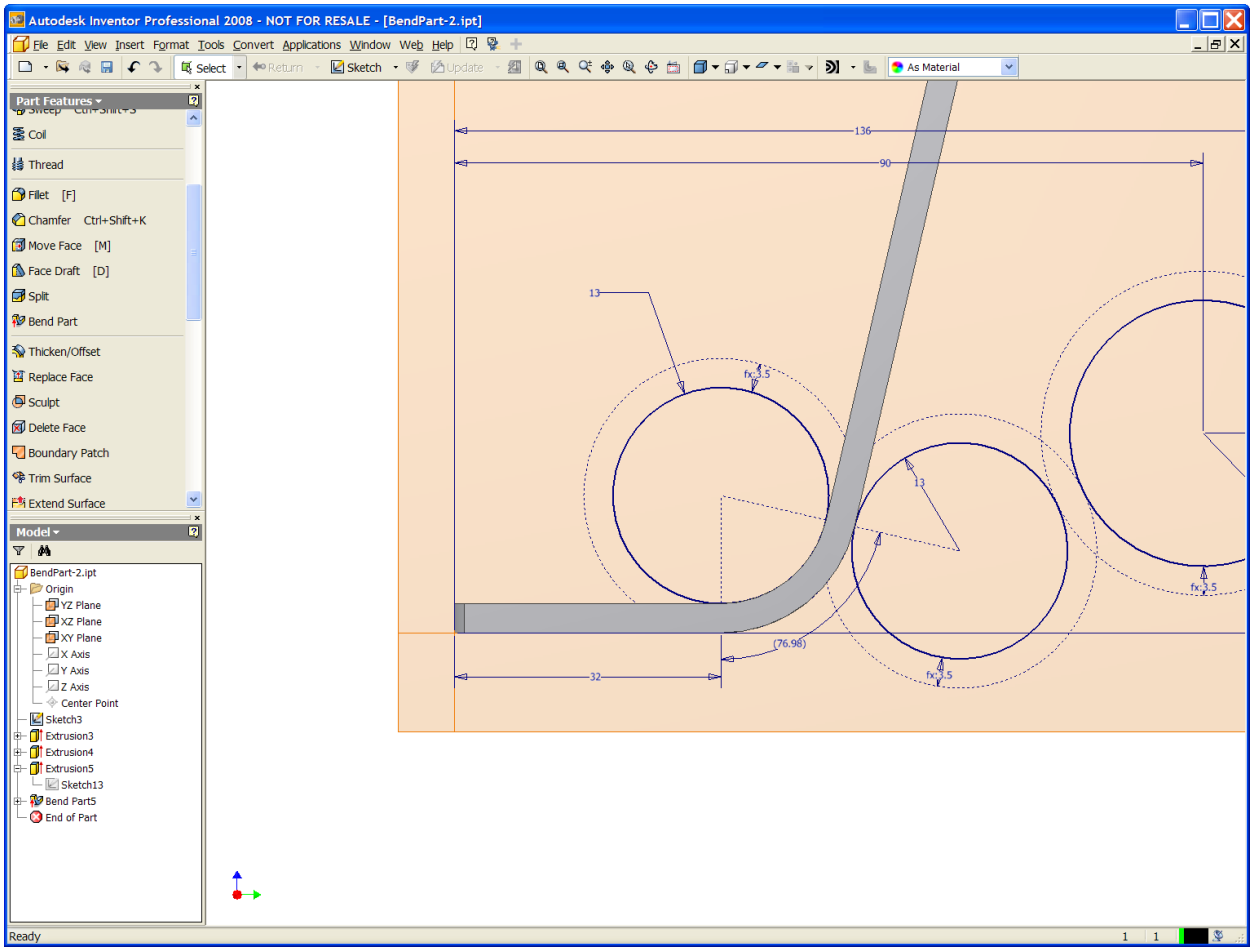
i. Bend Line: Just sketched

ii. Radius: select dimension (13mm) in layout

iii. Angle: select dimension (76.98) in layout

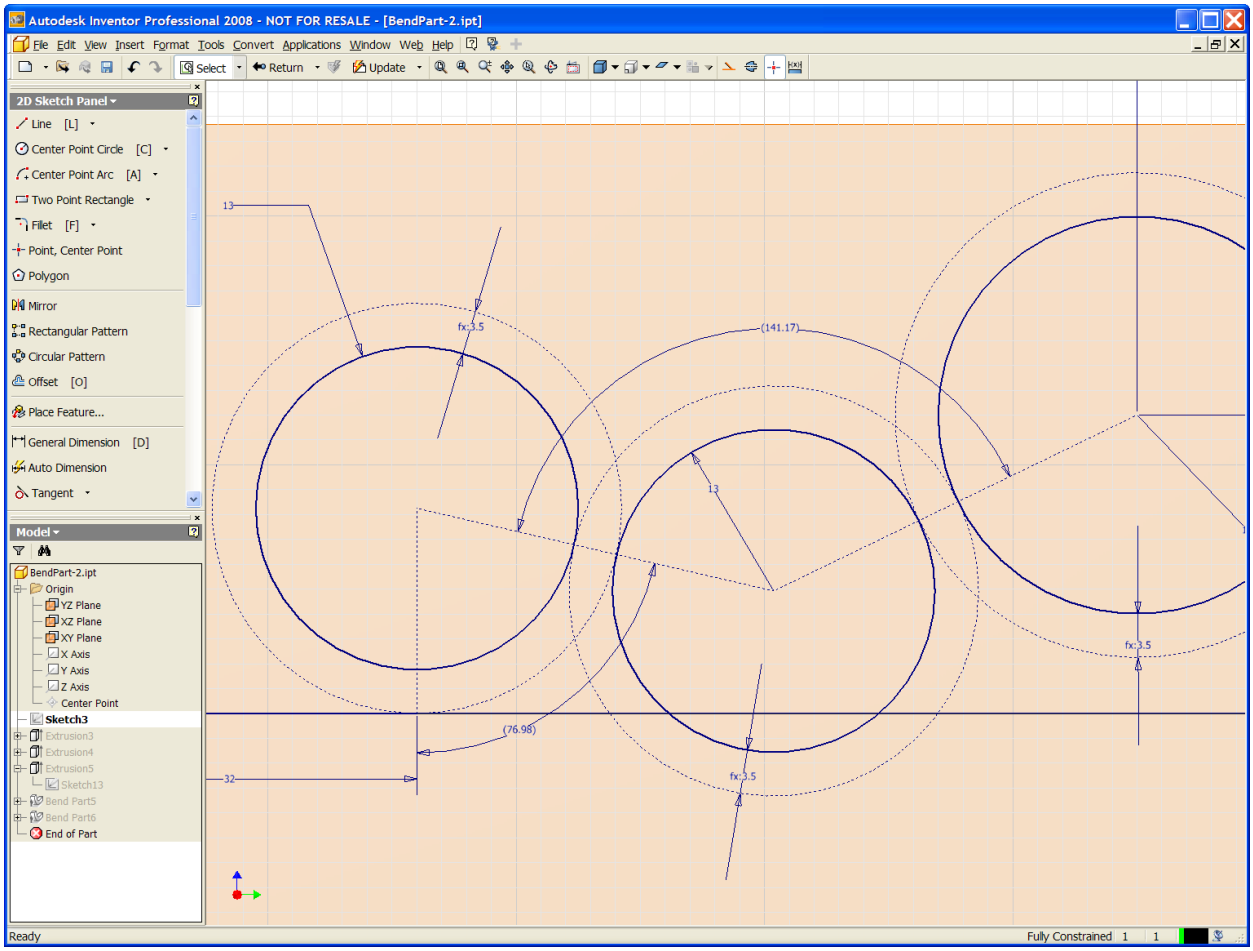


d. Verify the feature



6) Next Feature: Second Bend

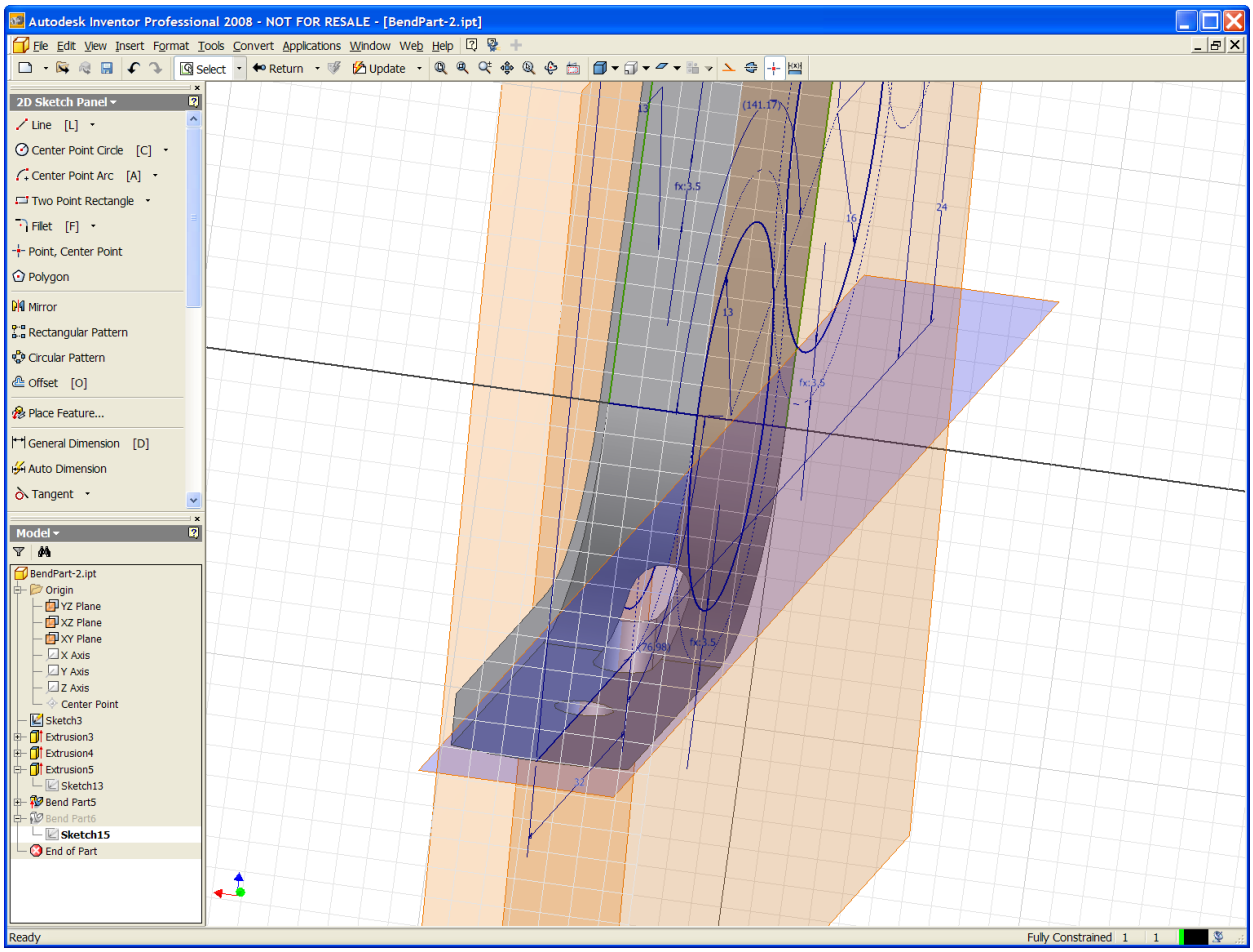
a. Edit Layout / Determine the bend angle = 141.7



b. Sketch the bend line.

i. Select the face of the part that is attached to the bend radius(13)

ii. Use project geometry, select the end of the previous bend

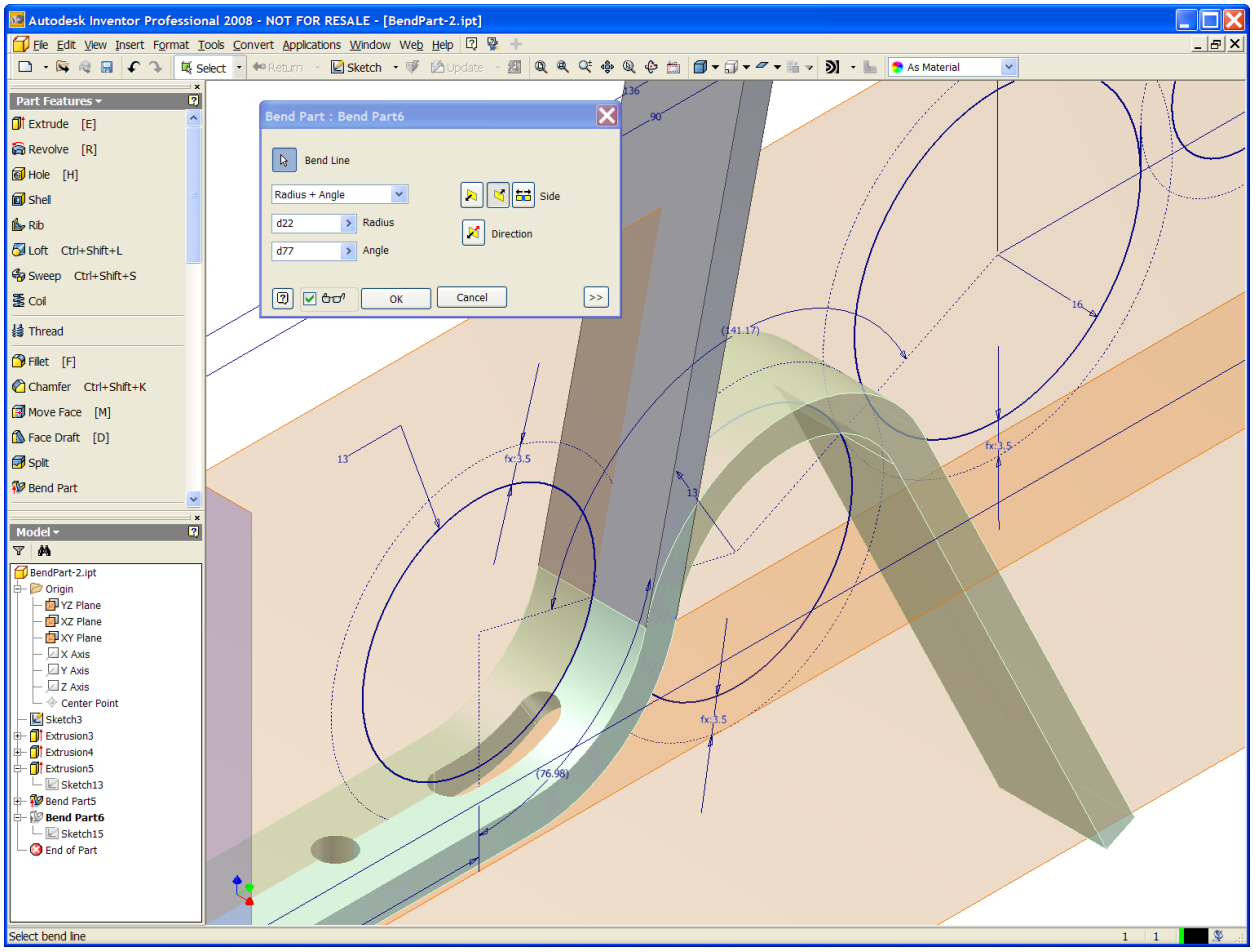


c. The feature

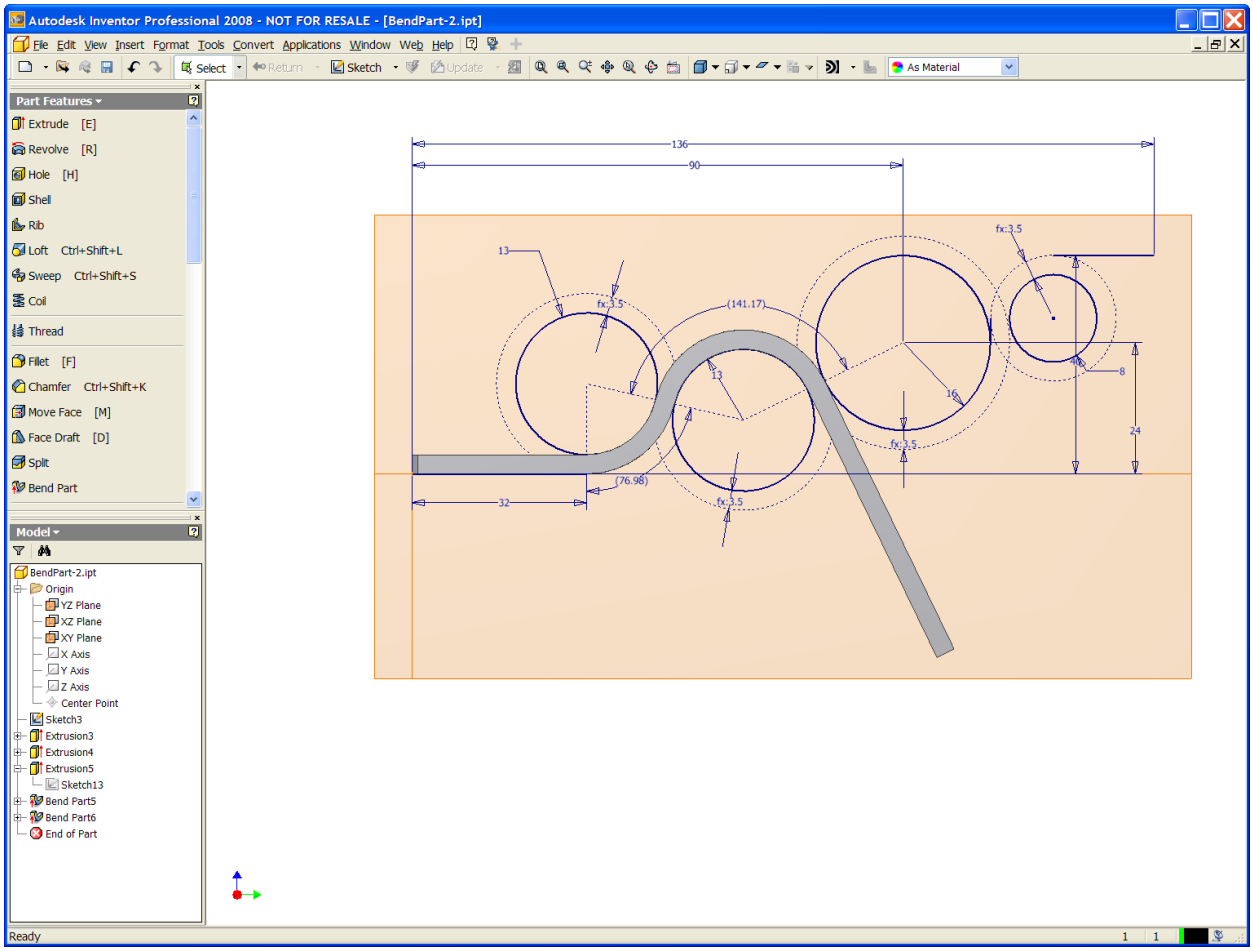
i. Bend Line: Just sketched

ii. Radius: select the dimension (13) in the layout

iii. Angle: select the dimension (141.7) in the layout

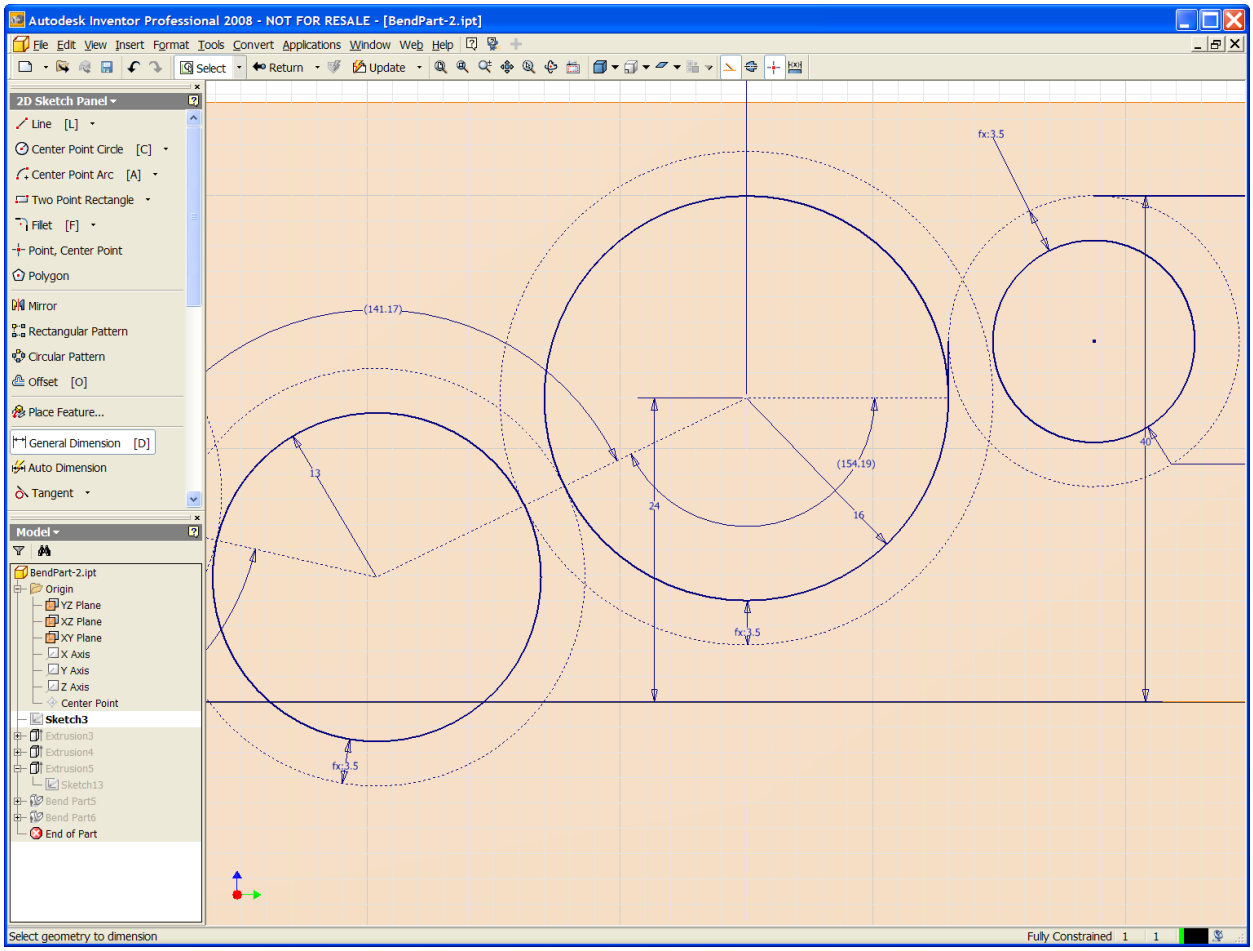


d. Verify the feature / Compare it to the layout.



7) Next Feature

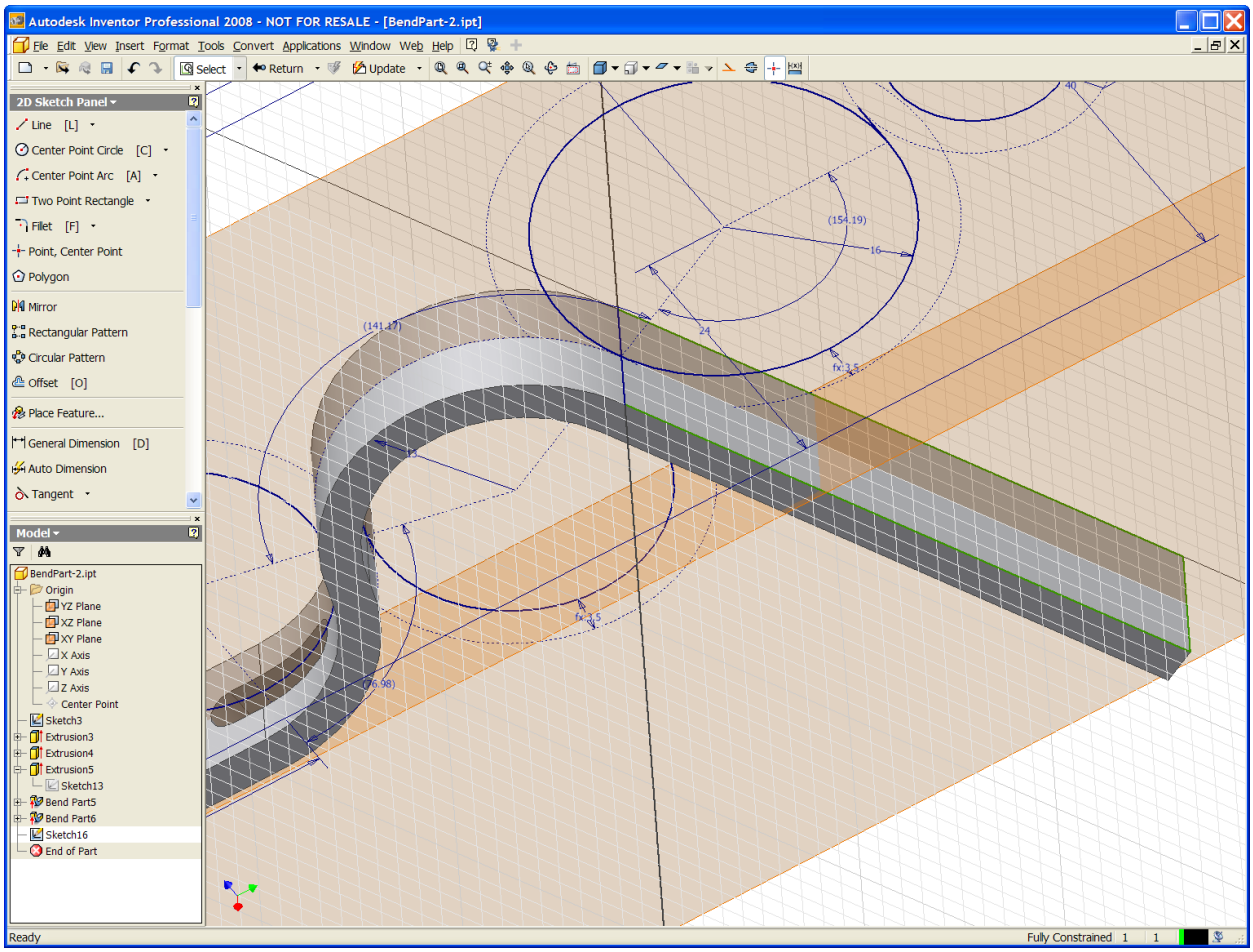
a. Edit the layout / determine the required bend angle (154.19



b. Sketch the bend line

i. Select the face that is attached to the bend radius(16)

ii. Sketch the line, align to the start of the flat face

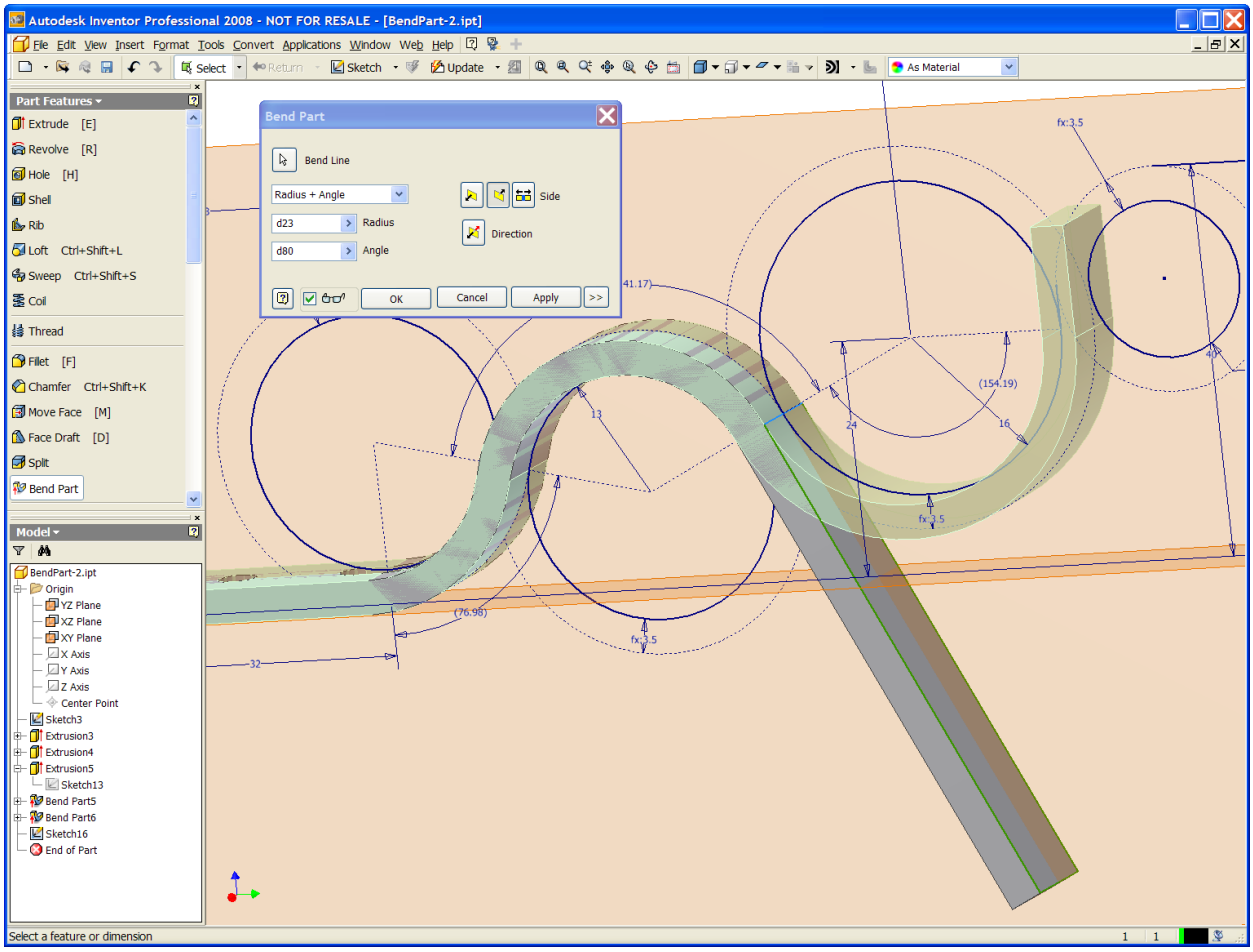


c. The feature

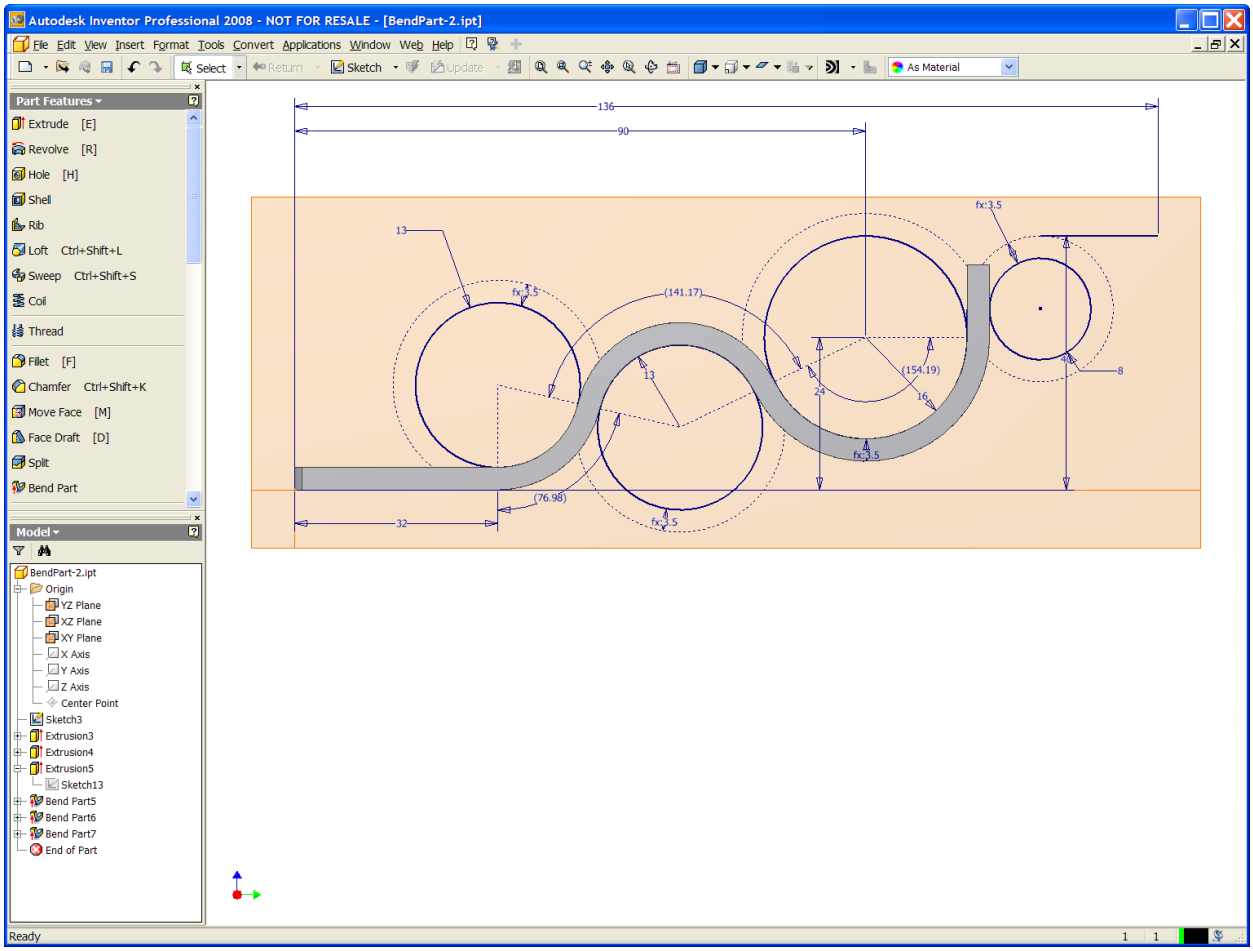
i. Bend line: From previous sketch

ii. Radius: Select dimension (16) in layout sketch

iii. Angle: Select dimension (154.19) in layout sketch



d. Verify the feature



8) Next feature

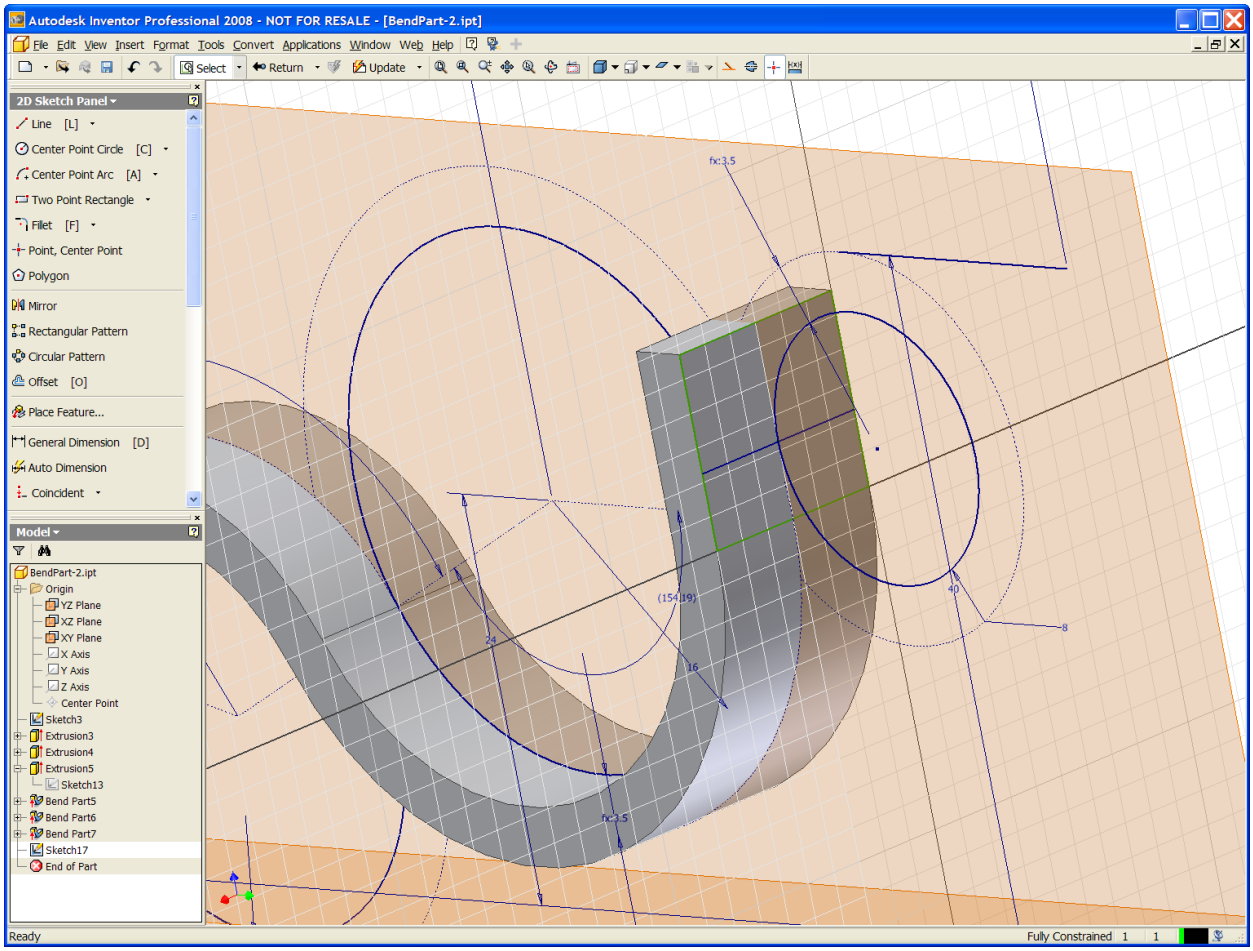
a. Edit the layout: Not required all dimensions are known. For the sake of consistency, it would have been a good idea to add a couple of dimensions used below. Then the part would have been entirely dependent on the layout.

b. Sketch the bend line

i. Sketch Plane: attached to the bend radius(8)

ii. Project Geometry: select the center point of the 8mm bend radius

iii. Sketch a line. Constrain it to the projected point.

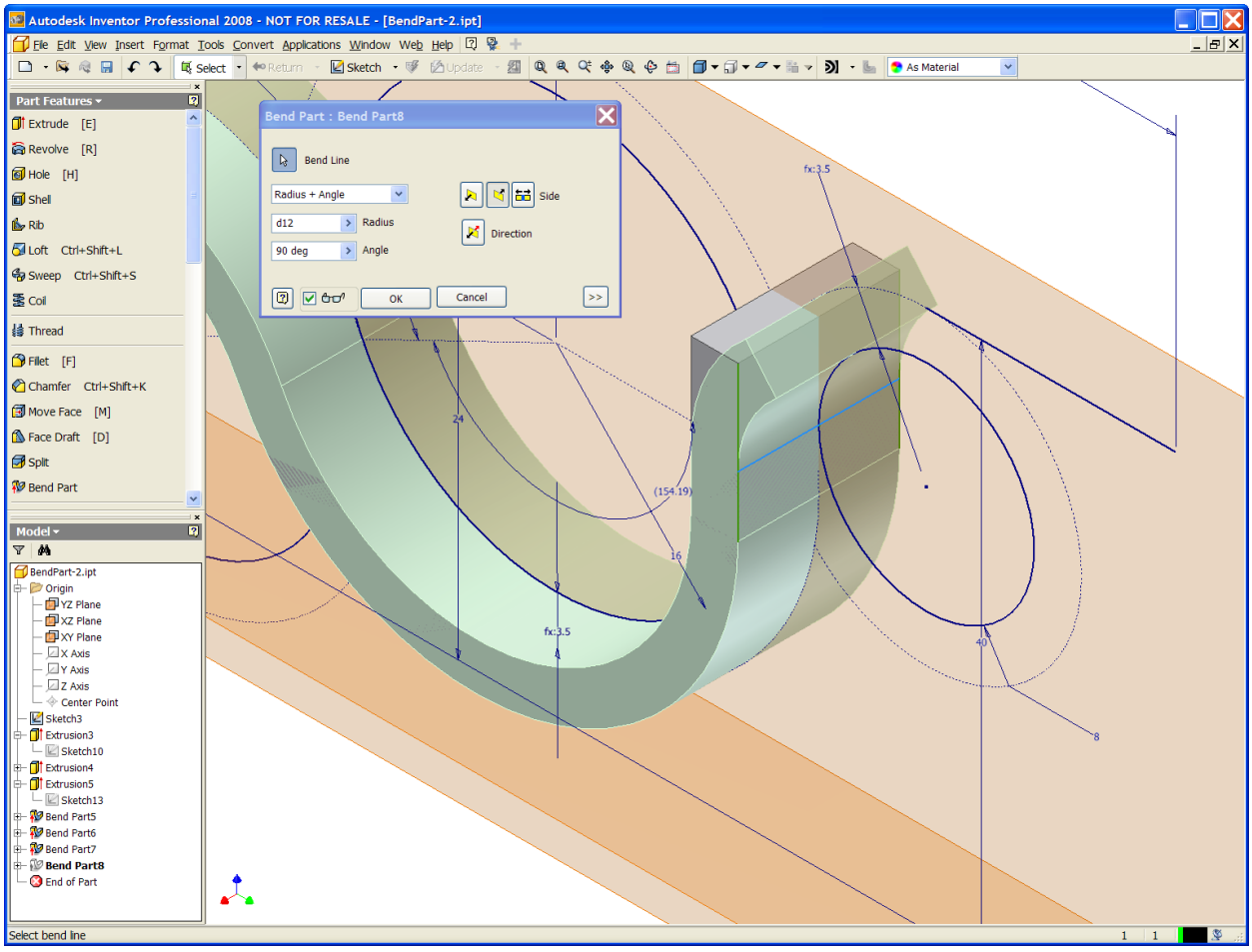


c. The feature

i. Bend Line: Previous Sketch

ii. Radius: Select the radius(8) in the Layout sketch

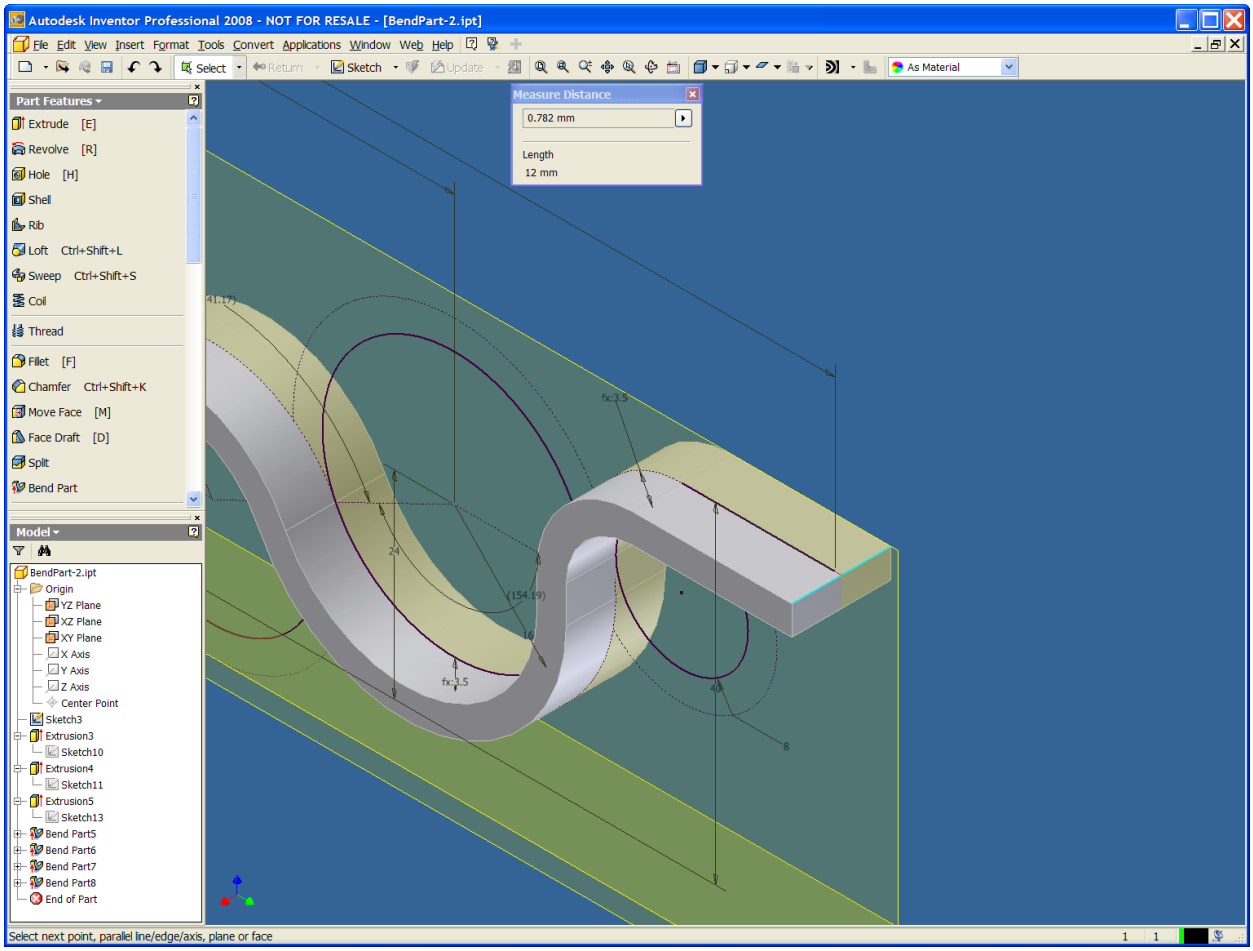
iii. Bend Angle: Entered the value: 90. (Probably should add/ use a 90 degree dimension to the layout, see previous note



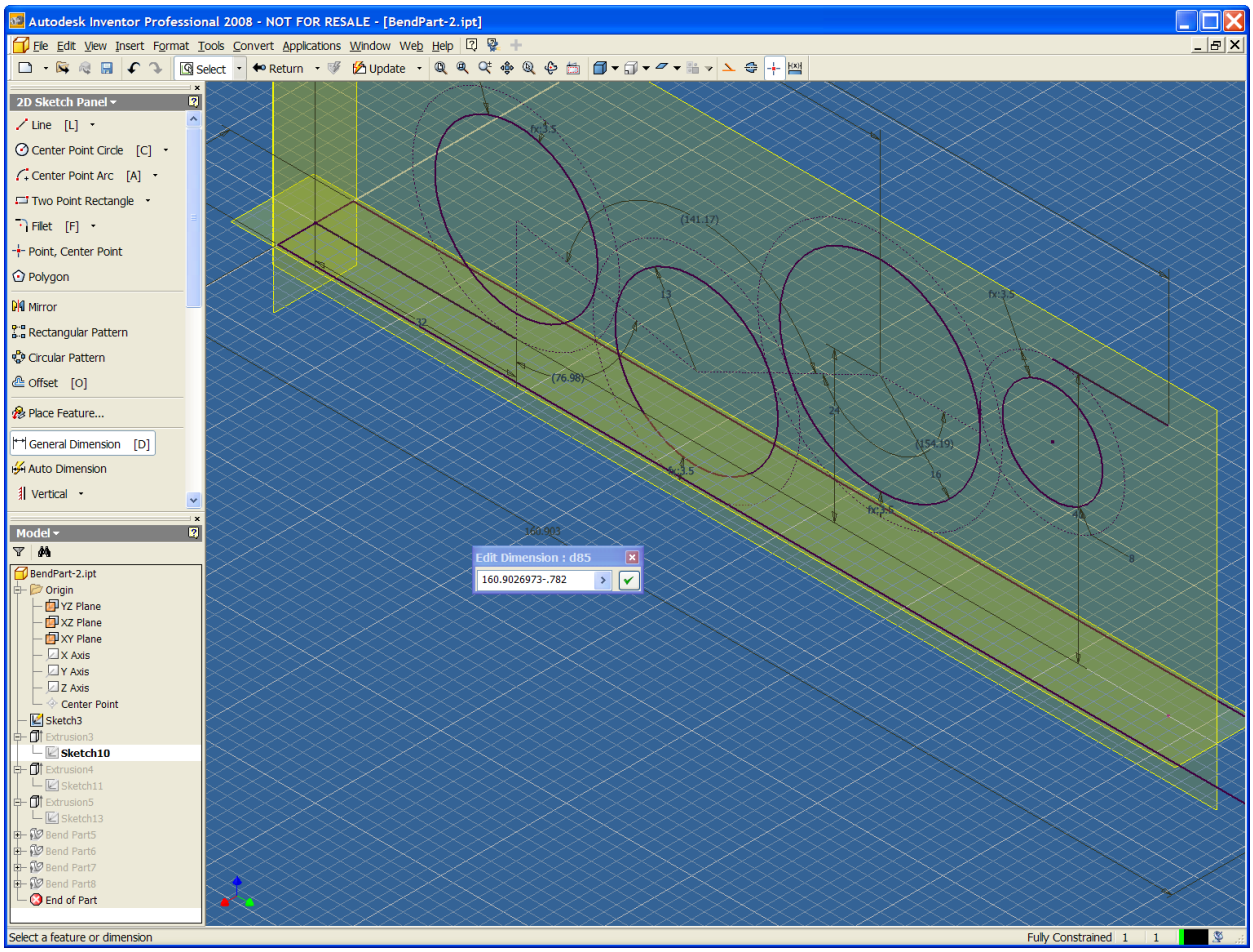
d. Verify: Note the length of the spring isn't sufficient.

i. Go back and lengthen the base extrusion, leave it underdefined in the base feature sketch. The final distance will be determined shortly.

ii. Use the measure distance tool to determine the distance from the end of the layout sketch to the end of part model. In this case= .782



iii. Modify the length of the base feature: Add the value determined in the previous sketch to the current length of the part. The base feature is now the required length.

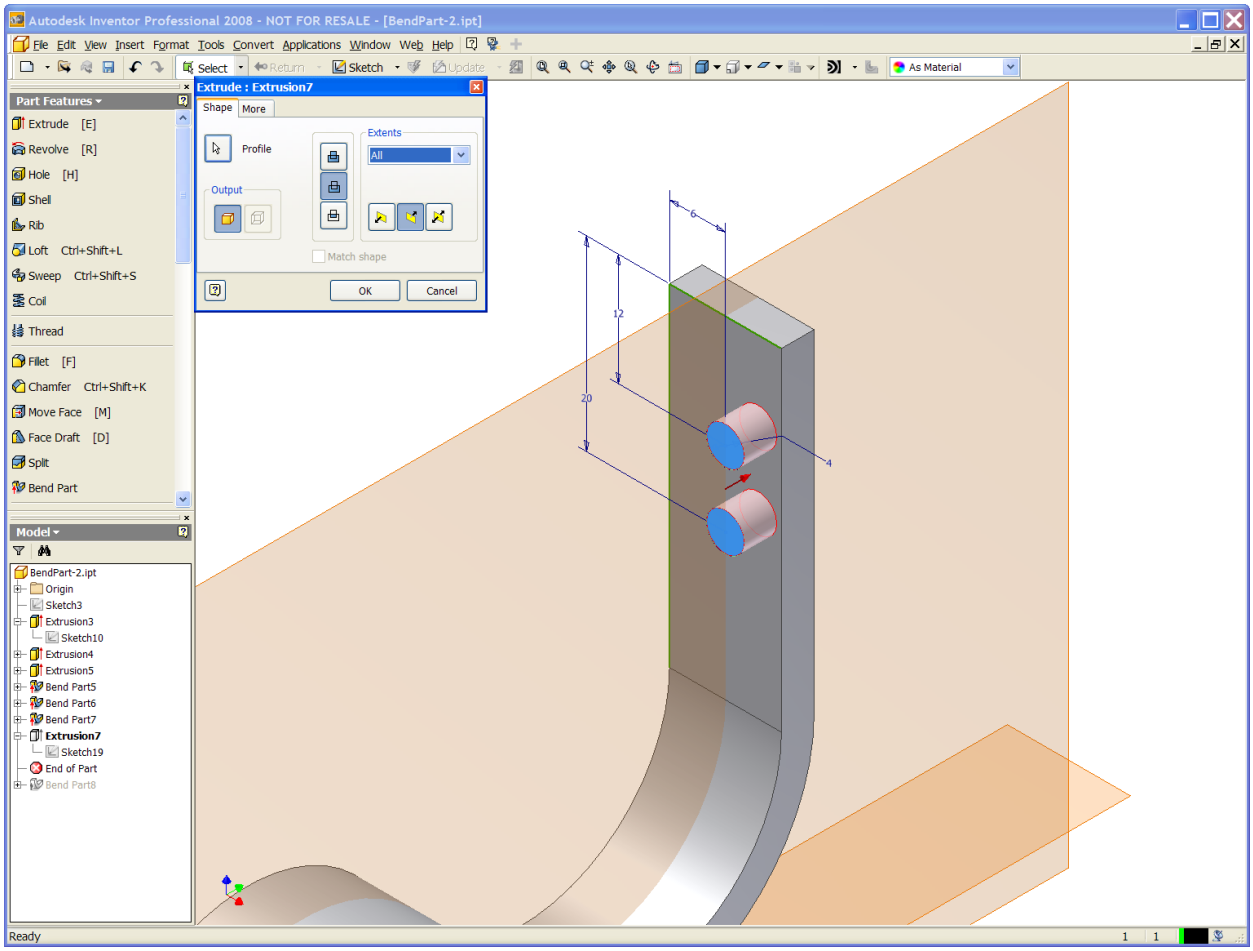


9) Next / Final Feature

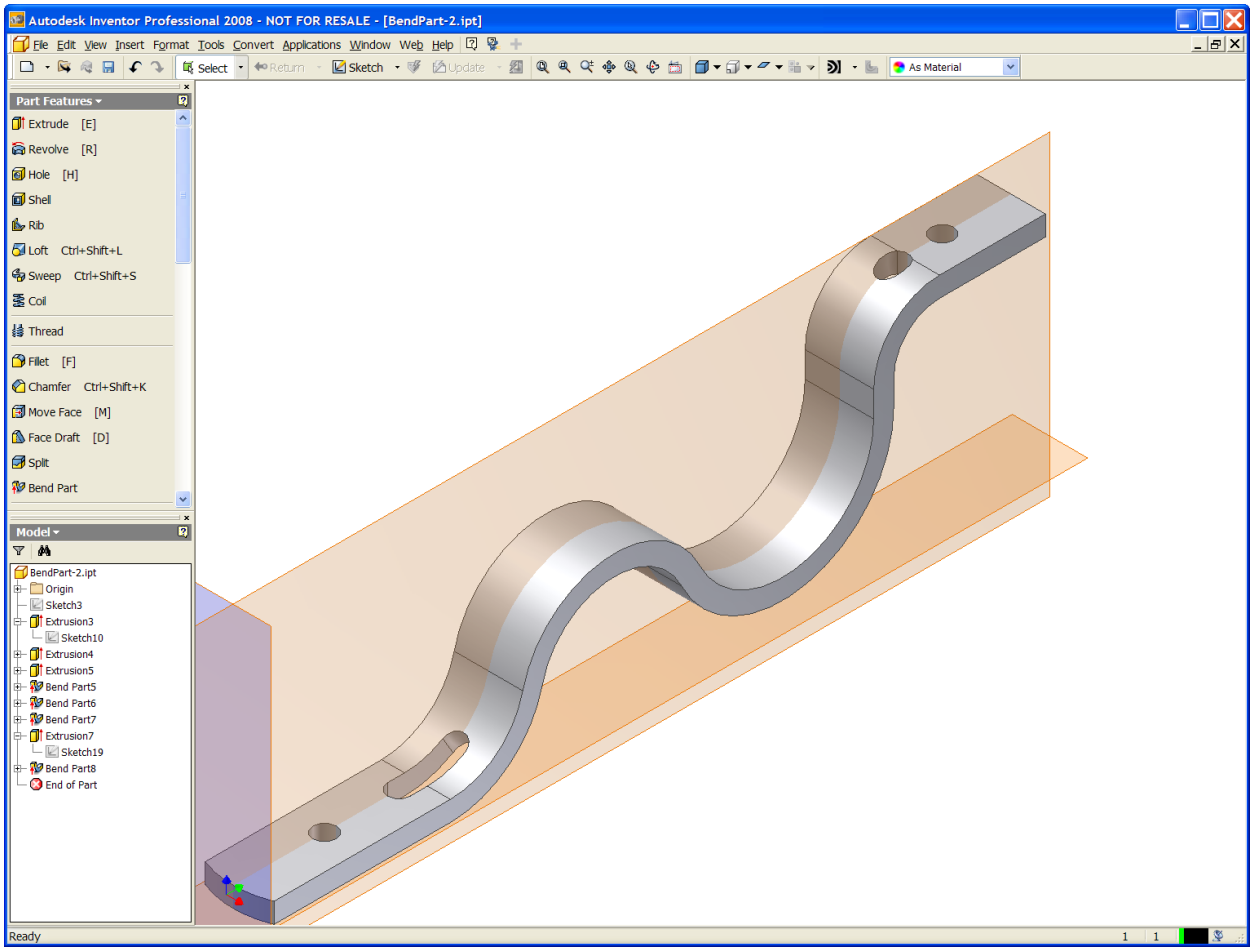
a. Move the End of Part back to just before the last bend.

b. Sketch / dimension the two holes

c. Extrude / Cut the holes



d. Verify: Move the End of Part back to the bottom of the list. This is the final / complete part.



Comments:

1. There is also the sheet-metal modeling option. Includes the ability to work in the flat pattern form. Seems like it would make sense to do this as a sheet metal part.
2. Inconvenient to have to figure out the length of a part.
    a. Could be done as an adaptive part in an assembly
    b. “Easier” to accomplish as sheet-metal
3. Otherwise, this seems to be another tool / feature to be aware of and know how to use.

(An idea: Work in an assembly. Part one contains the layout sketch. Part Two is the spring. Make the spring adaptive.