Department of Mechanical Engg.
INDIAN INSTITUTE OF TECHNOLOGY, DELHI

MEL110 GRAPHIC SCIENCE

Notes on

Autodesk Inventor Professional Software

Contents

1. What are the basic features of an Autodesk Inventor Professional Software?

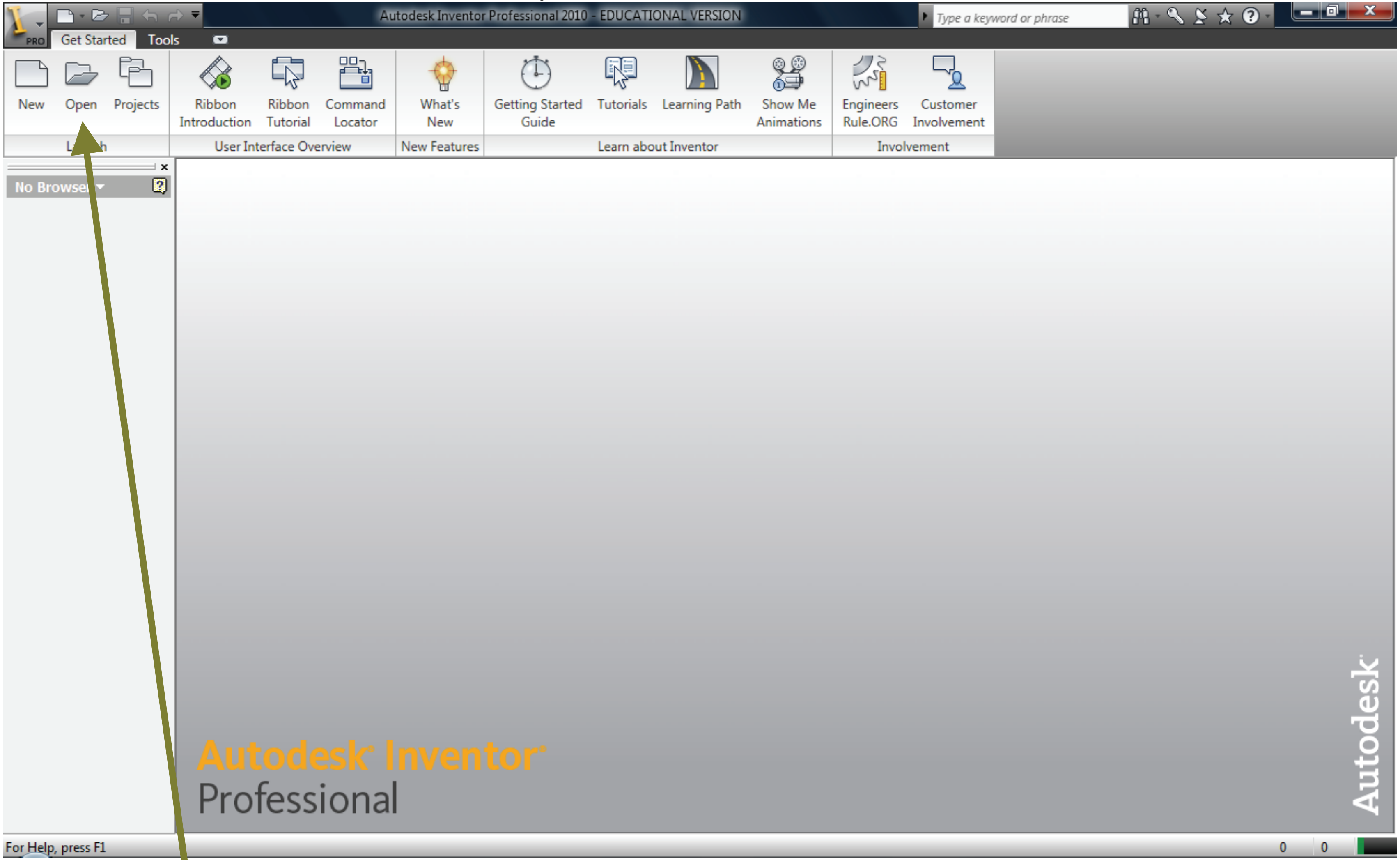
- It is a 3D mechanical engineering, design, visualization, and simulation software
- Autodesk Inventor is a parametric and feature-based solid modeling tool. It allows you to convert the basic 2D sketch into a solid model using very simple modeling options.
- It creates digital prototyping as opposed to physical prototyping (*which is costly and time consuming*) by integrating 2D AutoCAD drawings and 3D data into a single digital model
- It can quickly and easily create stunning renderings, animations, and presentations that improve communication
- It can easily generate and share production-ready drawings for manufacturing teams
- The automatic updating feature allows easy changes in models
- It has a simulation environment that allows motion simulation, static and modal finite element analysis (FEA) of parts, assemblies, and load-bearing frames

2. How is it useful in MEL110 course?

- It helps in understanding the underlying concepts of graphic science much faster and in a better manner by a beginner
- It helps in improving visualization of the solid model and understanding its drawing
- A solid model of the object is what is required to be created in the first step
- and rest of the process like creating a standard drawing in either first or third angle projection method with different views (like orthographic, isometric, auxiliary, sectional, perspective) is completely automated
- The process of adding dimensions (with all standard notations) to the drawing is also automated
- It gives flexibility in editing the solid model to visualize the effect of the changes

3. How to explore existing solid models?

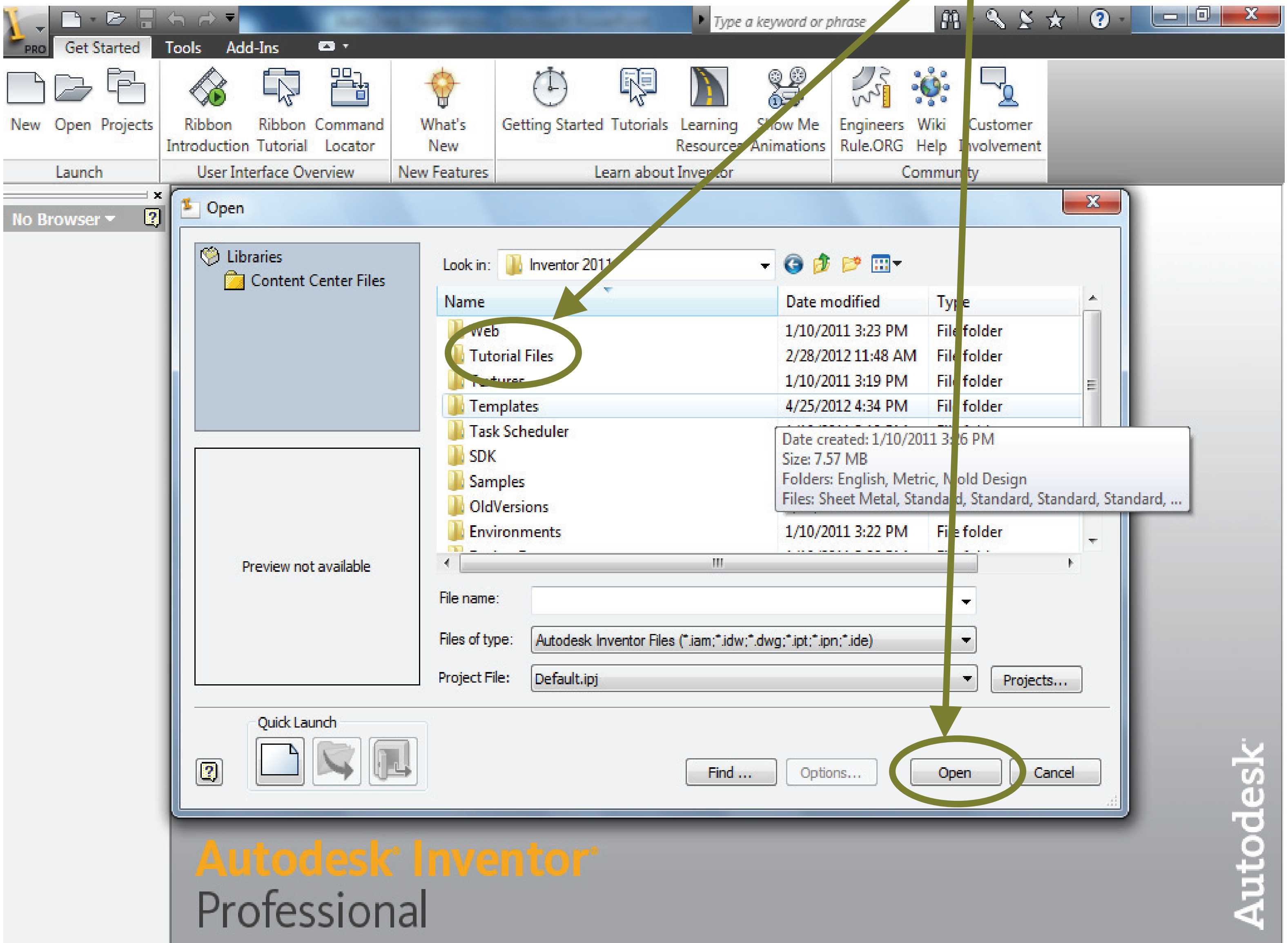
- Open Autodesk Inventor by double clicking its shortcut on desktop or by selecting it from program list
- The initial screen will be displayed, as shown below



- Use **"Open"** command to open an existing part file

3. How to explore existing solid models?

- Go to **Libraries>Documents>Autodesk>Inventor 2011>tutorial files>PivotBracket.ipt** (or any other part file) and click **open**



3. How to explore existing solid models?

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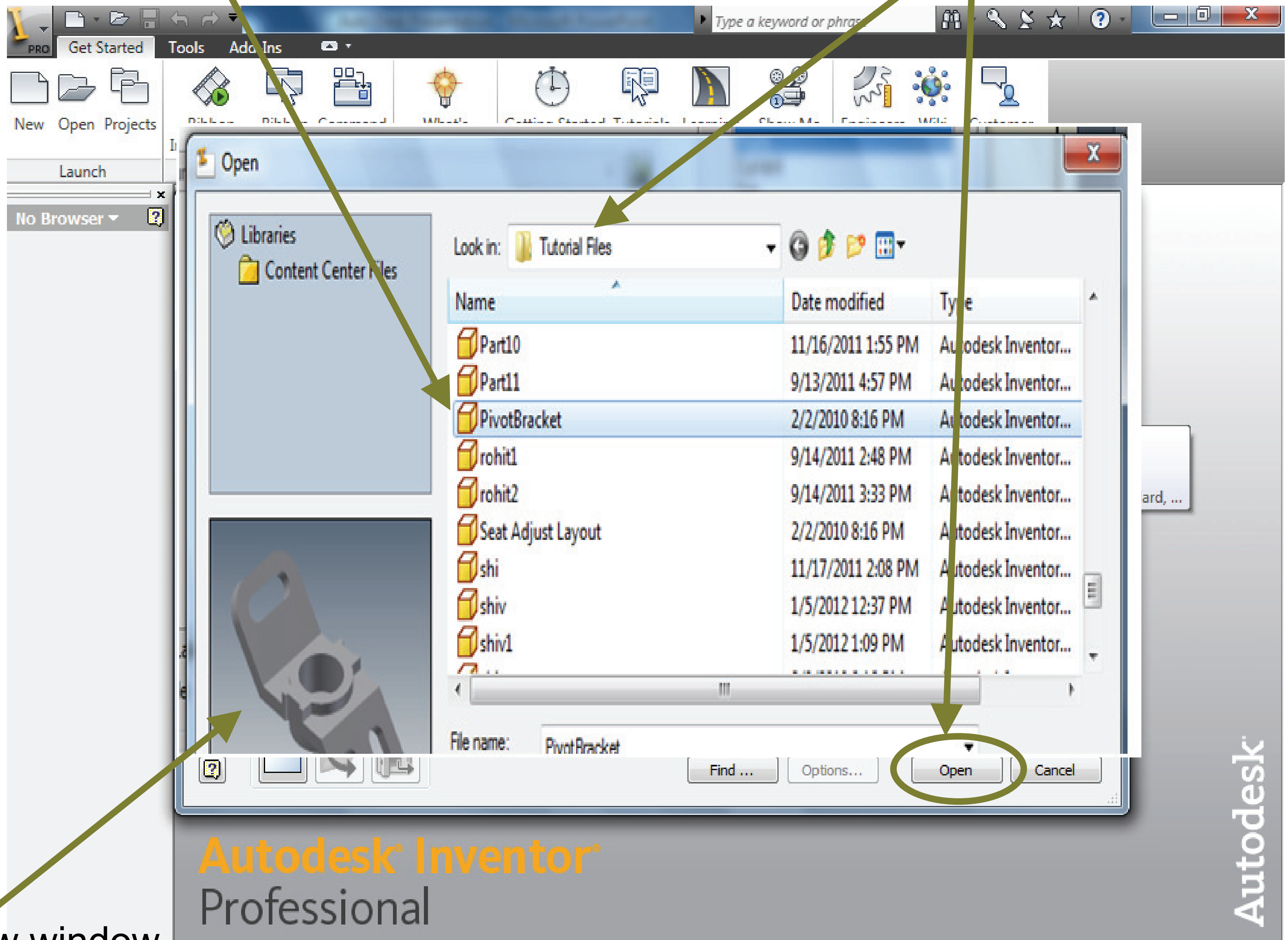
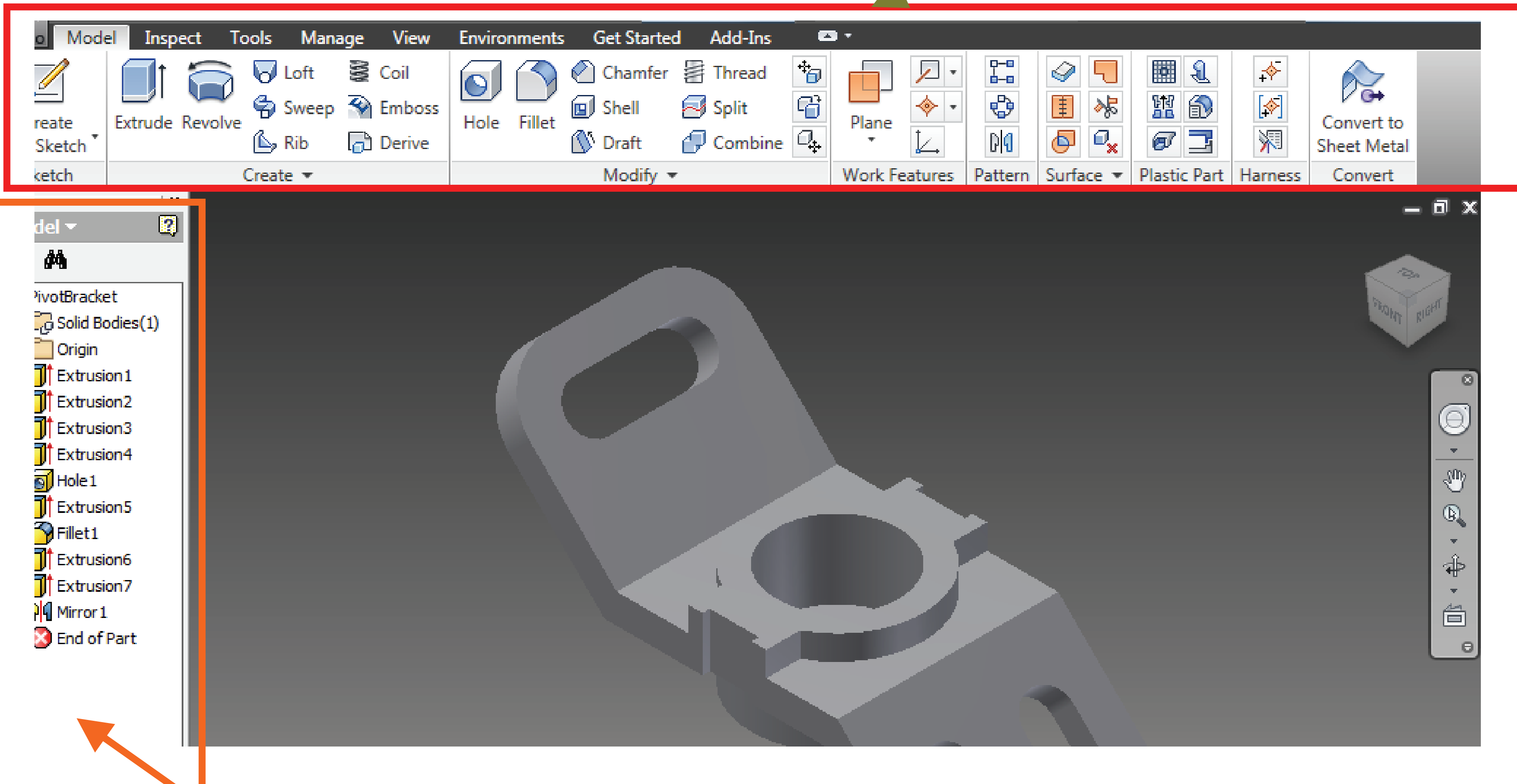


Preview window

3. How to explore existing solid models?

- This screen will appear

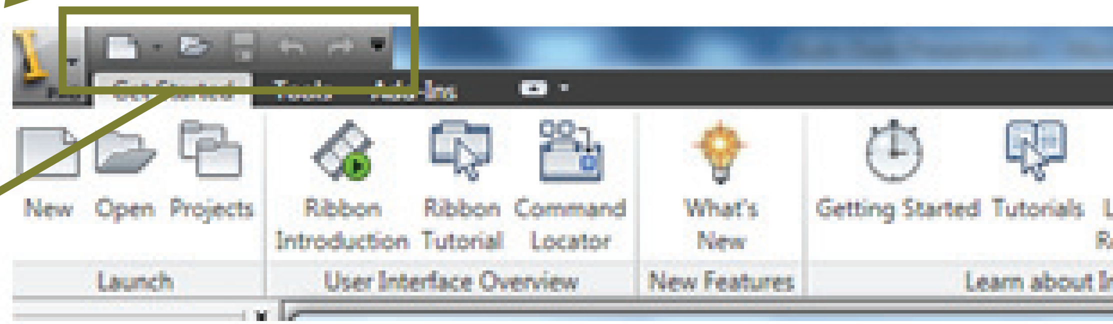
This is MODEL TAB. It provides all modeling tools that are used to convert sketch into feature

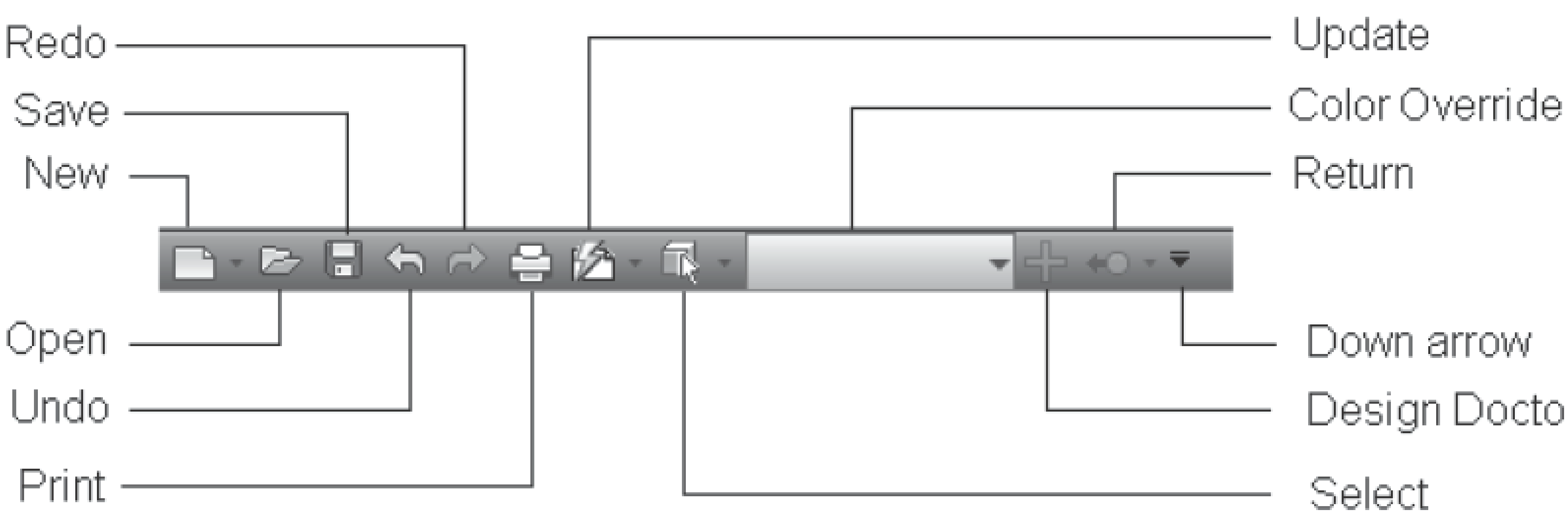


This is BROWSER WINDOW. It provides complete information about the model

3. How to explore existing solid models?

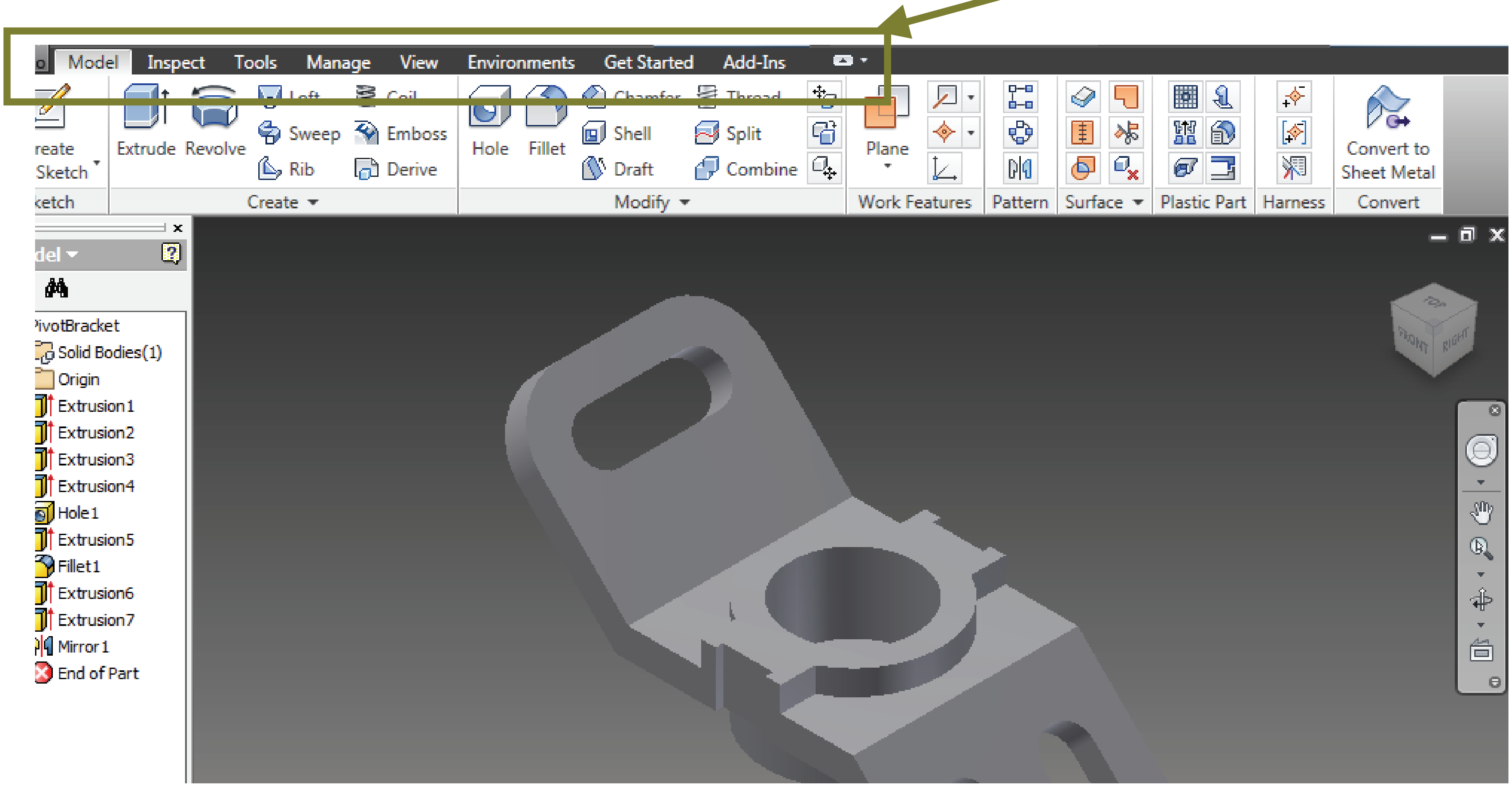
•The **Quick Access Bar** is placed here, for easy access and use of very frequently used commands





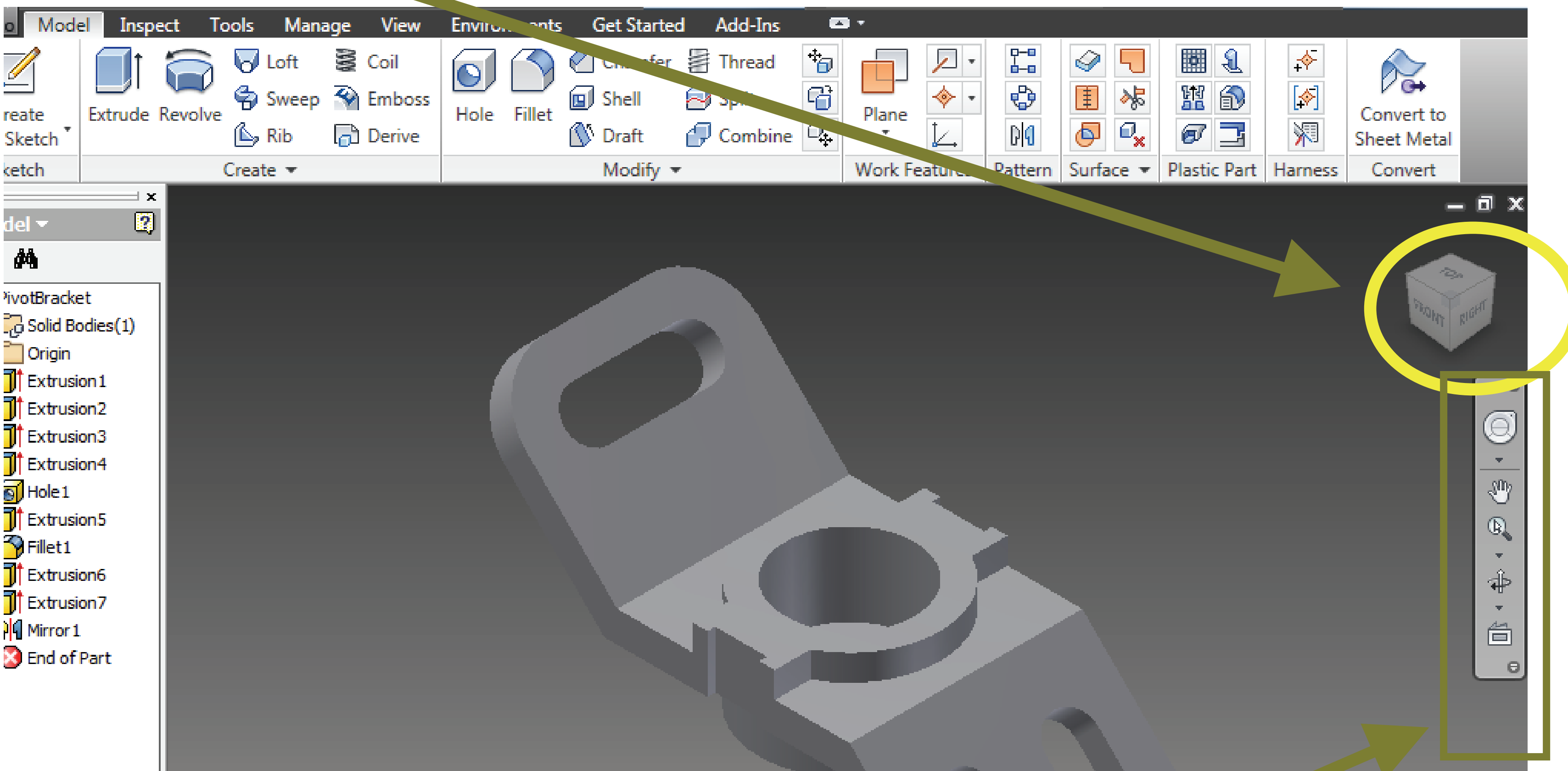
3. How to explore existing solid models?

•The commands are invoked from the tabs in the **Ribbon**. The **Ribbon** is a long bar below the **Quick Access Toolbar**.



3. How to explore existing solid models?

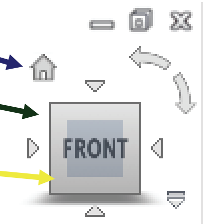
- **View cube**: It is displayed at the top right hand corner in the active area.



The tools on the **Navigation Bar** helps to control the view and orientation of components in the drawing window

3. How to explore existing solid models?

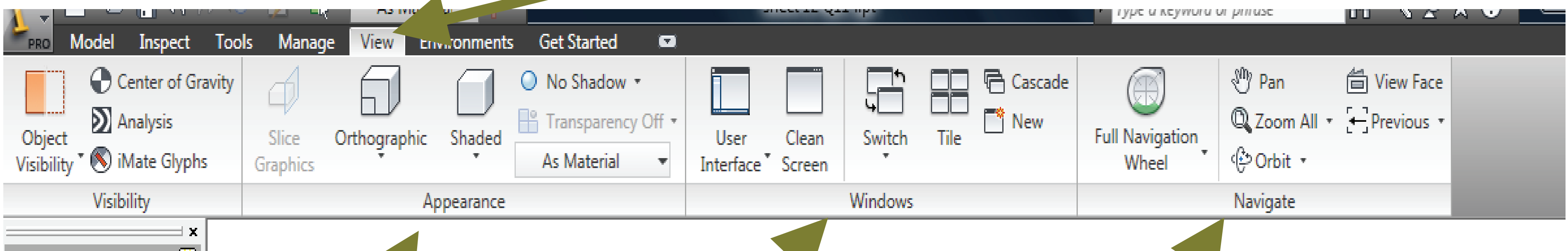
- **View Cube is** used to switch between standard and trimetric views of the model
- Click the '**Home button' t**o return to a user-defined base view
- Click the cube corners to snap the 3D model to trimetric views
- Click the faces to view orthographic views



- **Navigation Bar** is used to zoom, pan and rotate (orbit) the 3D model

3. How to explore existing solid models?

• the model can also be explored using the **VIEW TAB**



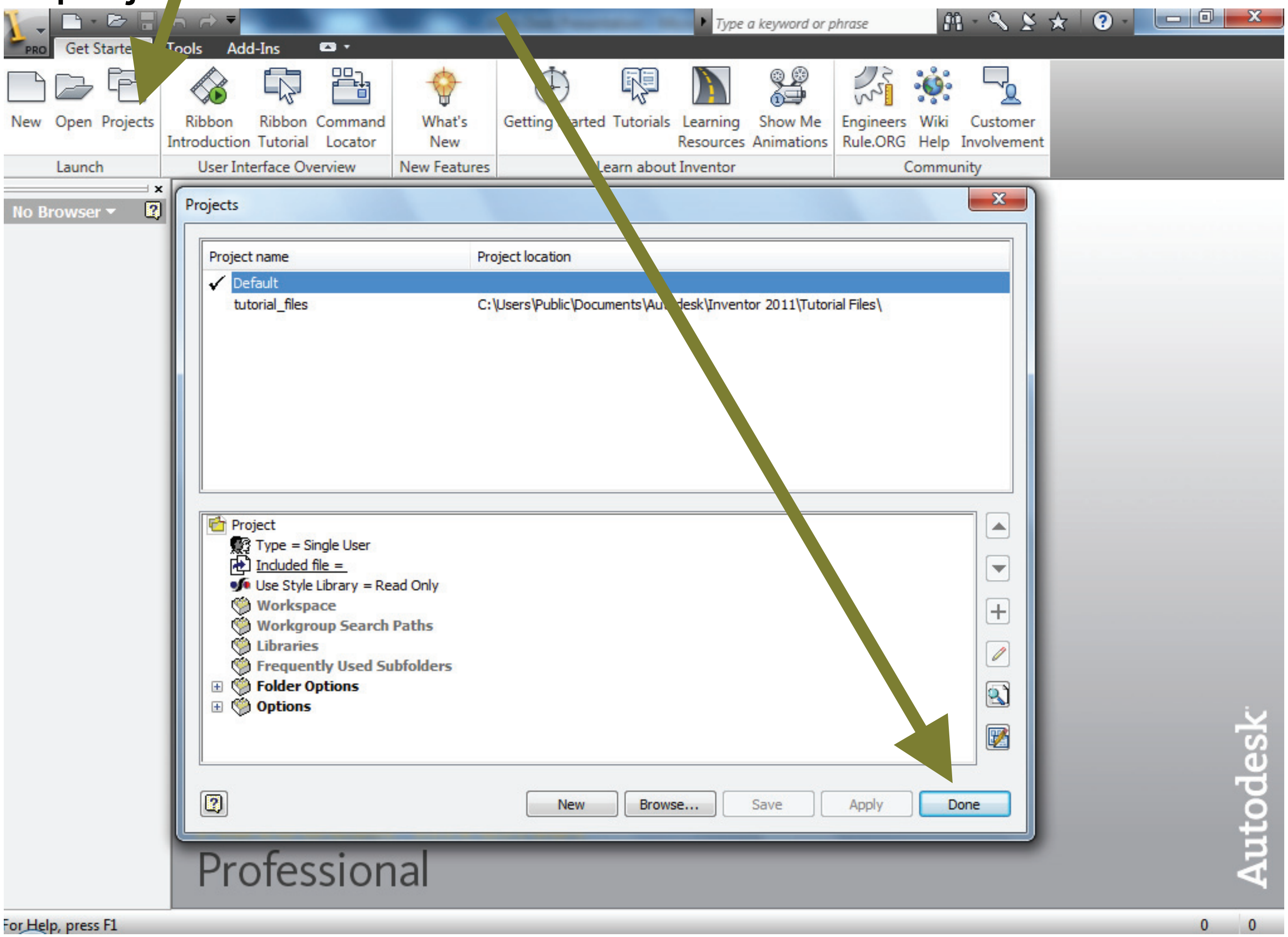
▪To zoom, pan and rotate (orbit) the 3D model

▪To change the windows

▪To change the appearance of the solid model

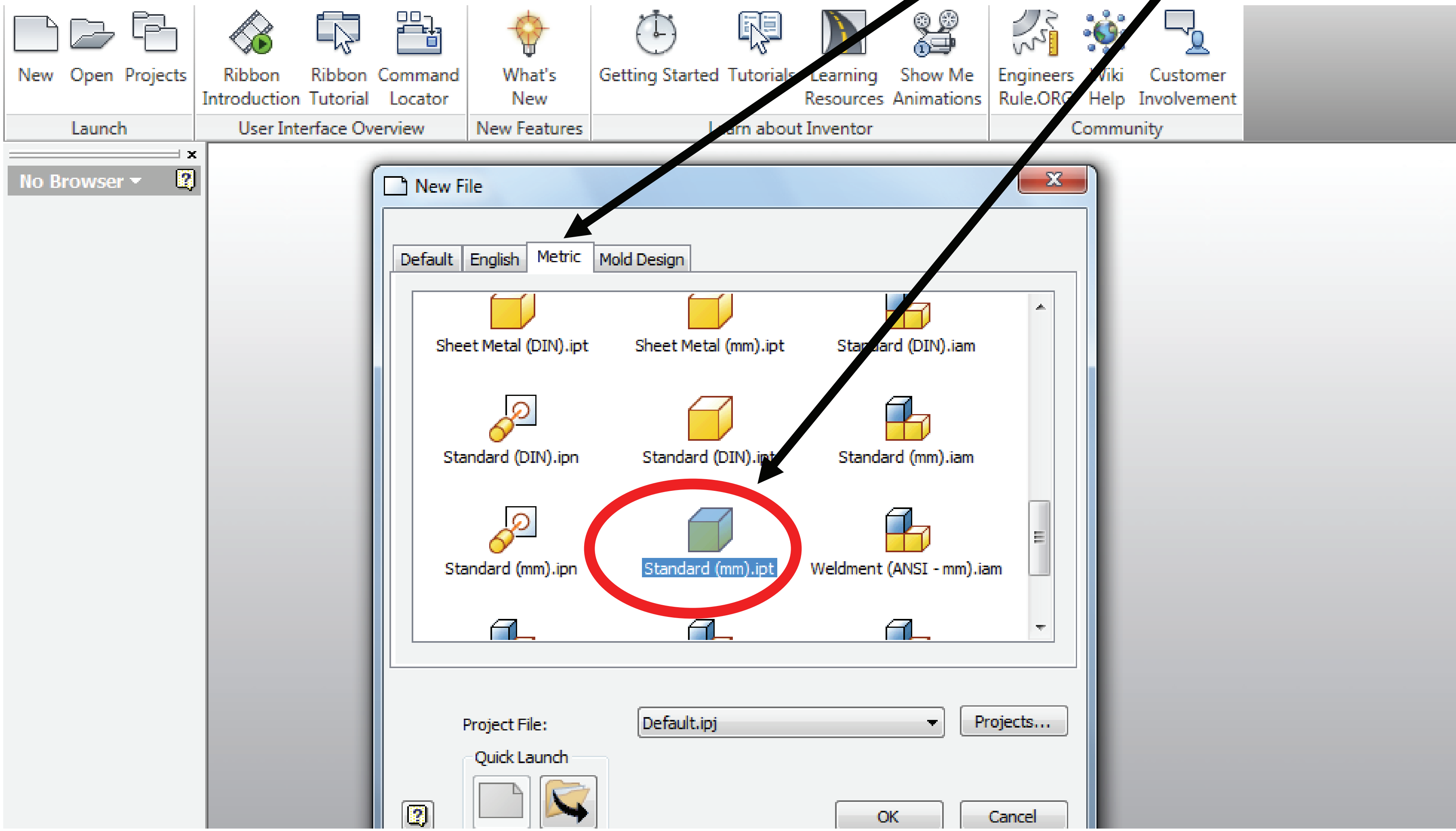
4. How to create solid models-I?

- First create a new sketch and then use modeling operations to create solid model
- Click the **‘projects’** command located in the launch panel and select the **‘default’** project and click **‘done’**



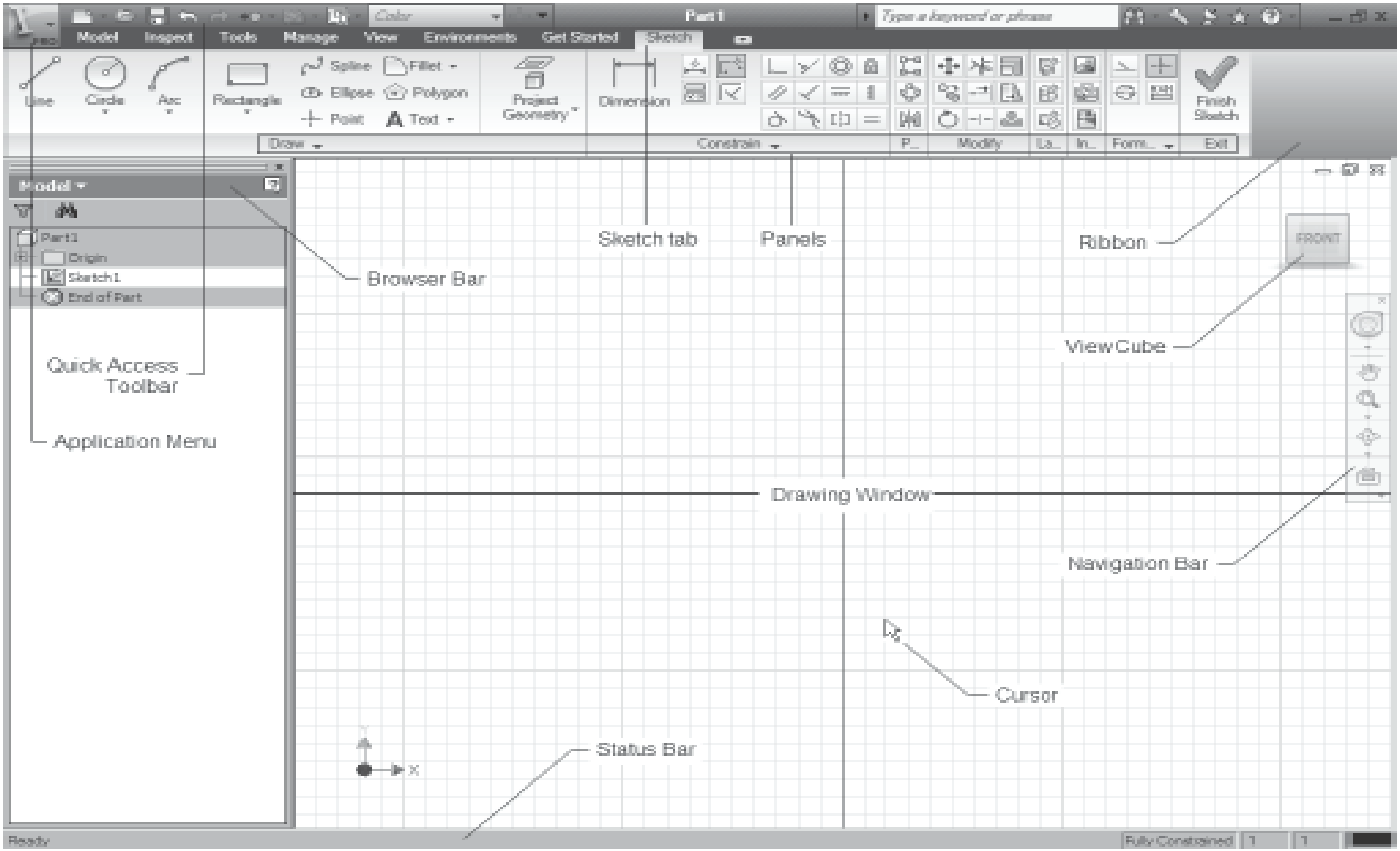
4. How to create solid models-I?

Create a new part file: In the ‘**Get Started**’ tab, go to **new** > **Metric** > **standard (mm).ipt** and click **OK**.



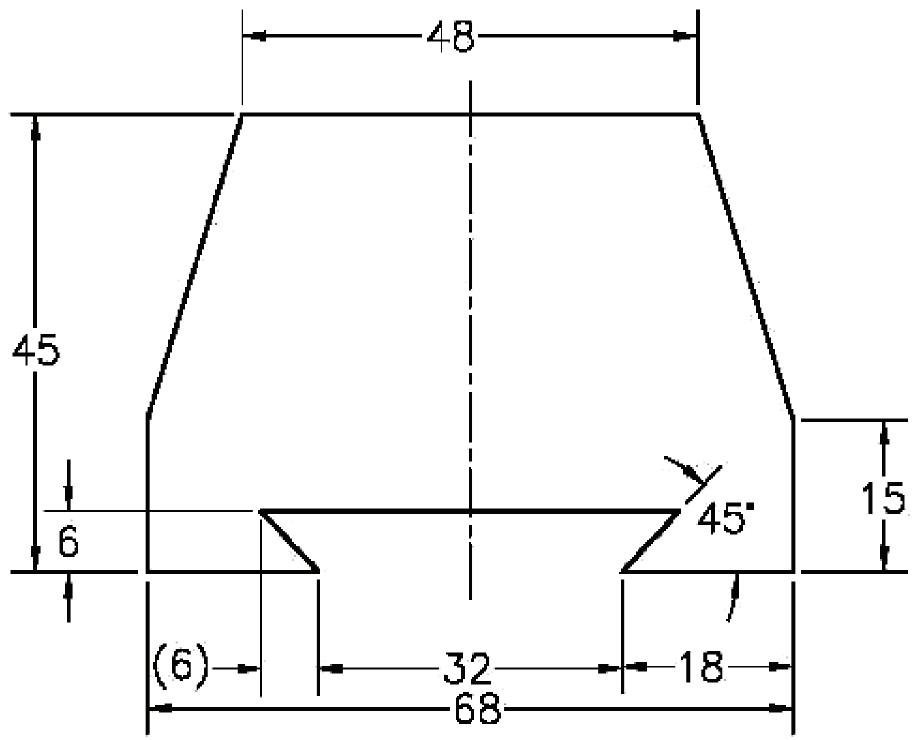
4. How to create solid models-I?

This will open the sketch mode with a default file name **'Part1'** (you can later save this part with a different name. It will have an extension .ipt)



4. How to create solid models-I?

- If the sketch mode is not activated, then select sketch 1 from the browser window which is located on the left side below the main tab.
- Select a working plane (XY, XZ or YZ plane) under **'origins'** tab from the browser window to create a 2D sketch or use the default working plane.
- Use the 2D **'sketch'** commands to create the following sketch using lines.



4. How to create solid models-I?

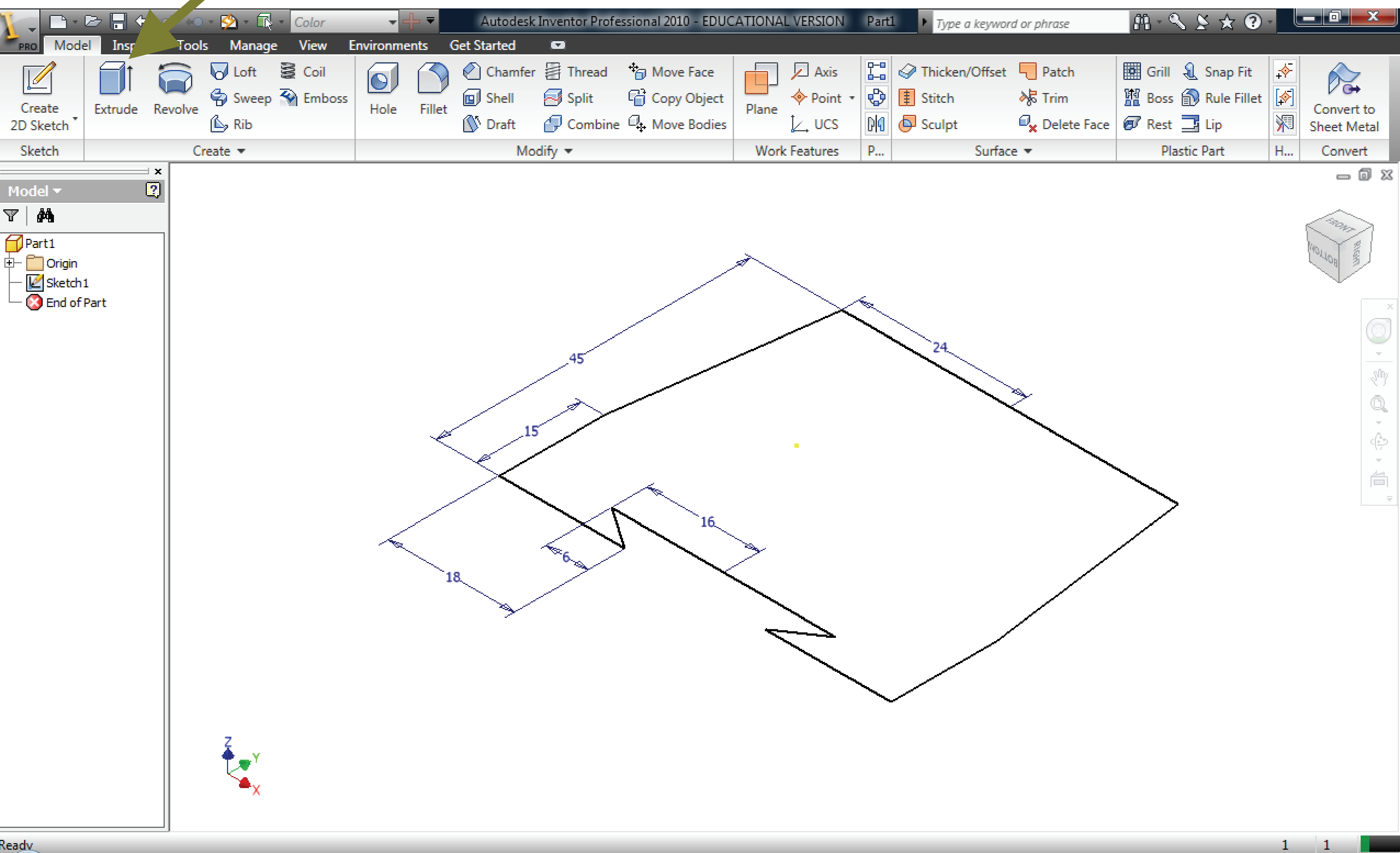
- Press ***'esc'*** button on the keyboard to exit out of any command.
- Now Dimension the sketch using '**dimension'** command located in the '**constrain'** command.

- Select the '**Finish sketch**' command to exit the sketcher after completing the sketch and '**close**' the active file.

A new window will appear with Model tab activated

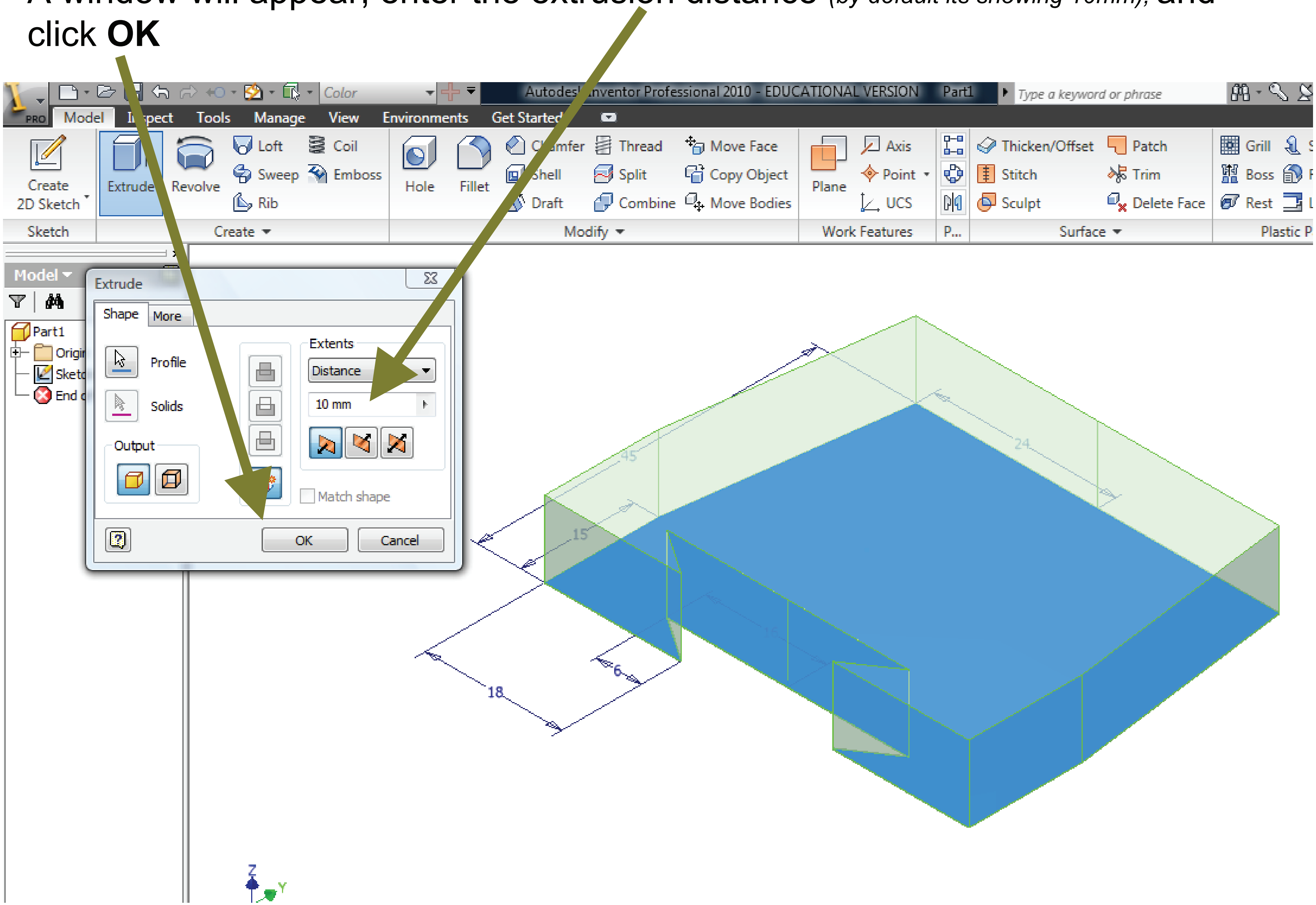
4. How to create solid models-I?

Now use the **"Extrude"** command to create a solid model of the sketch



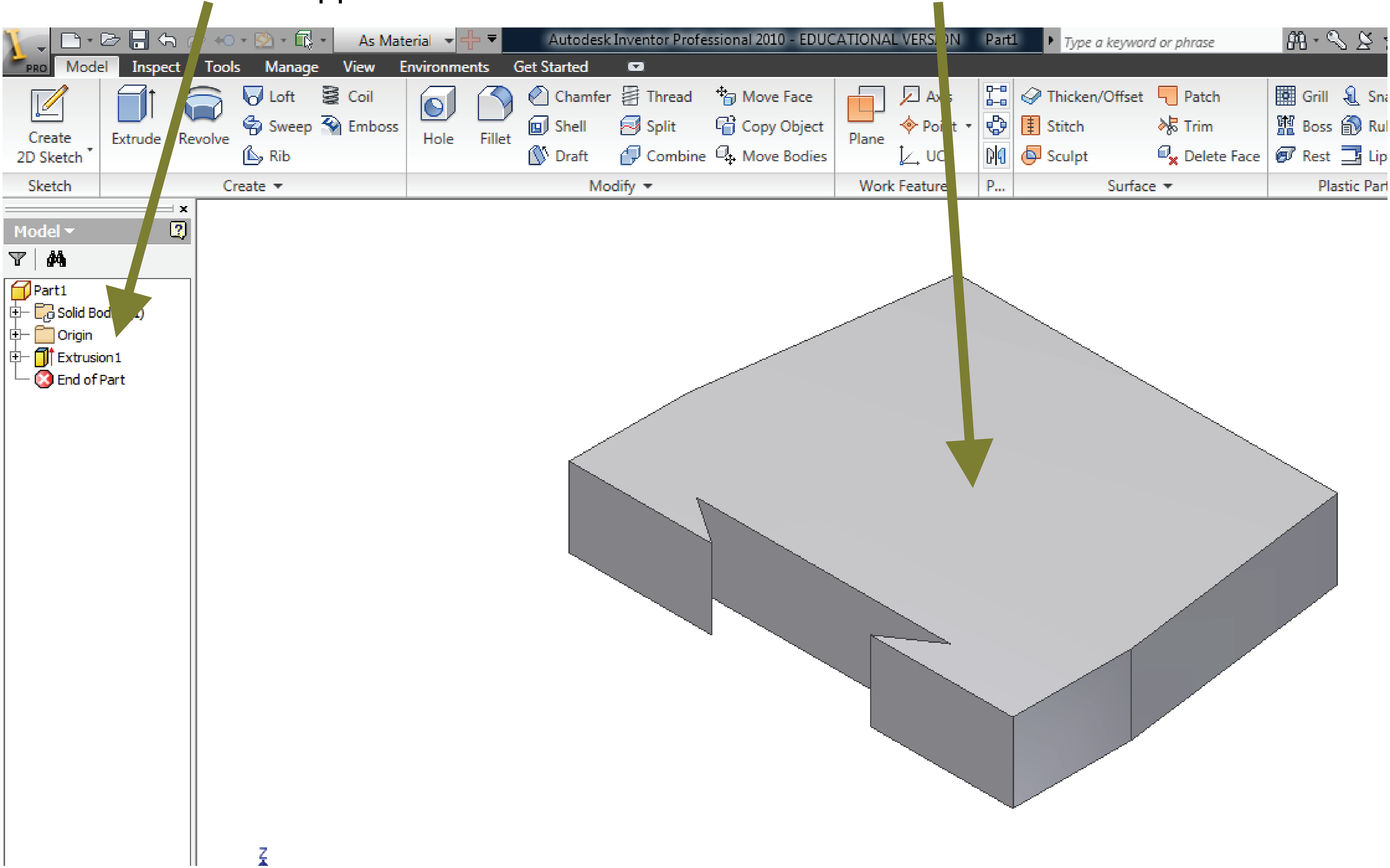
4. How to create solid models-I?

A window will appear, enter the extrusion distance *(by default its showing 10mm),* and click **OK**



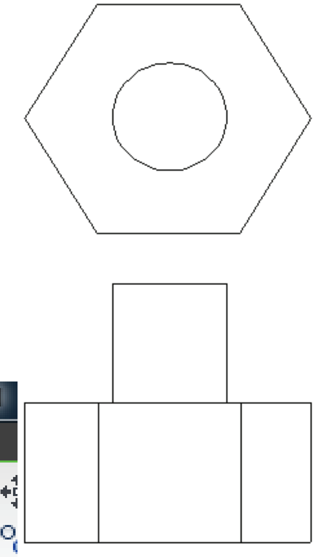
4. How to create solid models-I?

This extrusion appears in the browser window and solid model is created.



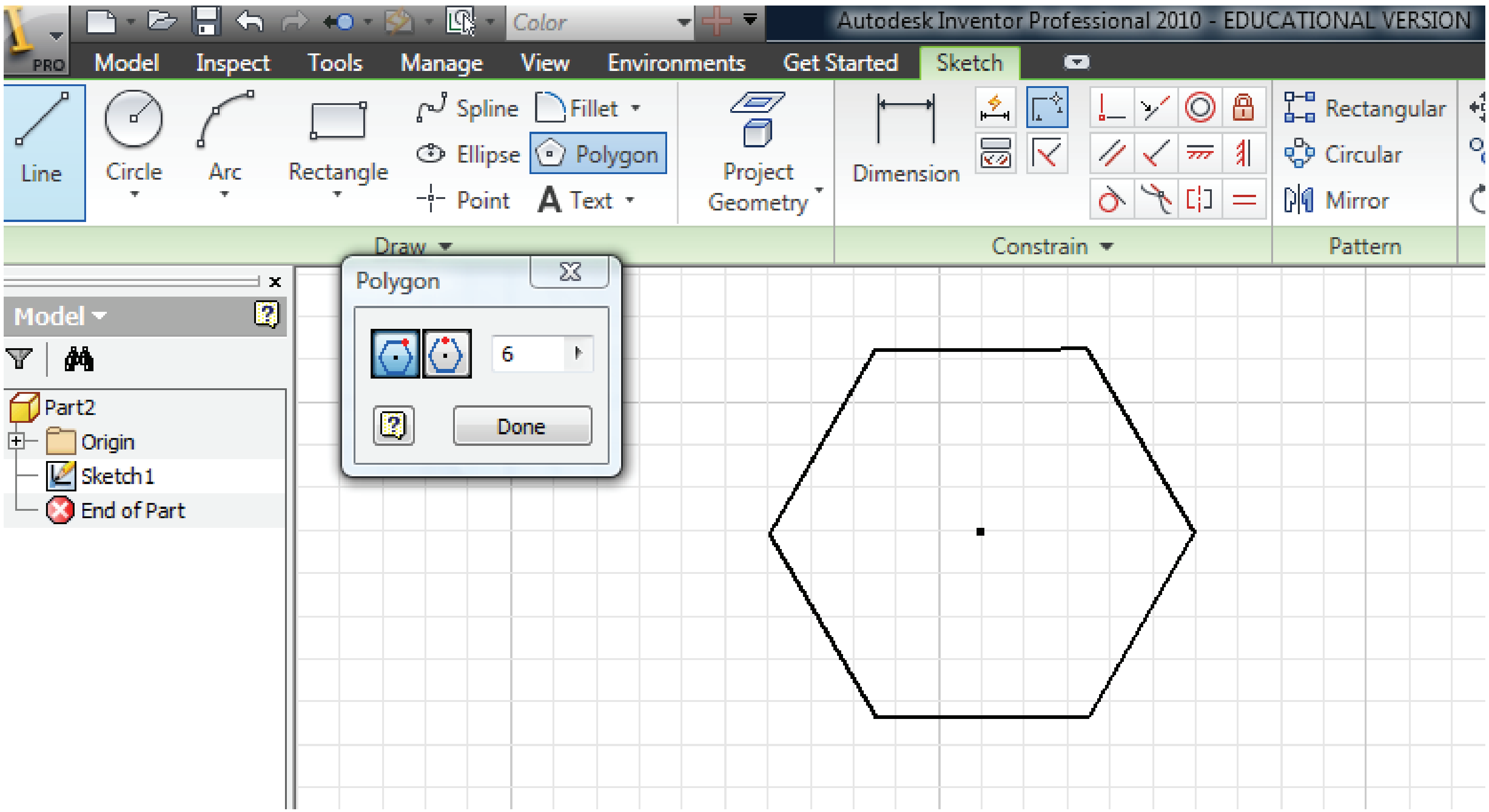
•Now use the view cube and navigation bar to explore this solid model

5. How to create a solid model using multiple extrusion?



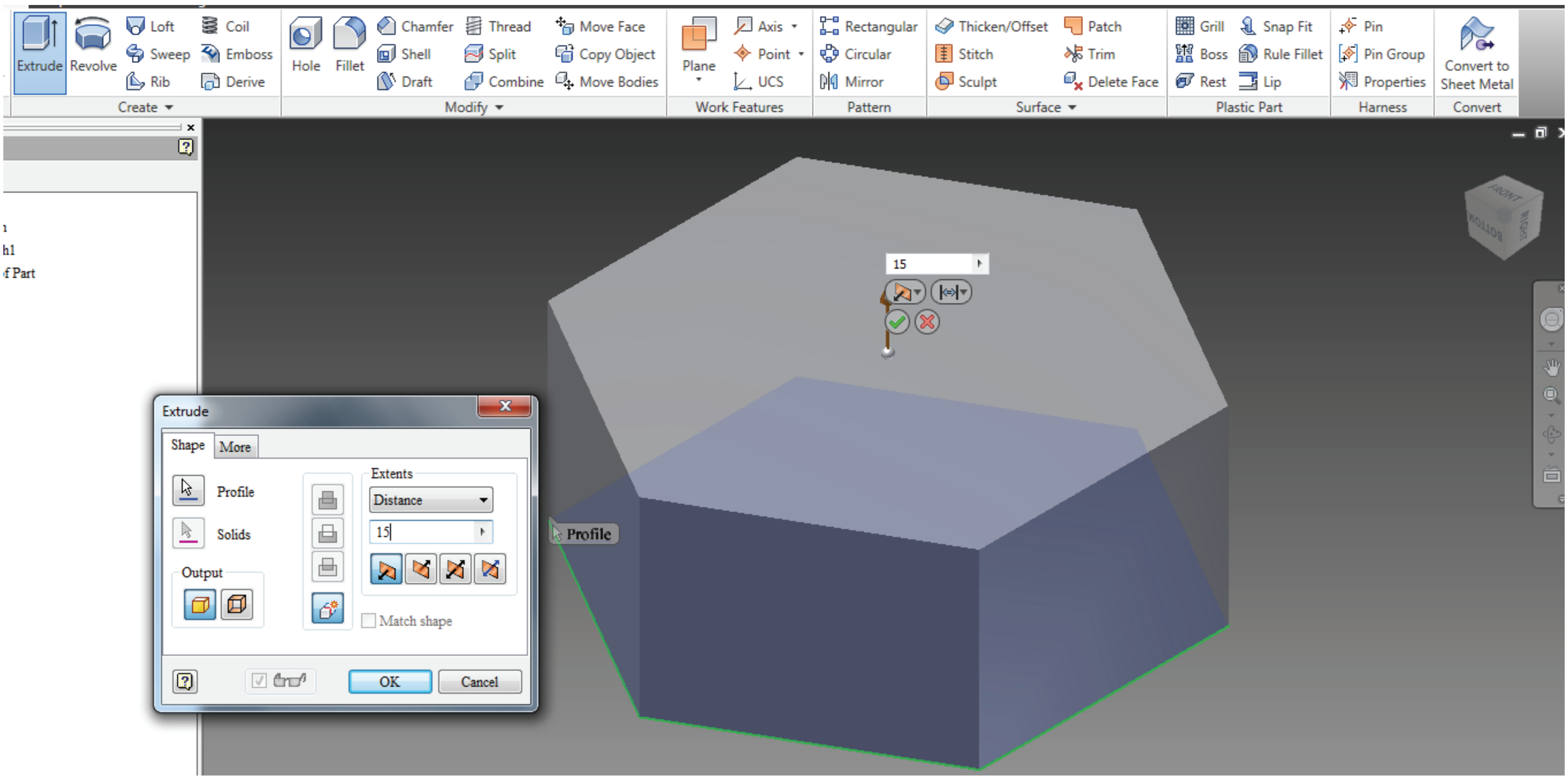
To create a cylinder at the top of a hexagonal prism

- Open a new part file and create sketch of a hexagon
- click finish sketch



5. How to create a solid model using multiple extrusion?

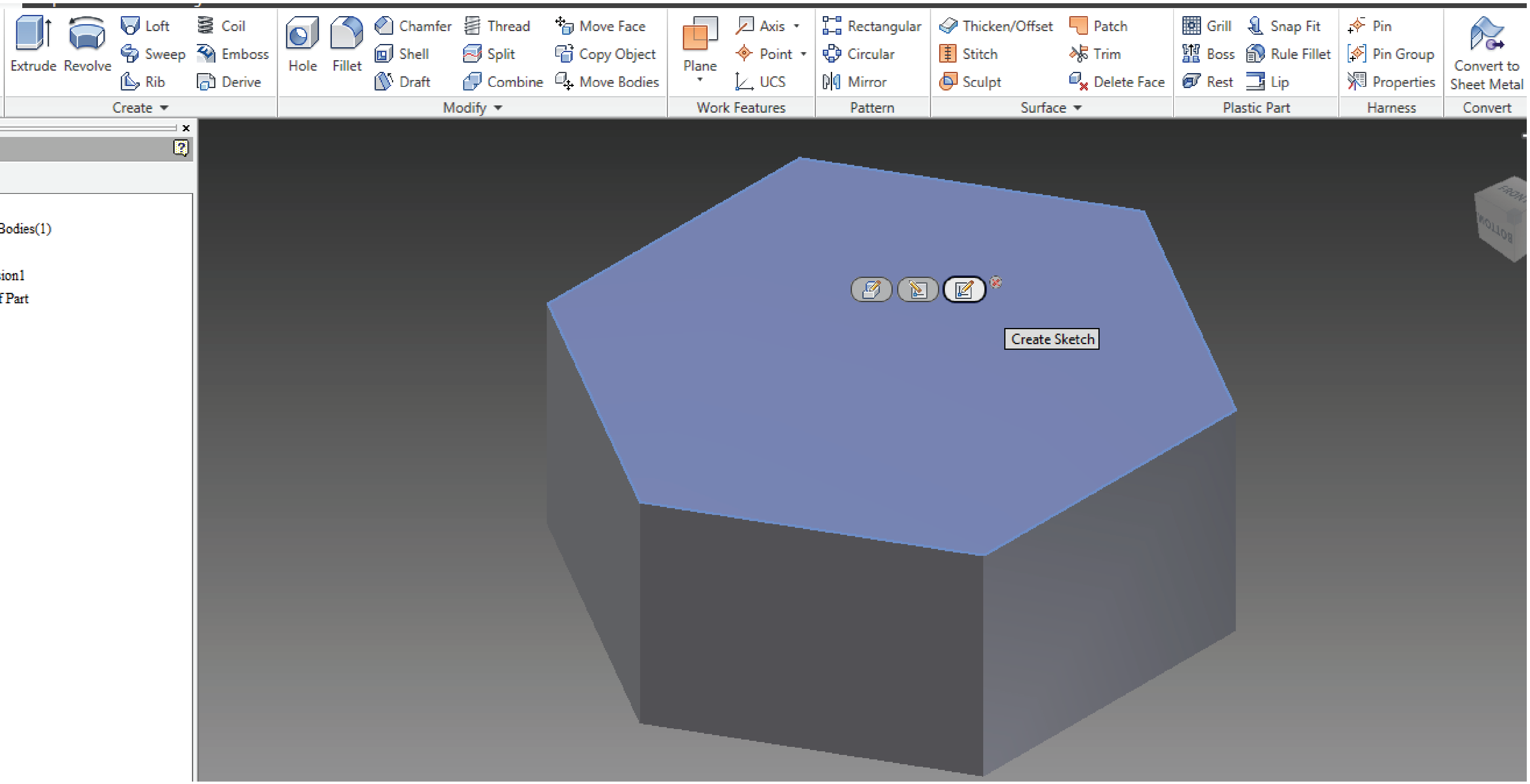
- In the Model tab click on the **Extrusion** command and extrude the hexagon to the required height and click **OK**



5. How to create a solid model using multiple extrusion?

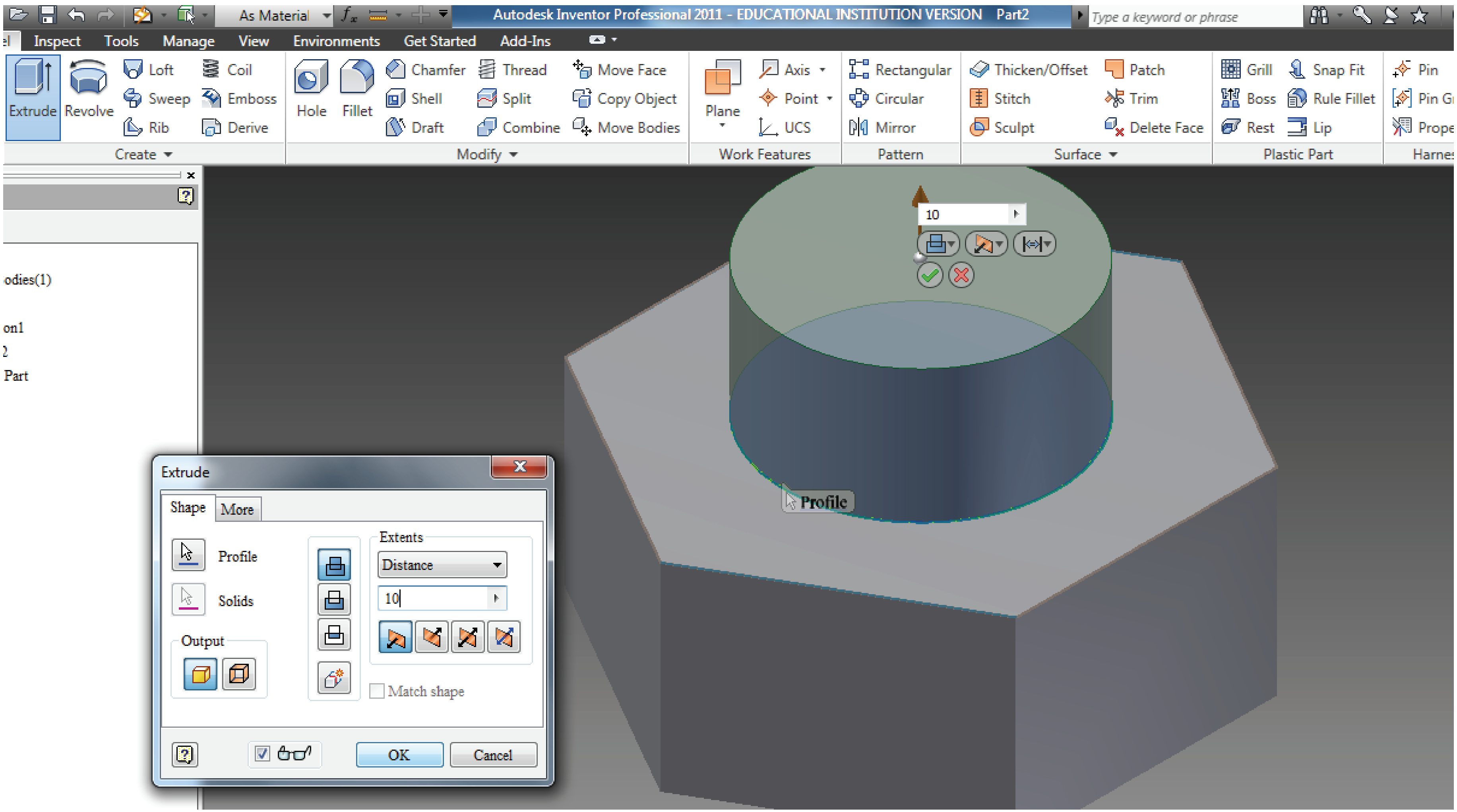
- Select the top face of hexahedral by double clicking that face.

- Click the option **Create Sketch** to activate the sketcher window



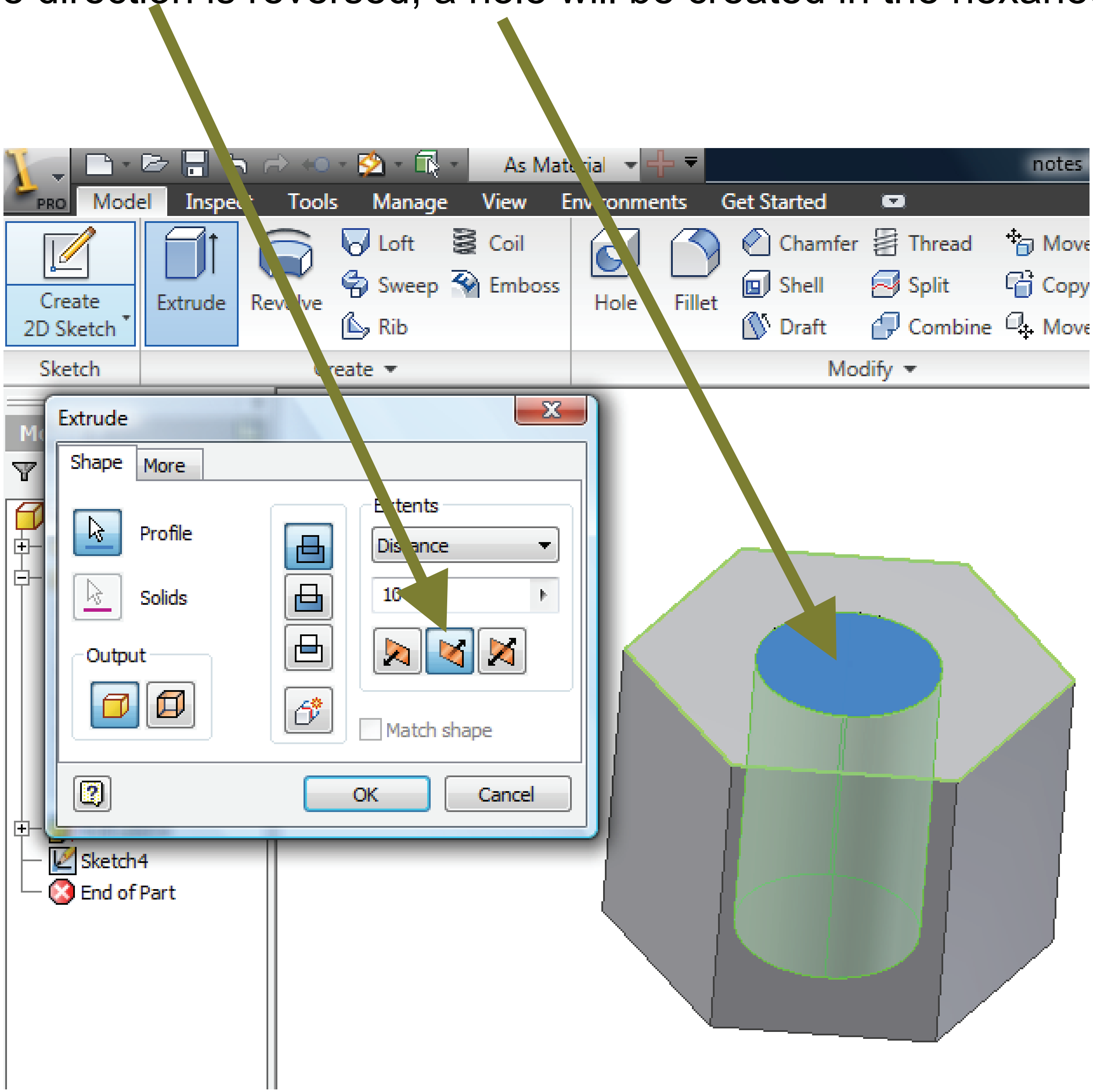
5. How to create a solid model using multiple extrusion?

- Create a sketch of a circle of required diameter and click on finish sketch
- In the Model tab click on the **Extrusion** command and extrude the hexahedron to the required height and click **OK**. The required solid model is complete.



5. How to create a solid model using multiple extrusion?

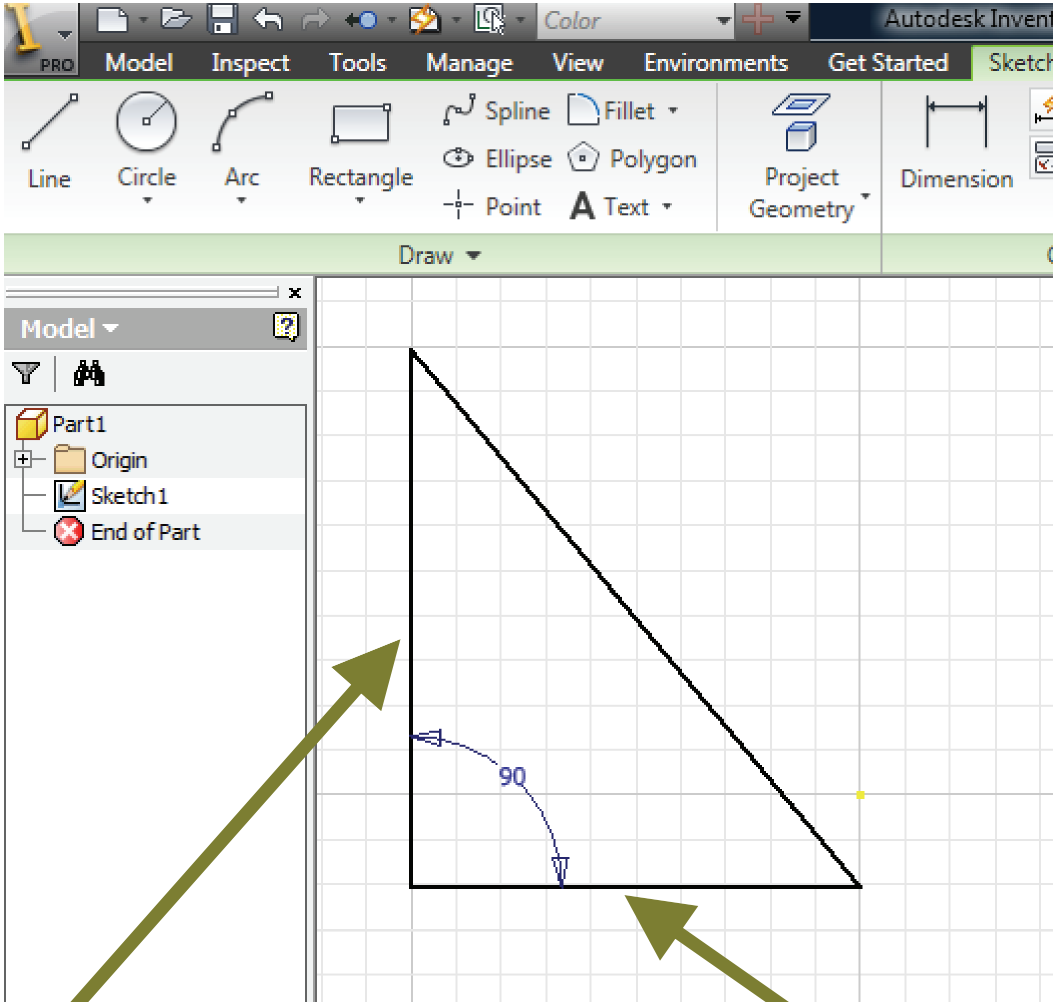
•If the extrude direction is reversed, a hole will be created in the hexahedron

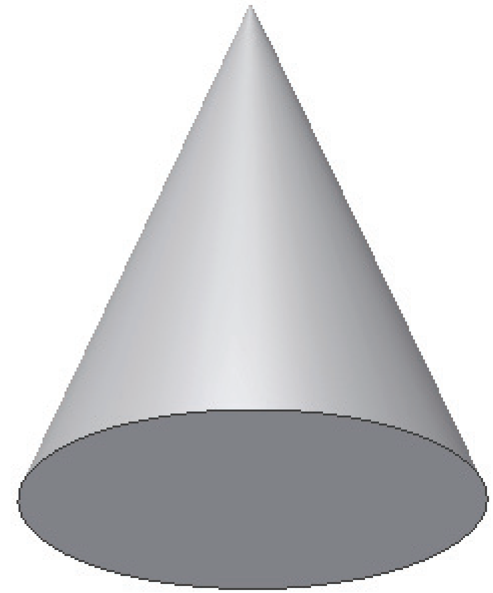


6. How to create a solid models-II?

- Create a sketch of a right angled triangle, finish sketch

How to create a Cone?



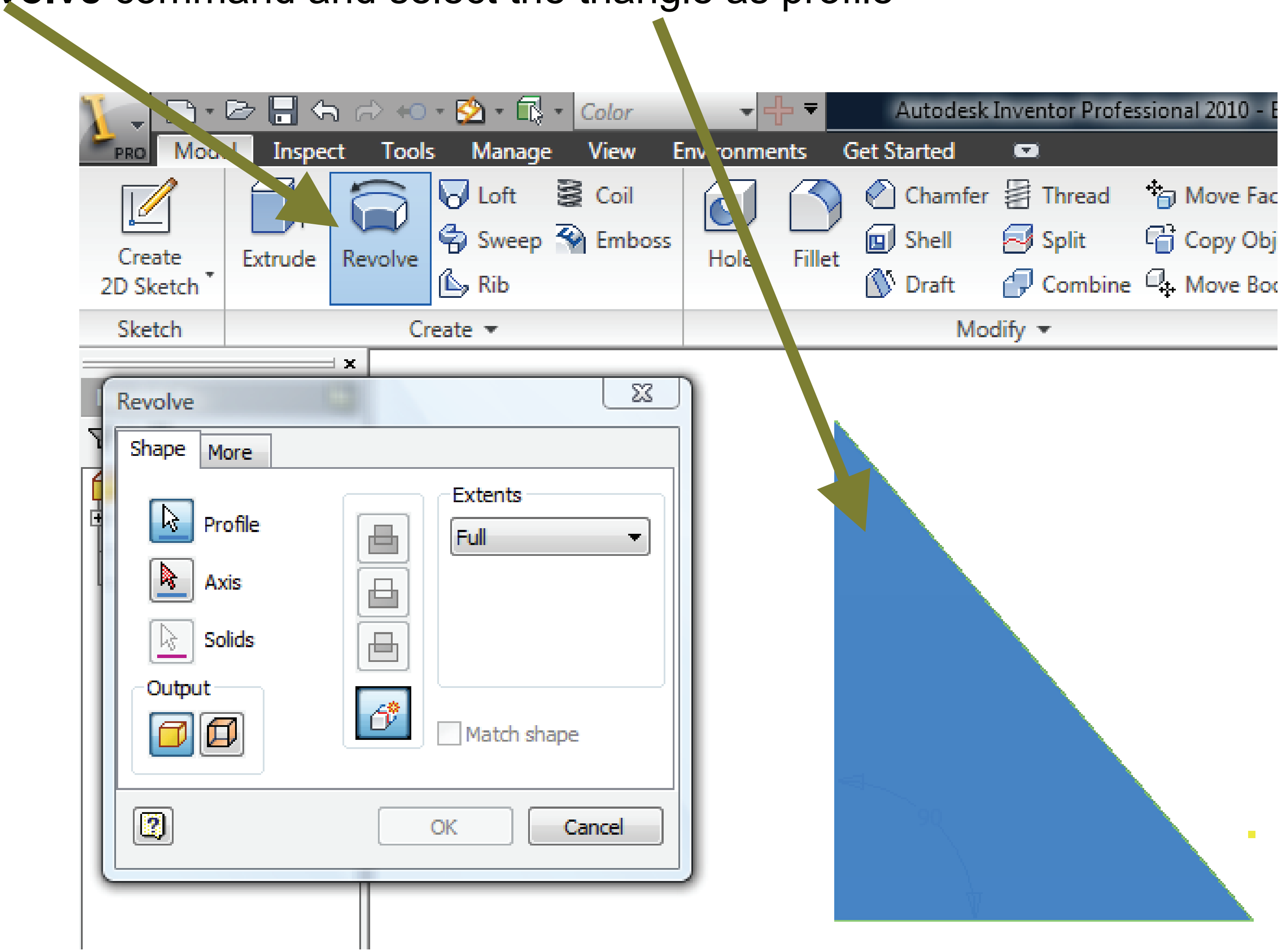


Cone height

Cone radius

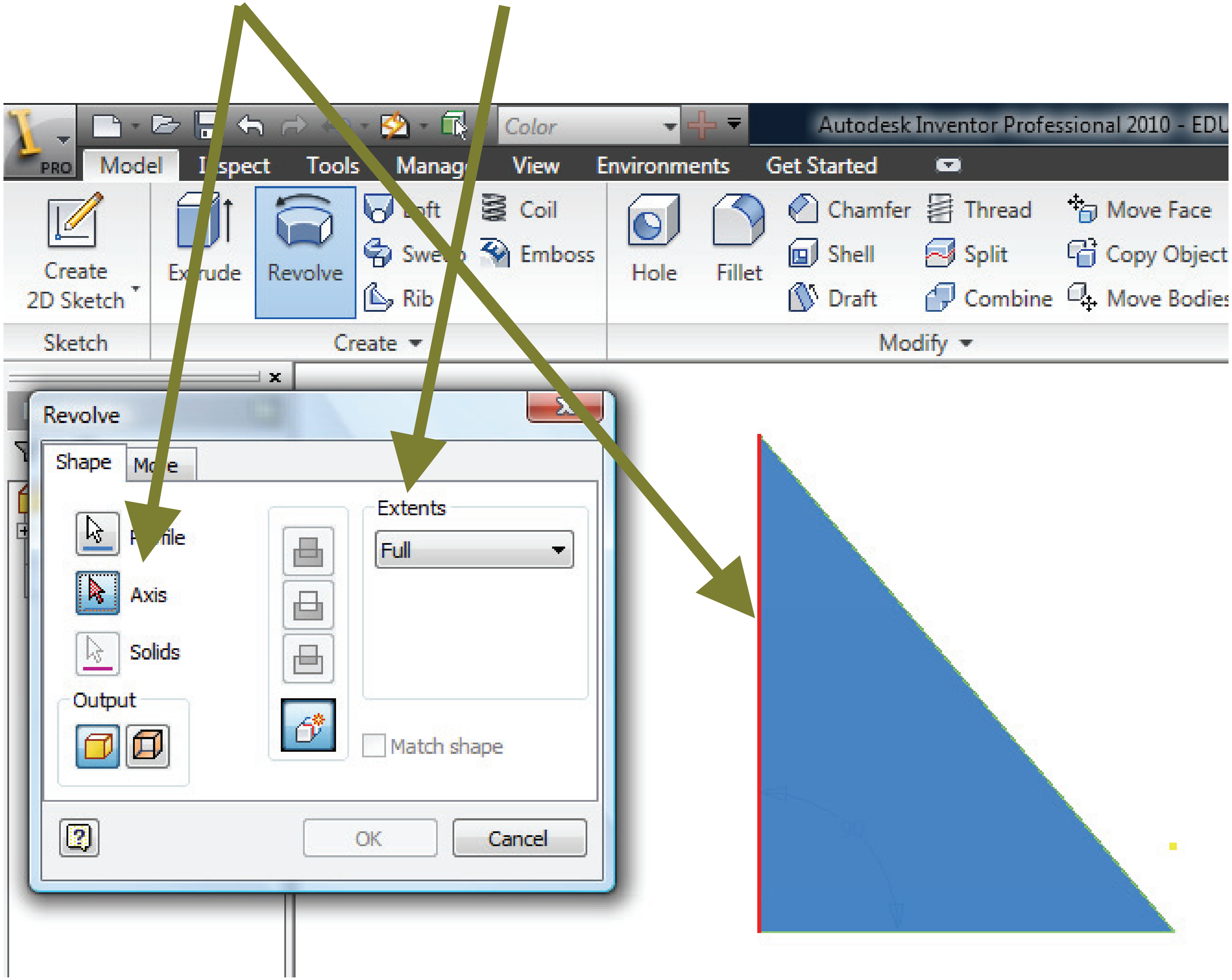
6. How to create a solid models-II?

•Use **Revolve** command and select the triangle as profile



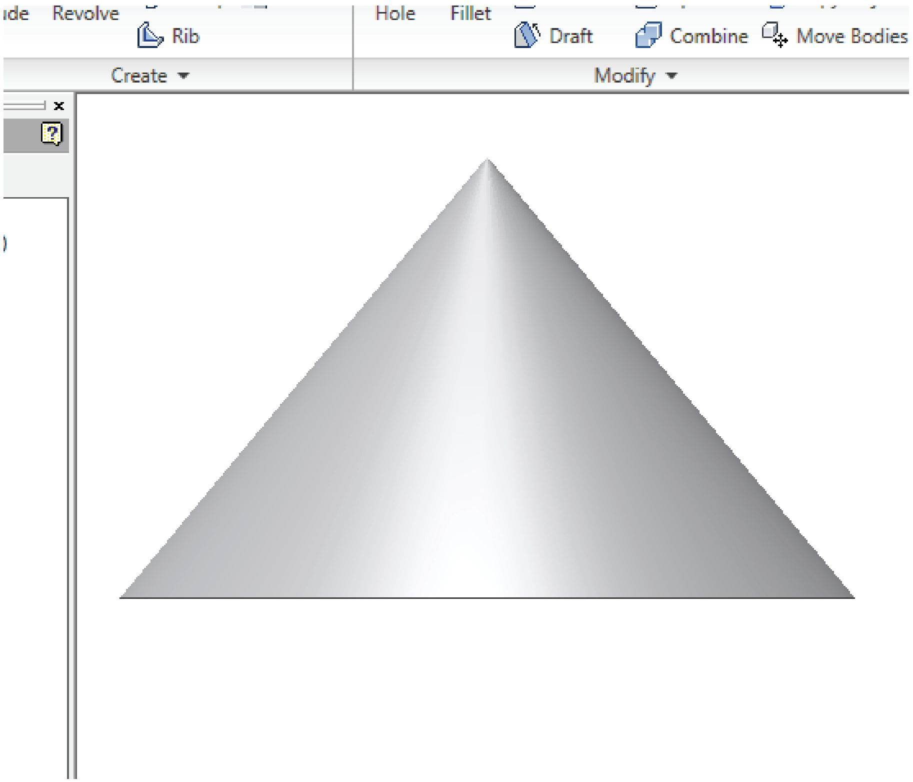
6. How to create a solid models-II?

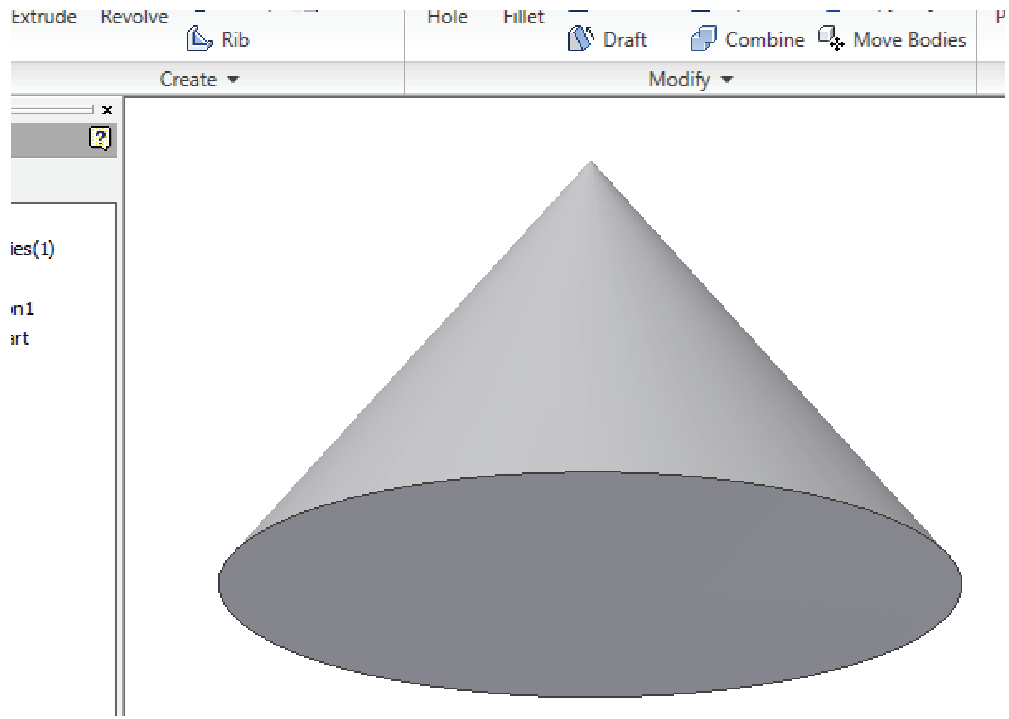
•Select vertical side as **Axis** and **Extents** as full (360 degrees) click **OK.**



6. How to create a solid models-II?

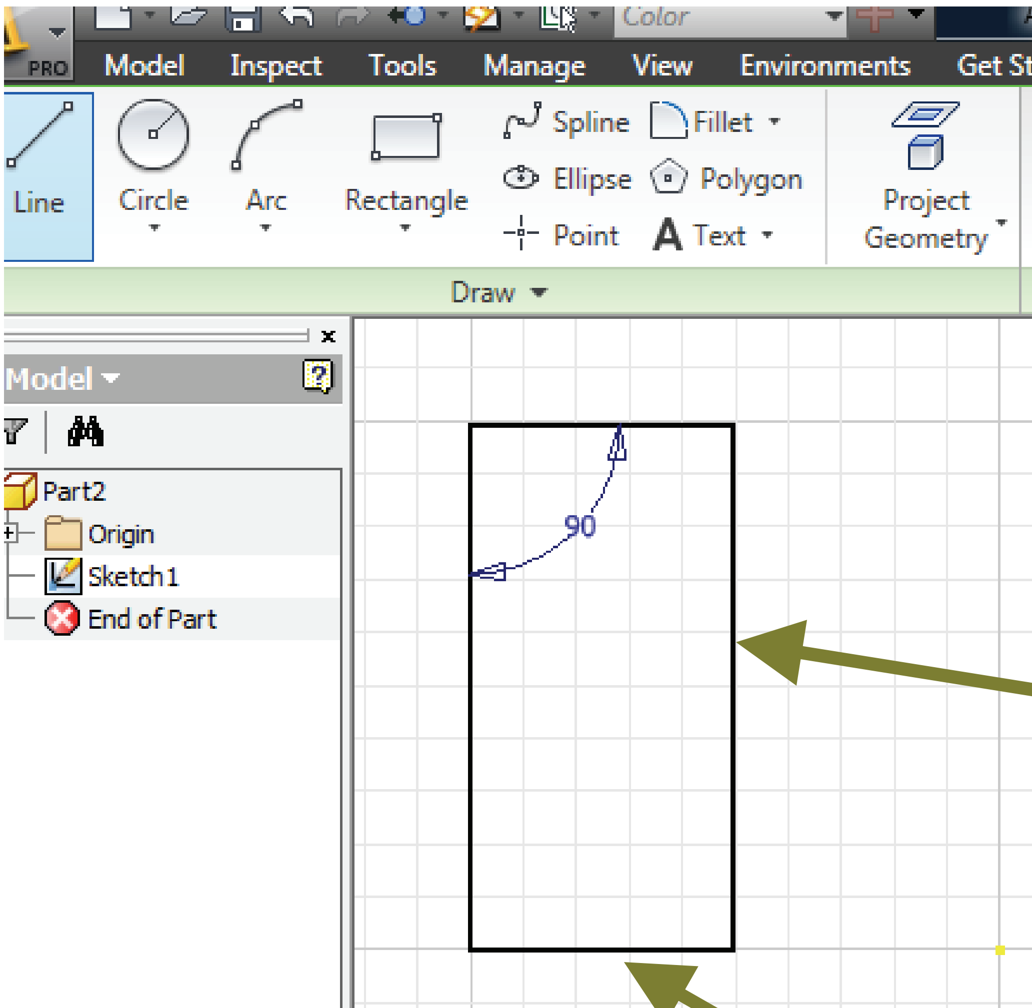
•A cone is thus created.



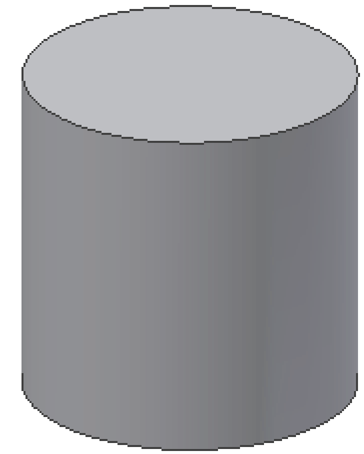


7. How to create a solid models-III?

•Create a sketch of a rectangle, finish sketch



How to create a Cylinder?

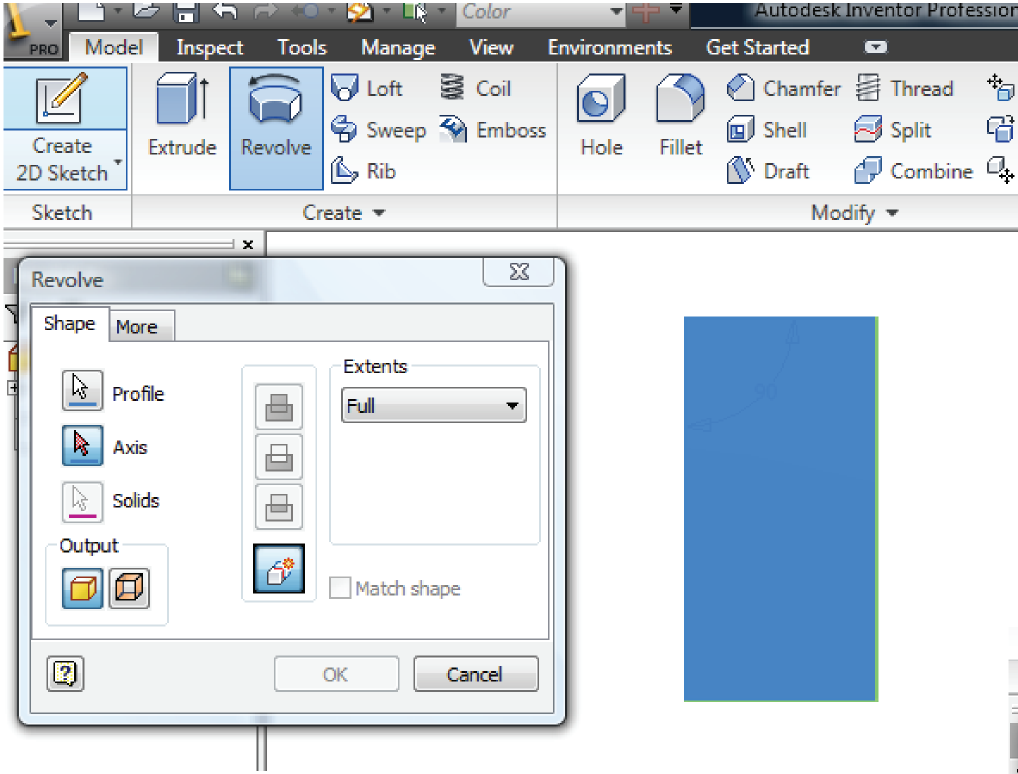


Cylinder height

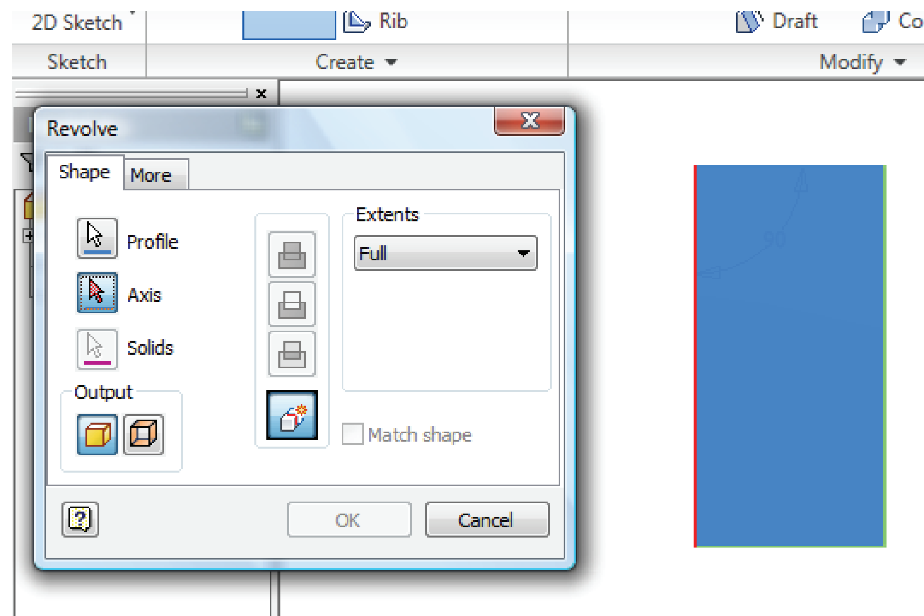
Cylinder diameter

7. How to create a solid models-III?

•Use **Revolve** command and select the rectangle as profile

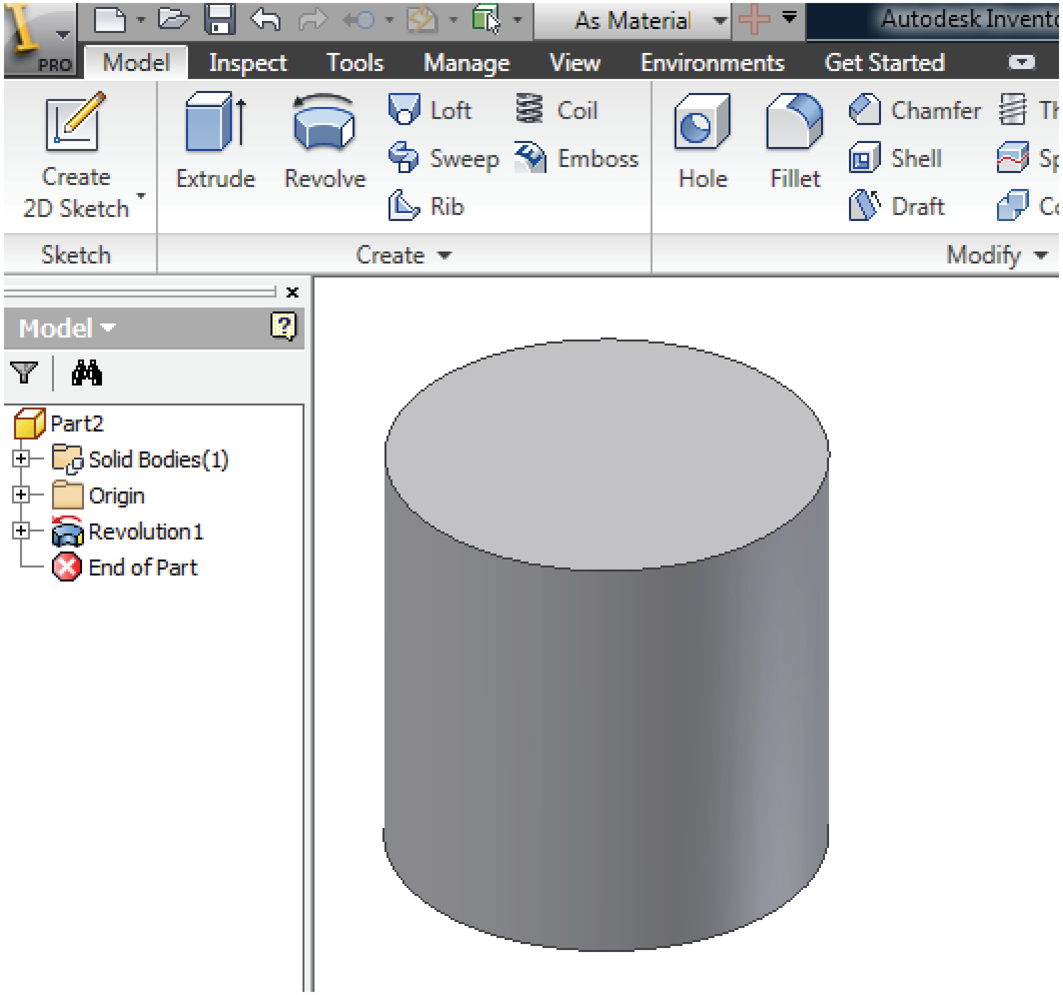


•Select vertical side as **Axis** and **Extents** as full (360 degrees) click **OK.**



7. How to create a solid models-III?

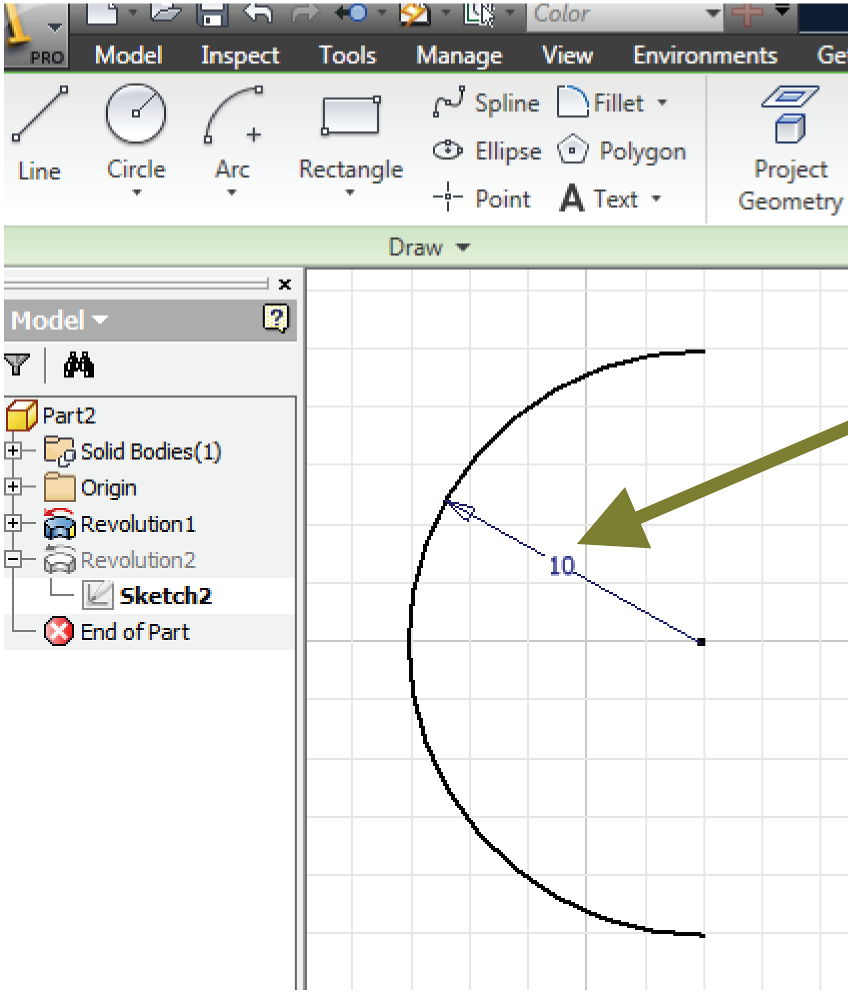
- A cylinder is thus created.



8. How to create a solid models-IV?

•Create a sketch of a semicircle.

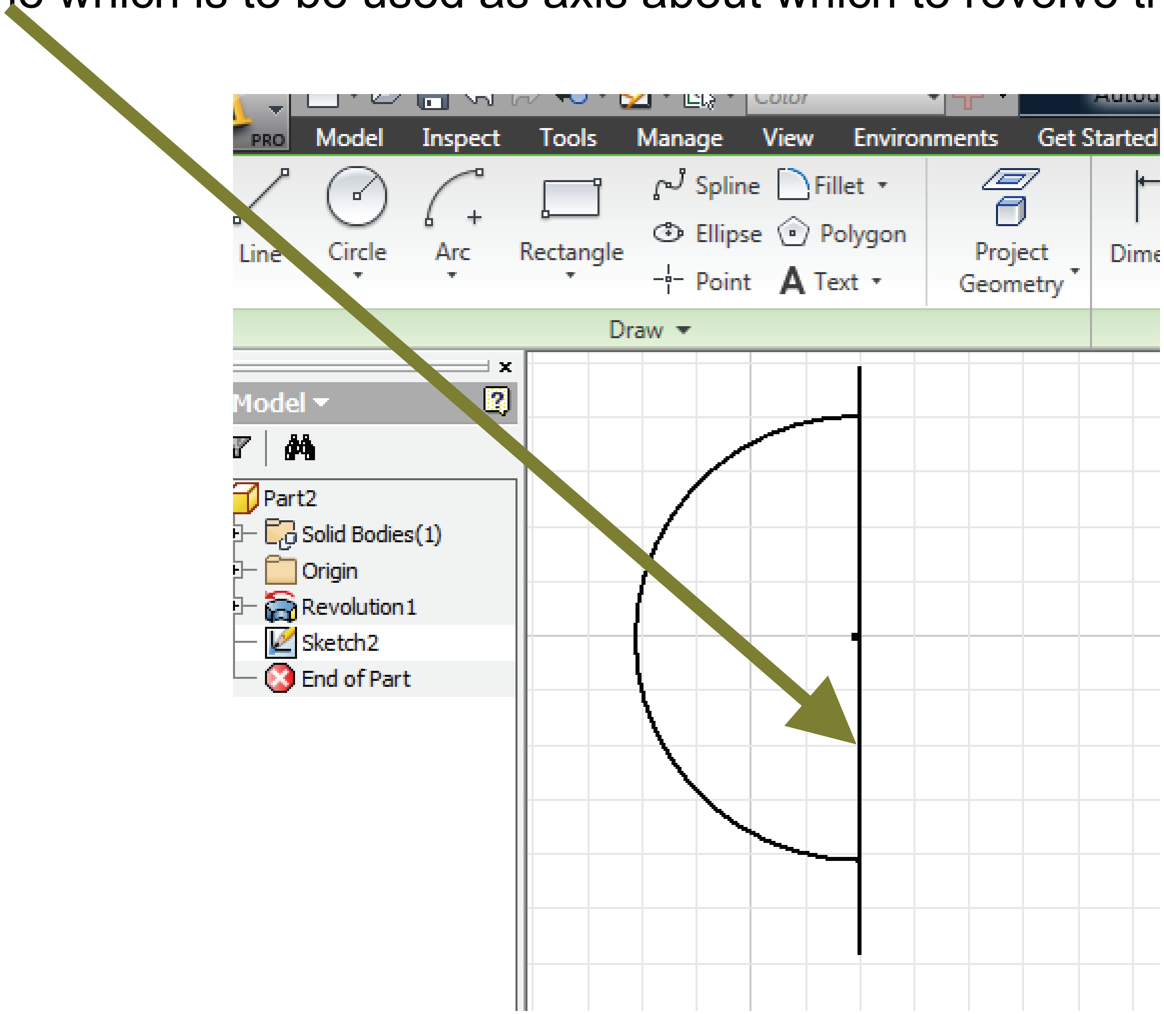
How to create a Sphere?



Sphere radius

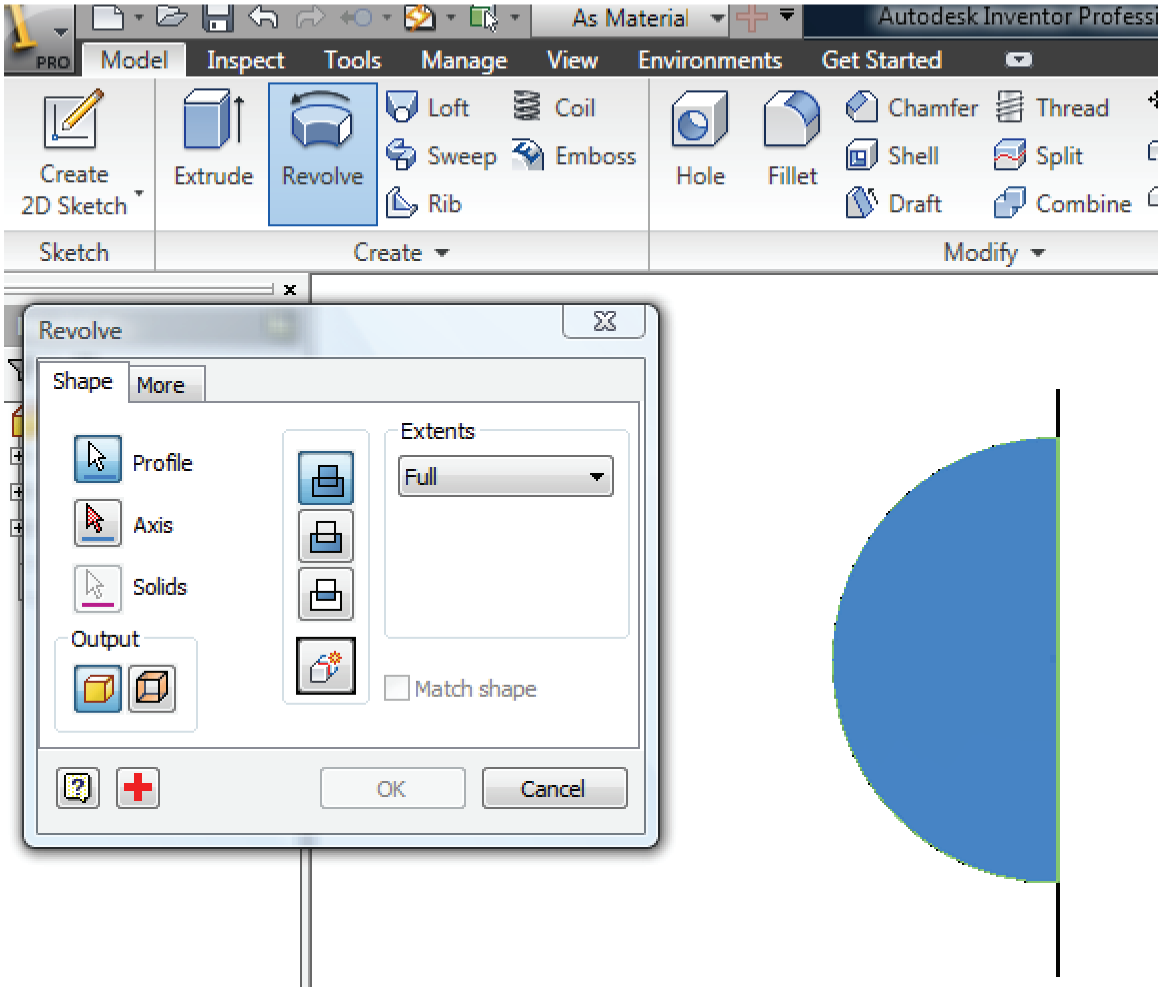
8. How to create a solid models-IV?

•Create a line which is to be used as axis about which to revolve the sketch, finish sketch



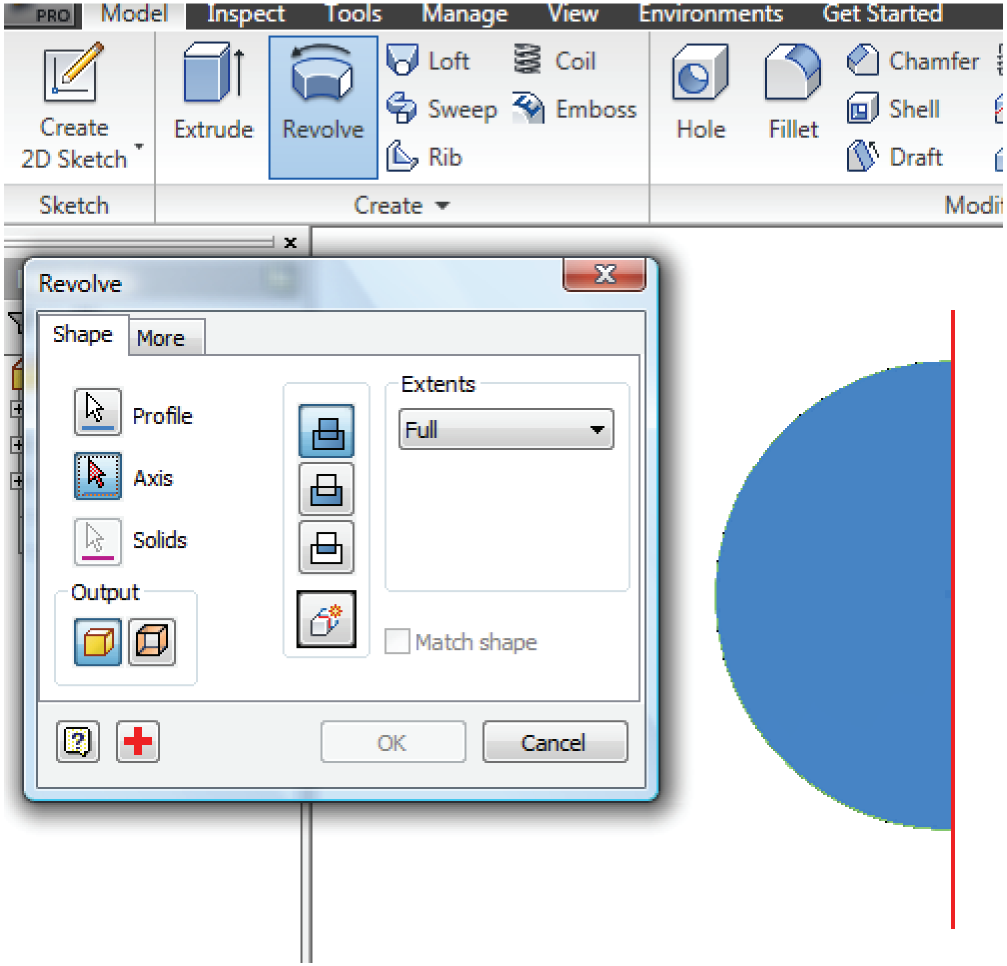
8. How to create a solid models-IV?

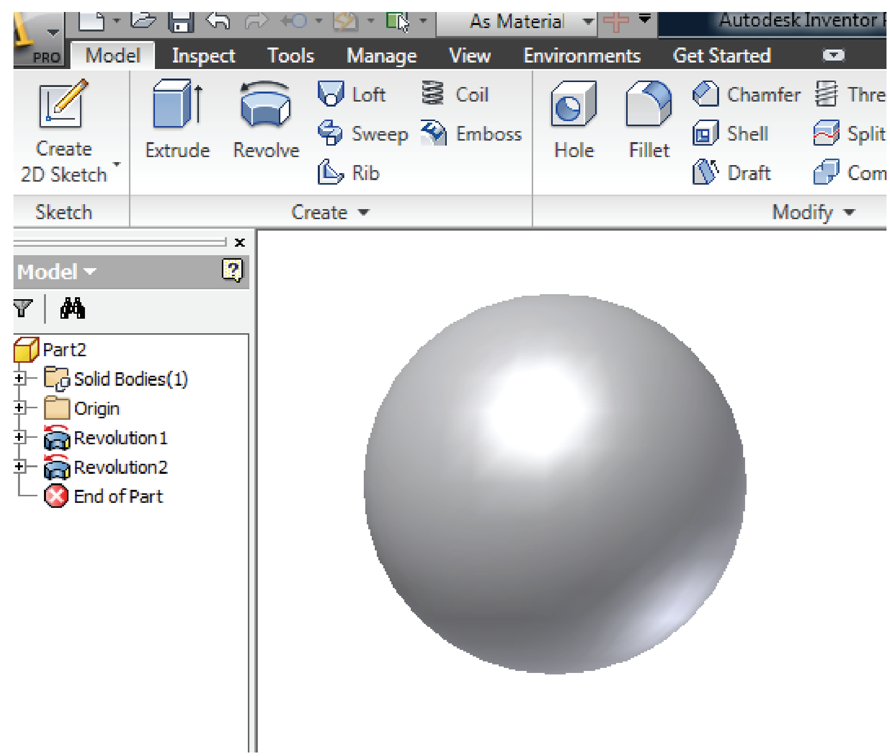
•Use **Revolve** command and select the semi-circle as profile



8. How to create a solid models-IV?

•Select the vertical line as **Axis** and **Extents** as full (360 degrees) click **OK**

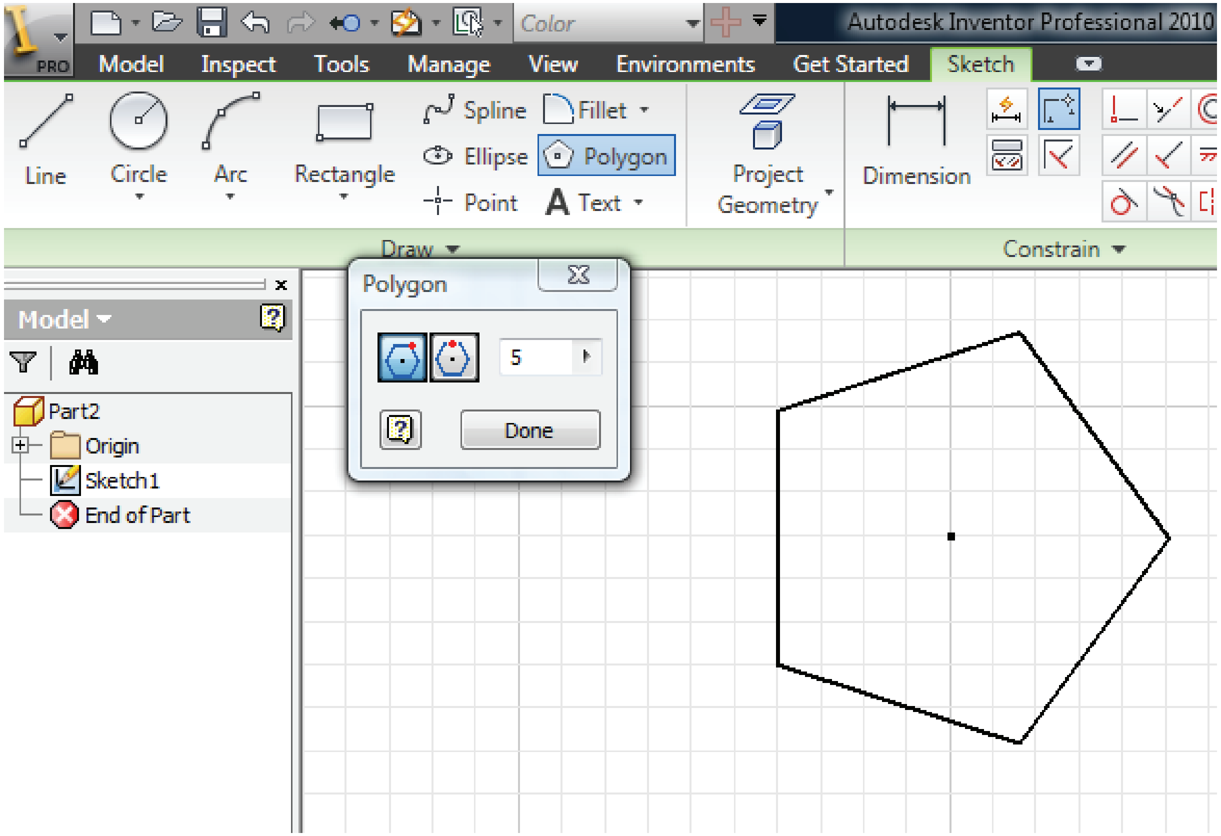




•A sphere is thus created.

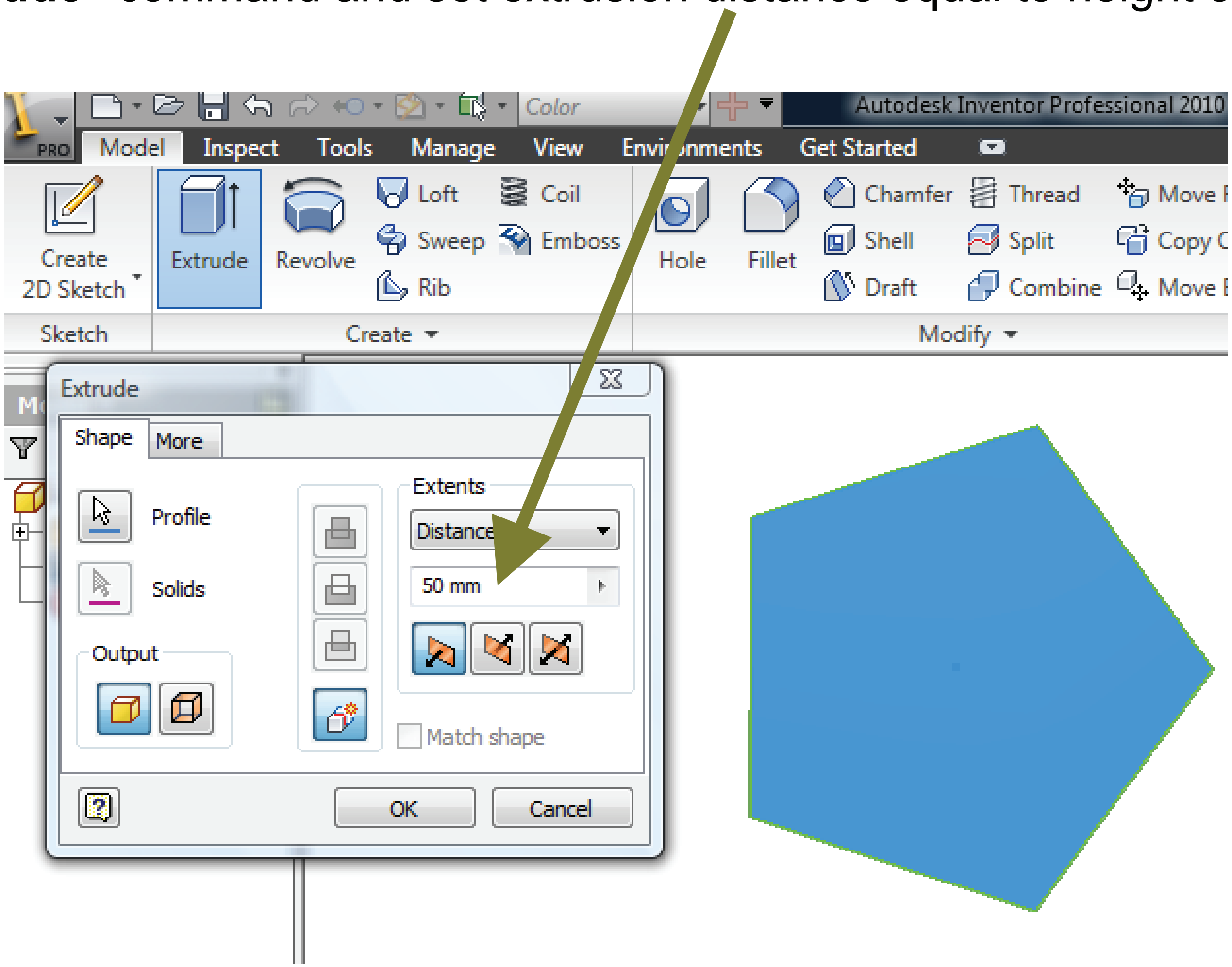
9. How to create a solid model-V?

- Open a new part file and create a sketch of the base of the pyramid, click **Done** and **finish sketch**



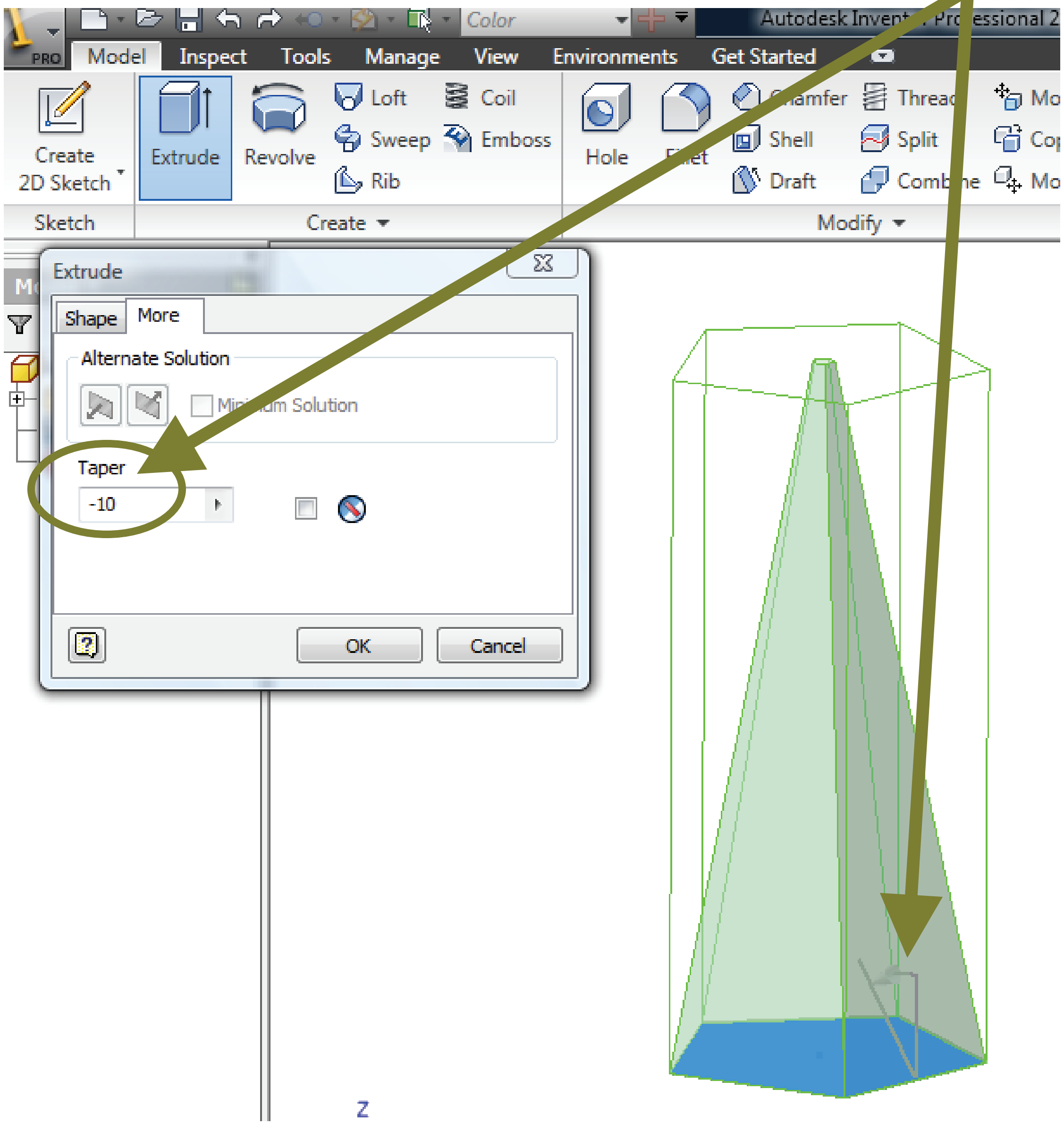
9. How to create a solid model-V?

- Use **“Extrude”** command and set extrusion distance equal to height of pyramid.



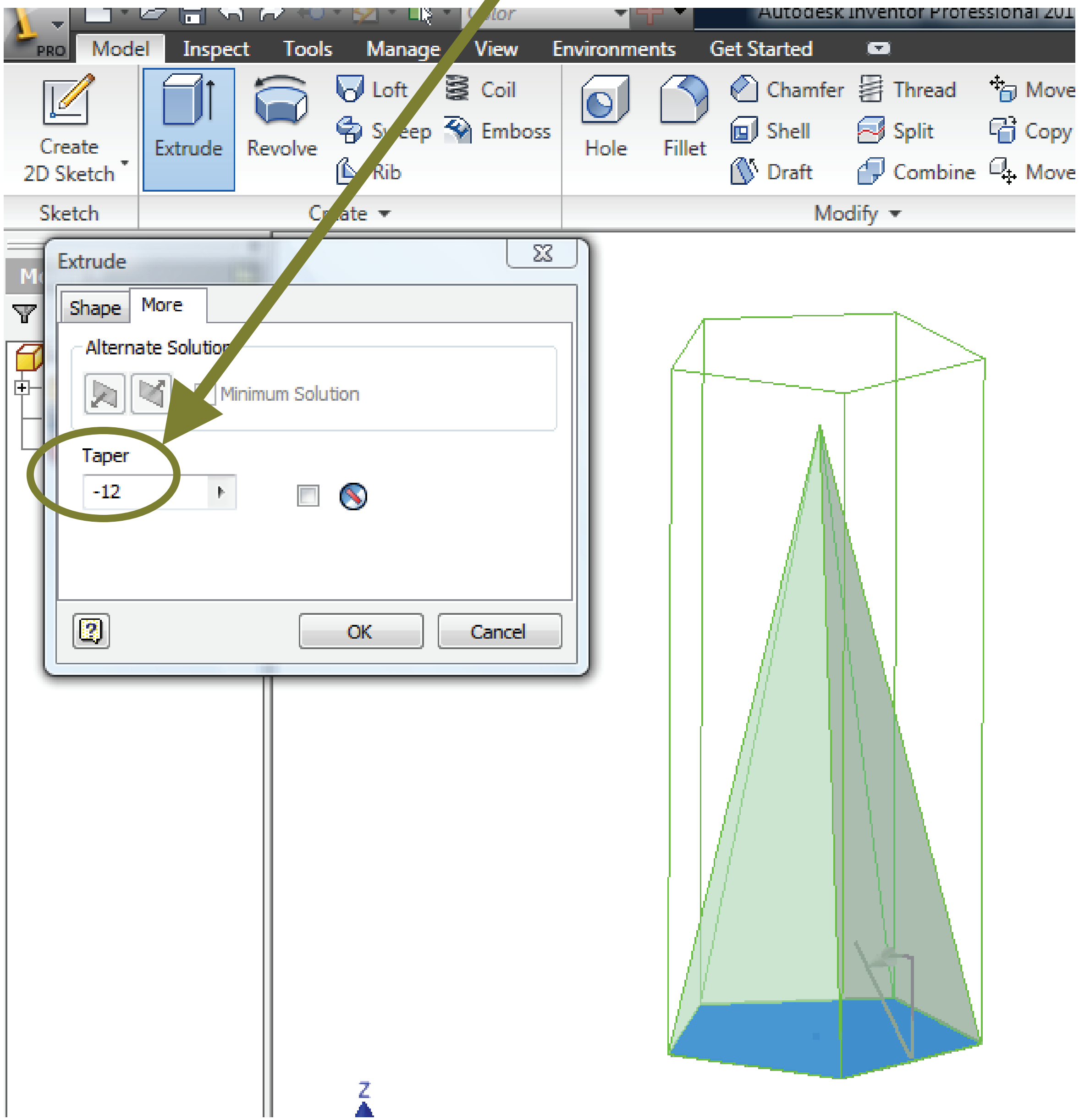
9. How to create a solid model-V?

- Now use **"More"** command in Extrusion and by trial and error set the taper angle so that pyramid is created



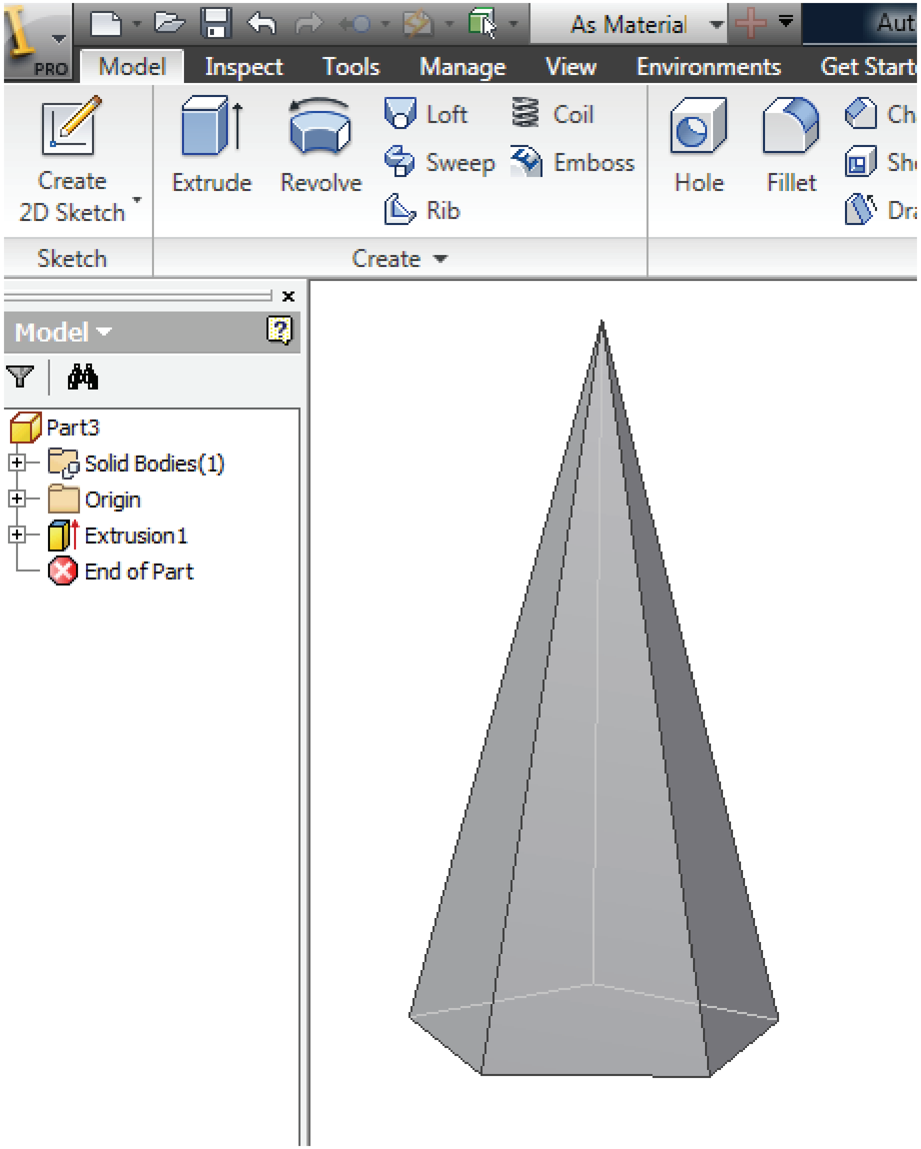
9. How to create a solid model -V?

continue trial and error to set the taper angle so that pyramid is created, click **OK**



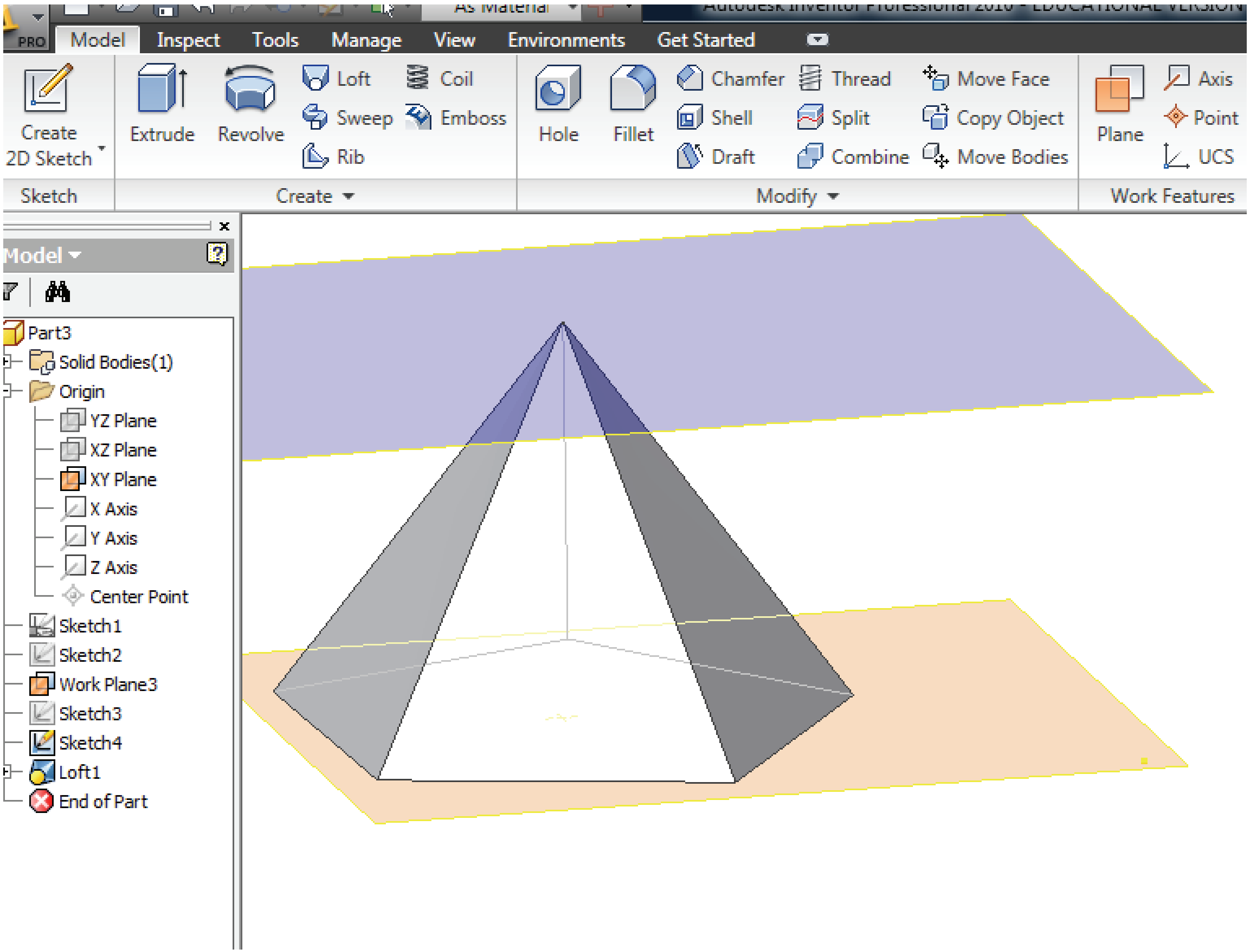
9. How to create a solid model -V?

Pyramid is thus created



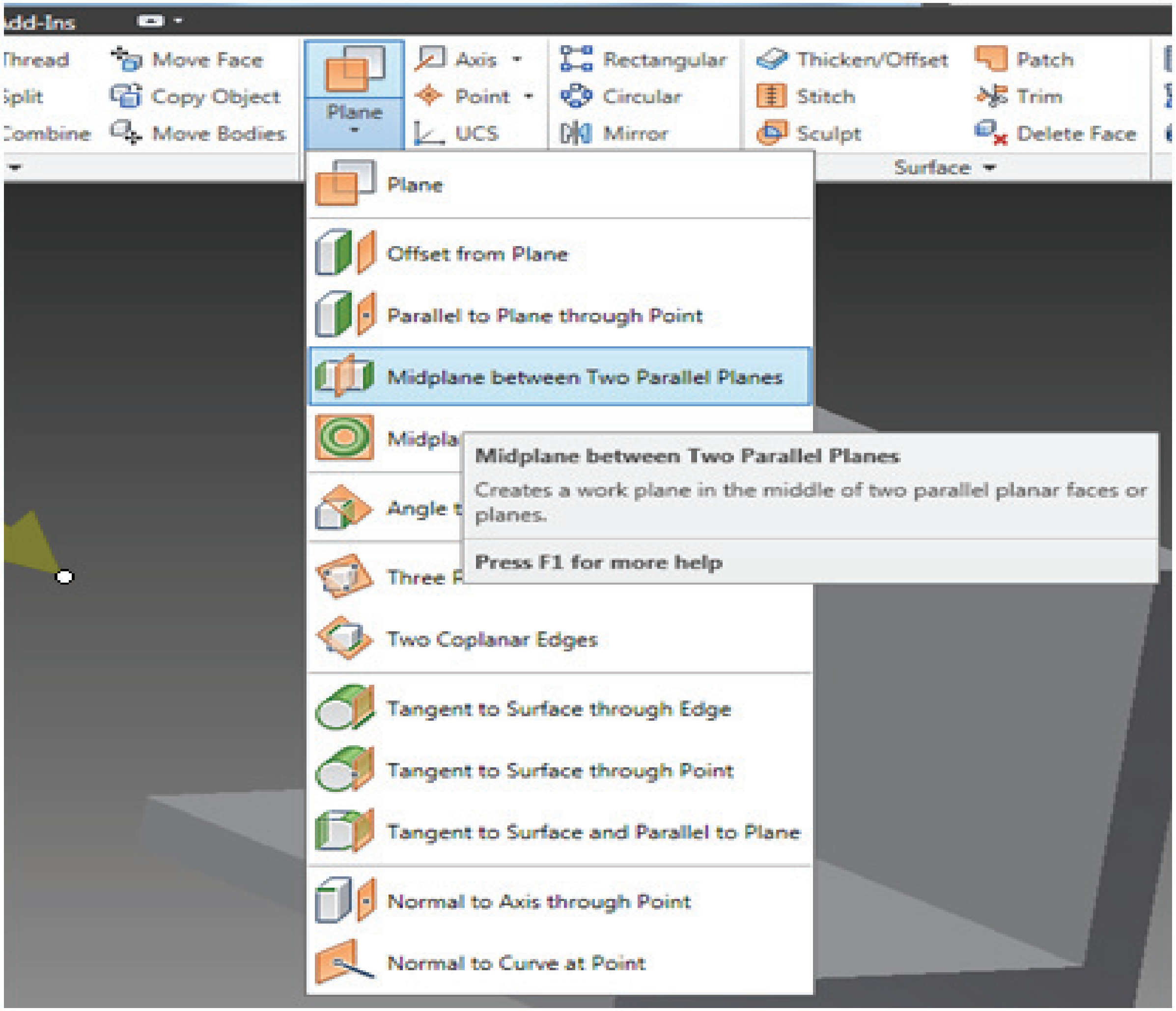
9. How to create a solid model -V?

Alternatively, create a point on offset plane and use **“Loft”** command to make pyramid



10. How to add Datum Feature to a sketch/solid model?

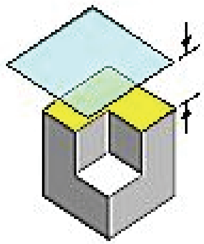
- **Datum features** are used during the construction of other features.
- Working planes, axes and curves are some of the common datum features.
- Datum features do not change the properties of the model
- In the **Model tab**, go to **Work Features** panel
- Select **Plane command** to define a work plane using feature like vertices, edges, faces,



10. How to add Datum Feature to a sketch/solid model?

Some examples of Work planes

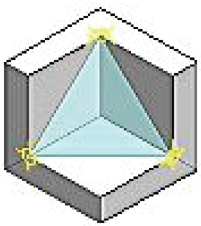
Adding
Working planes



Work plane offset from face

Select: A planar face. Click the edge of the face and drag in the direction of the offset. Enter a value in the edit box to specify the offset distance.

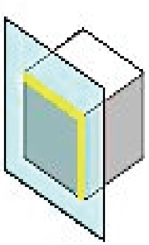
Result: Creates a work plane parallel to the selected face at the specified offset distance.



3-point work plane

Select: Any three points (endpoints, intersections, midpoints, work points).

Result: Positive X axis is directed from first point to second point. Positive Y axis is perpendicular to the positive X axis through the third point.



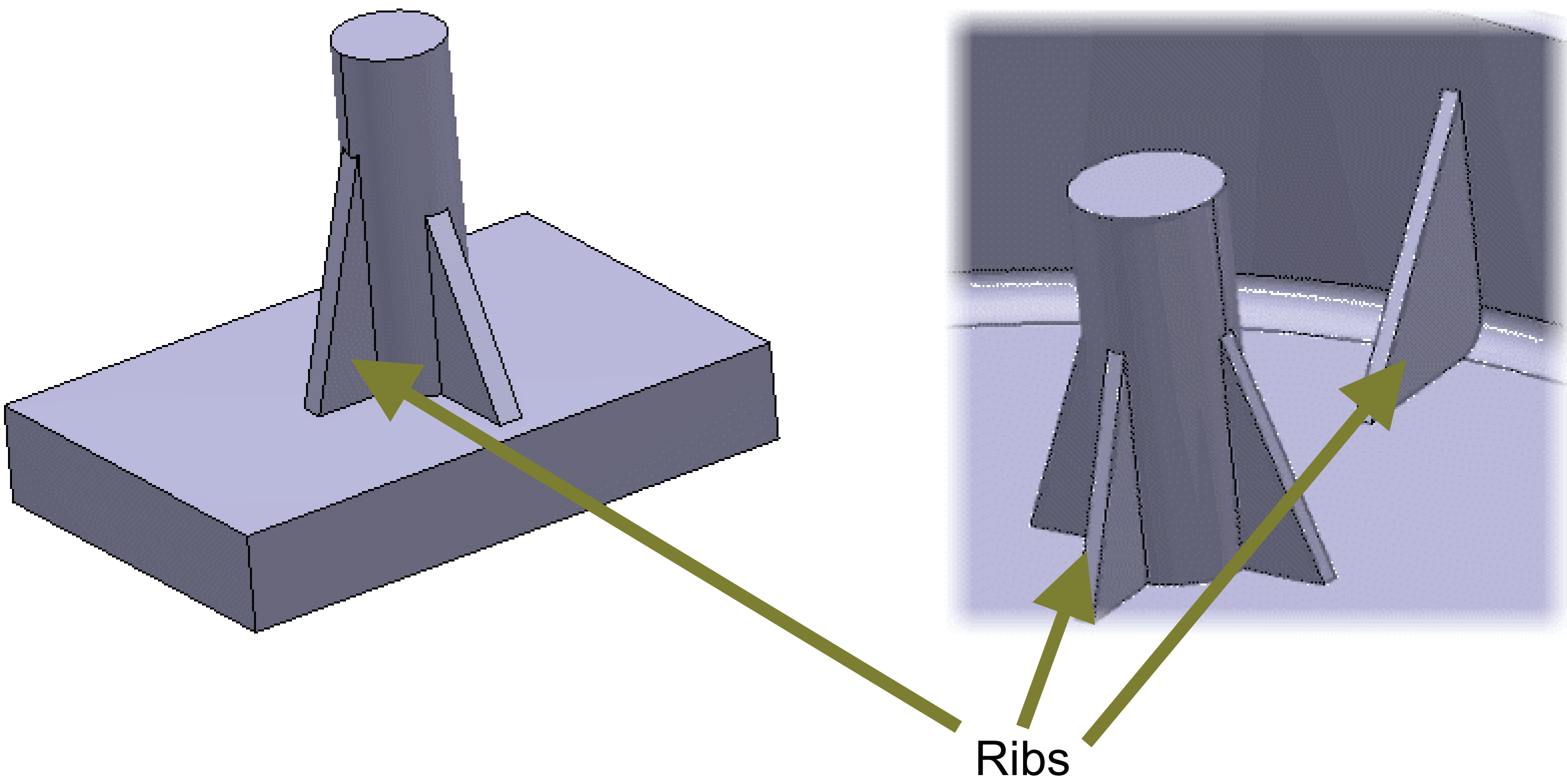
Work plane through two coplanar edges

Select: Two coplanar edges.

Result: The positive X axis is oriented along the first selected edge.

11. How to create a rib in a solid model ?

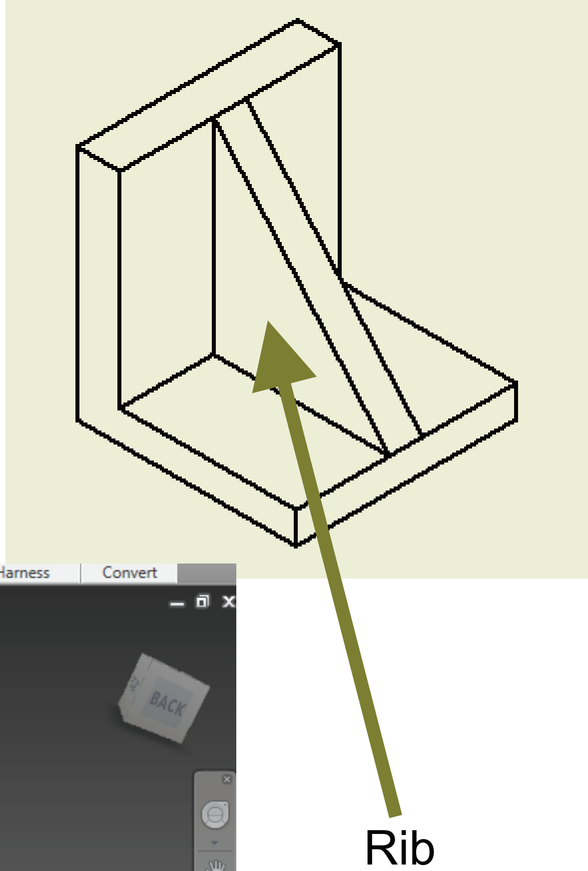
- Use of ribs or stiffeners is one of the most common ways of adding stiffness to localized areas of a structure.
- Ribs are usually attached to the surface of an existing structure by casting, welding, gluing or bolting.

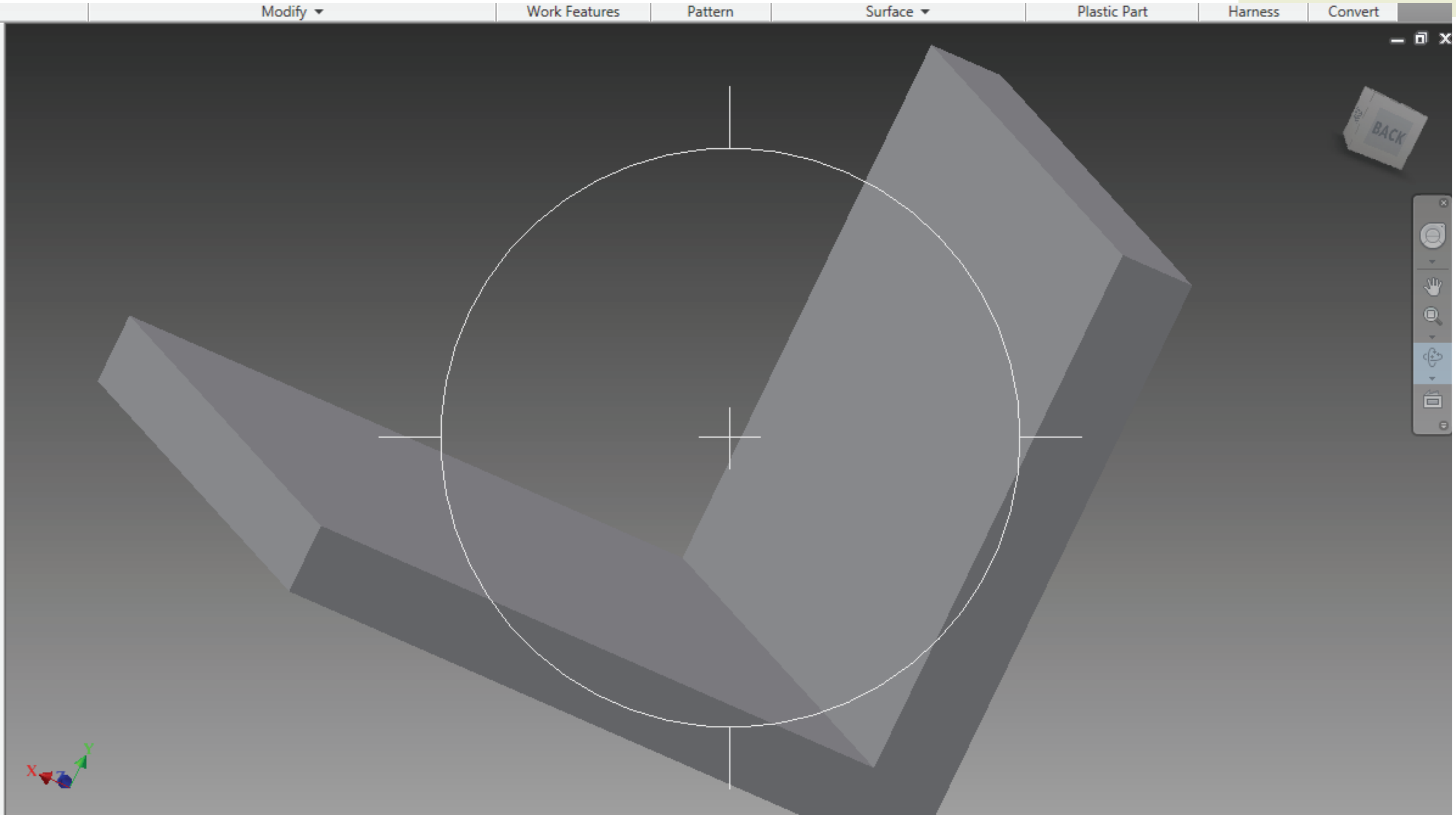


11. How to create a rib in a solid model ?

•Create a solid model as shown below.

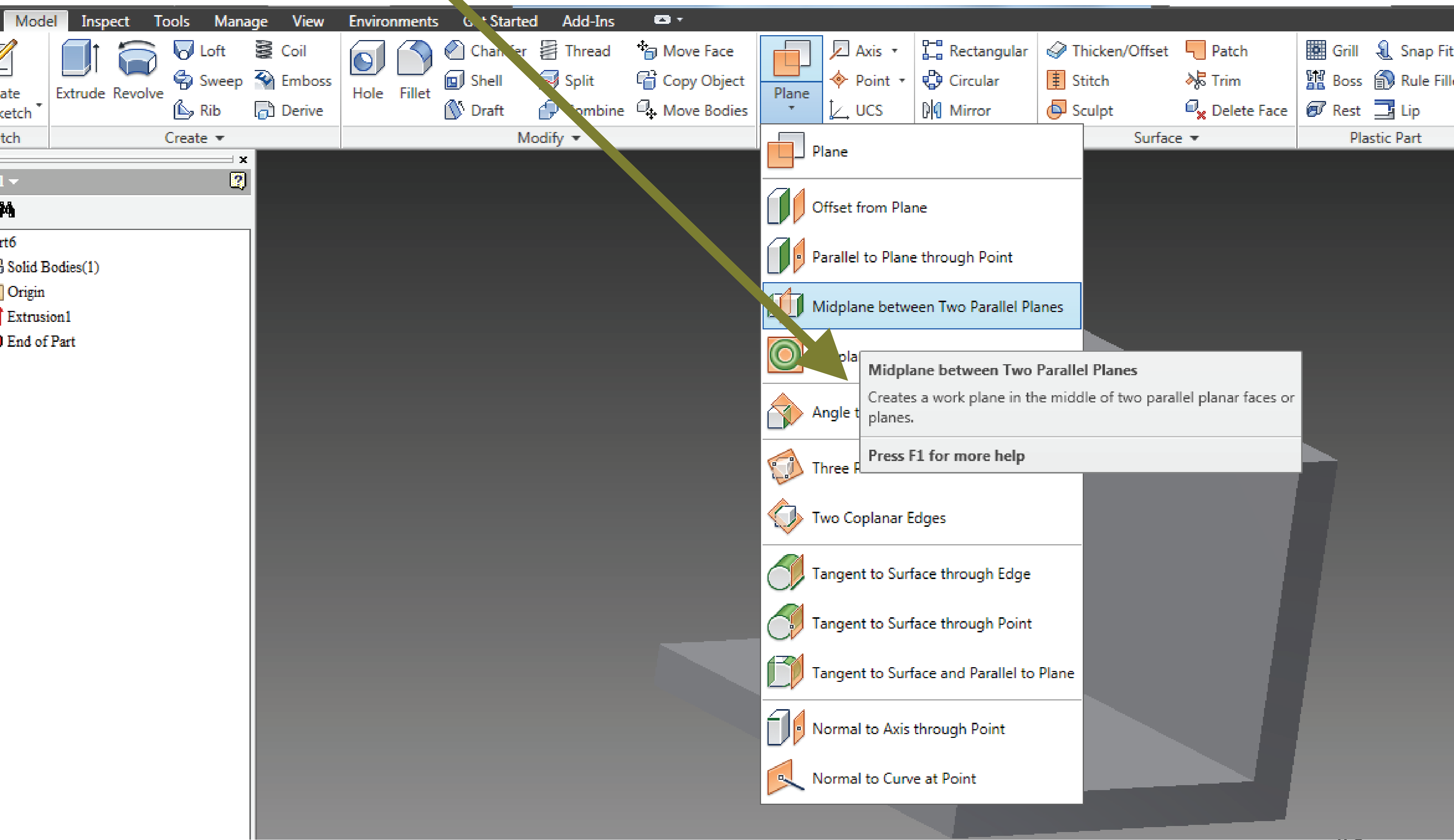
- Click Free Orbit and rotate the object as shown below
- Click Done/Esc to come out of orbit command





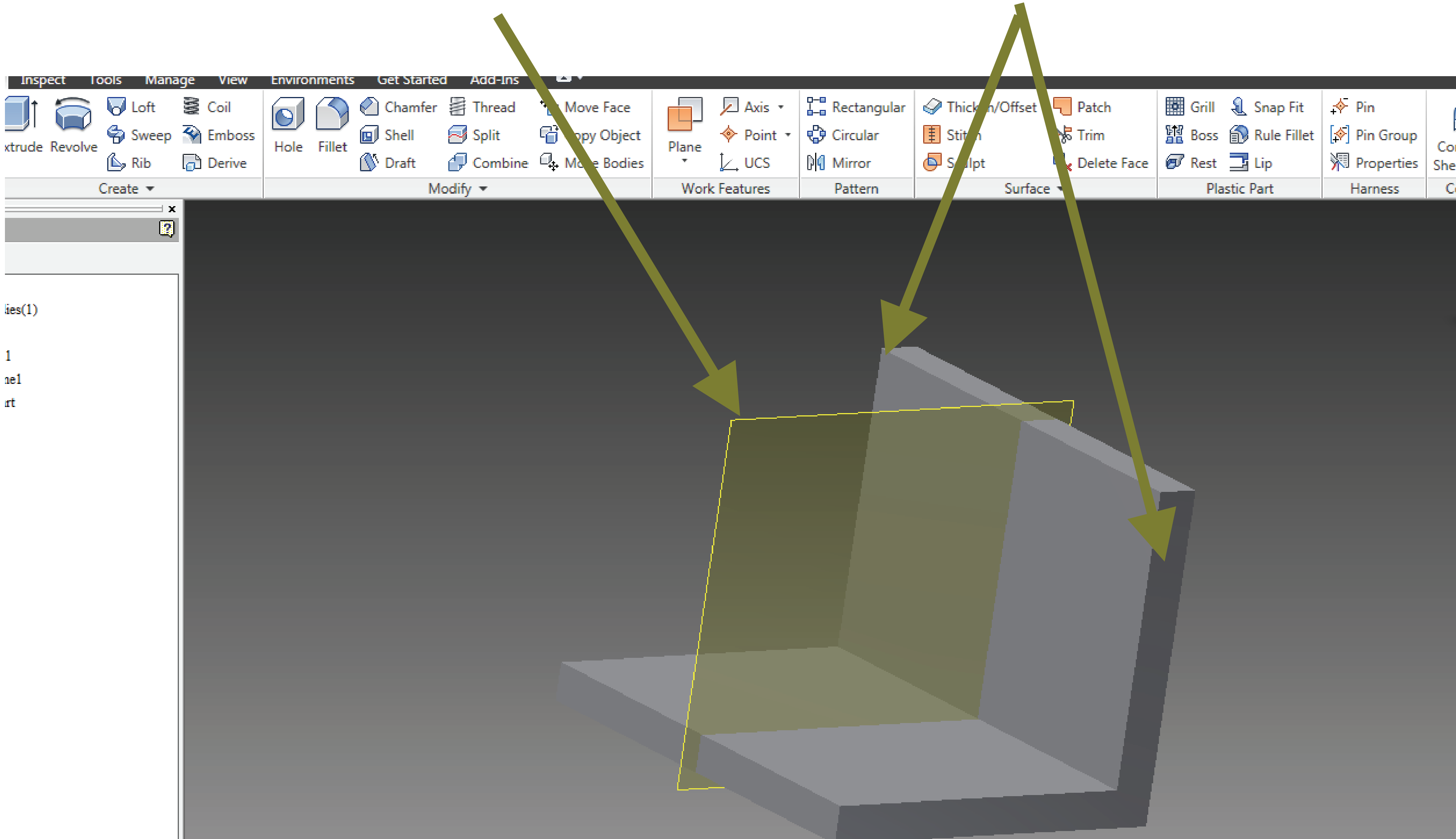
11. How to create a rib in a solid model ?

Create a new **working plane** by activating the **Plane** command from the Model Tab. Select the option **Mid plane between two parallel planes** in Plane sub menu



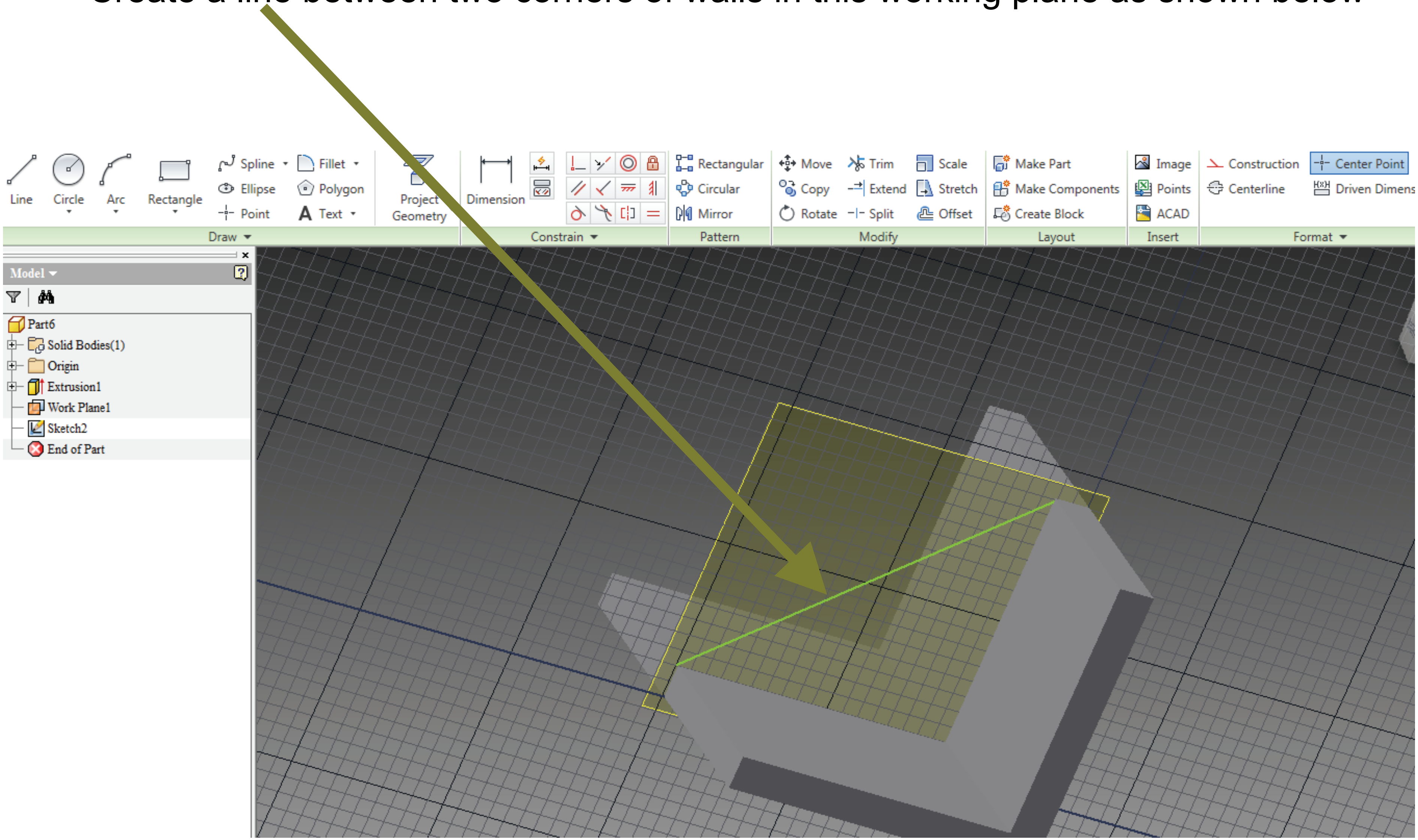
11. How to create a rib in a solid model ?

- This creates a new working plane (mid plane) between two parallel planes



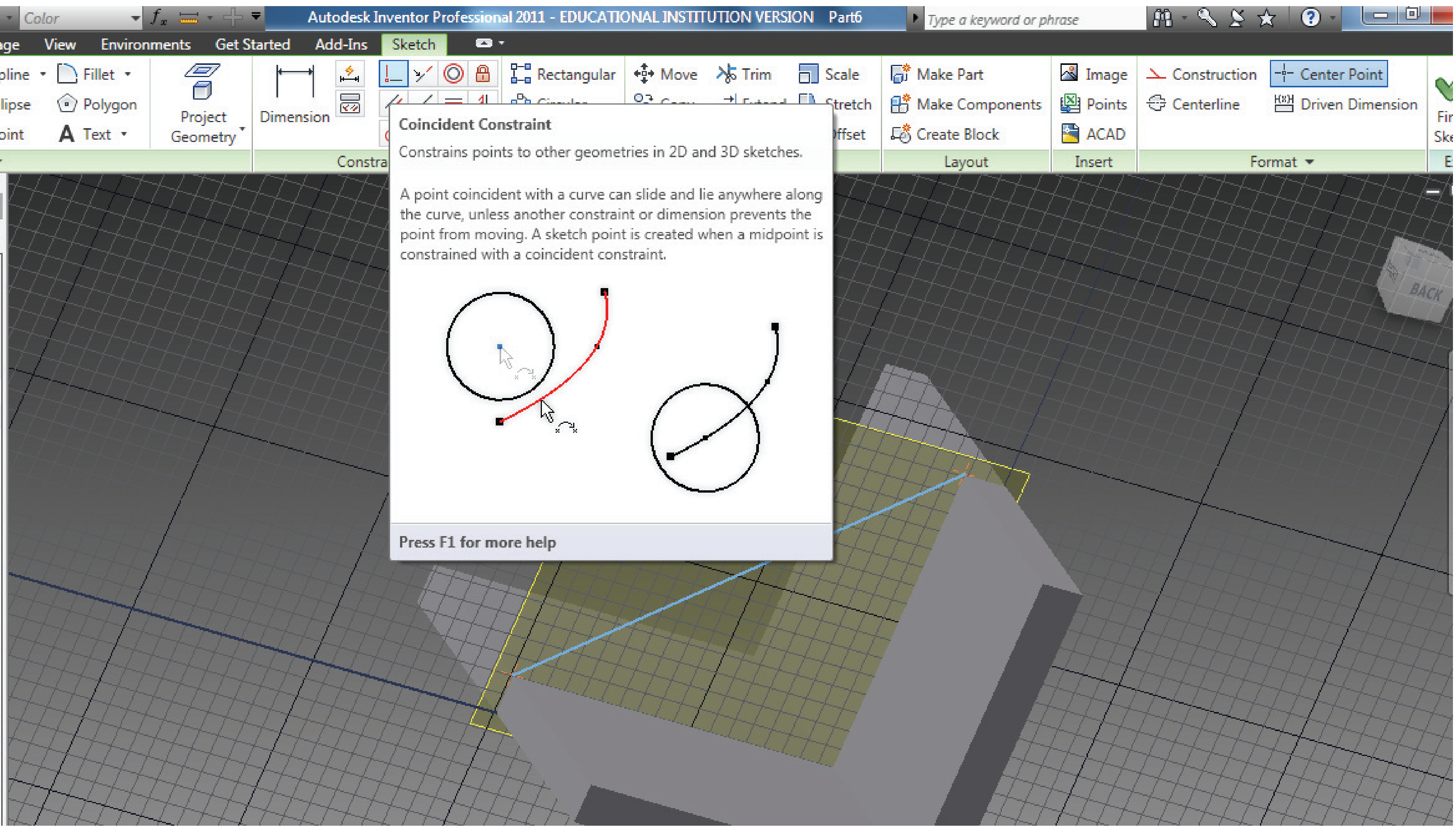
11. How to create a rib in a solid model ?

- Click at the corner of the plane and right click Create Sketch
- Create a line between two corners of walls in this working plane as shown below



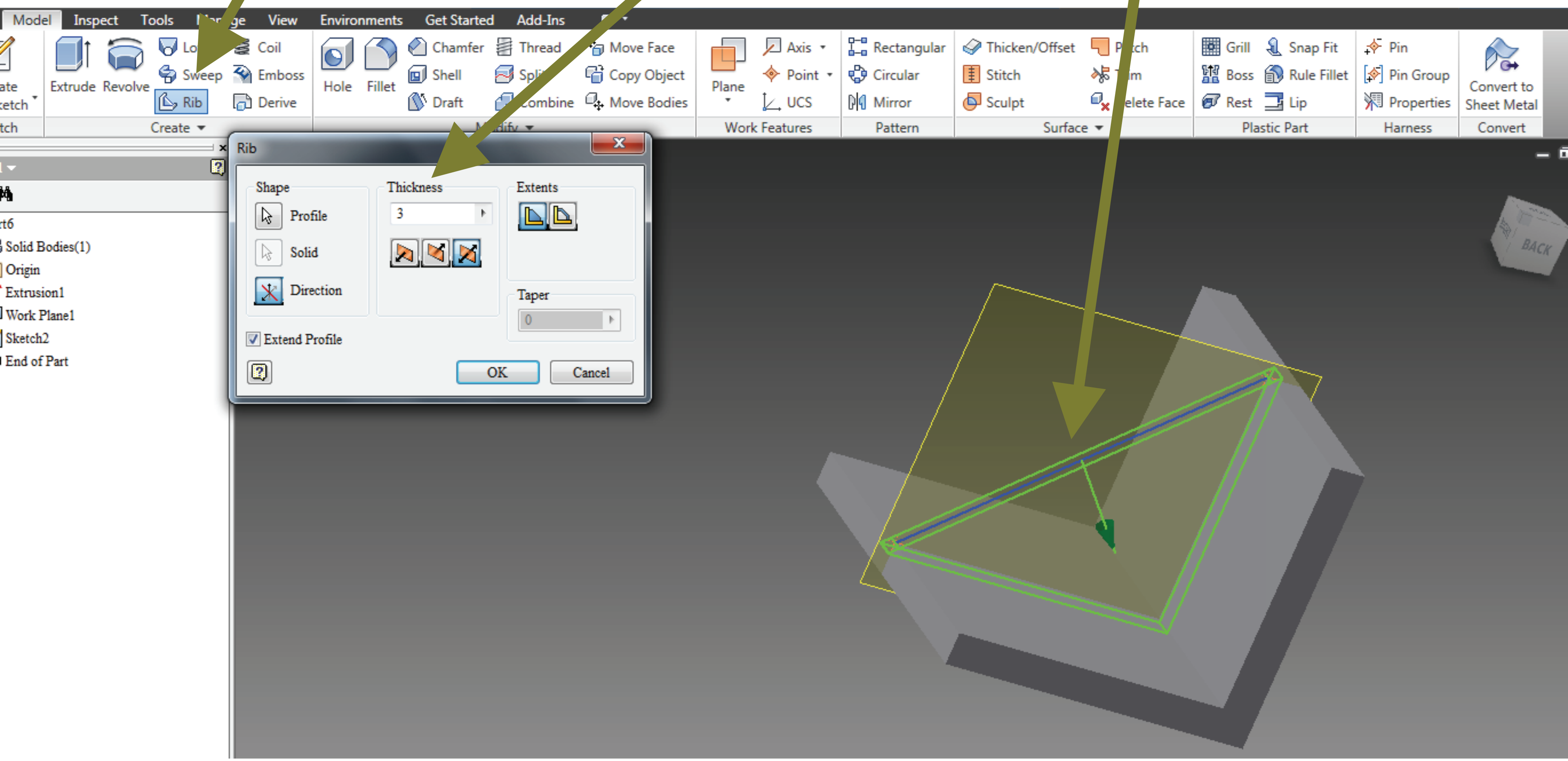
11. How to create a rib in a solid model ?

•Use **Coincident Constraint** for coinciding the edges of the walls with the end points of the lines. Click on **Finish Sketch** Command and exit the sketcher window.



11. How to create a rib in a solid model ?

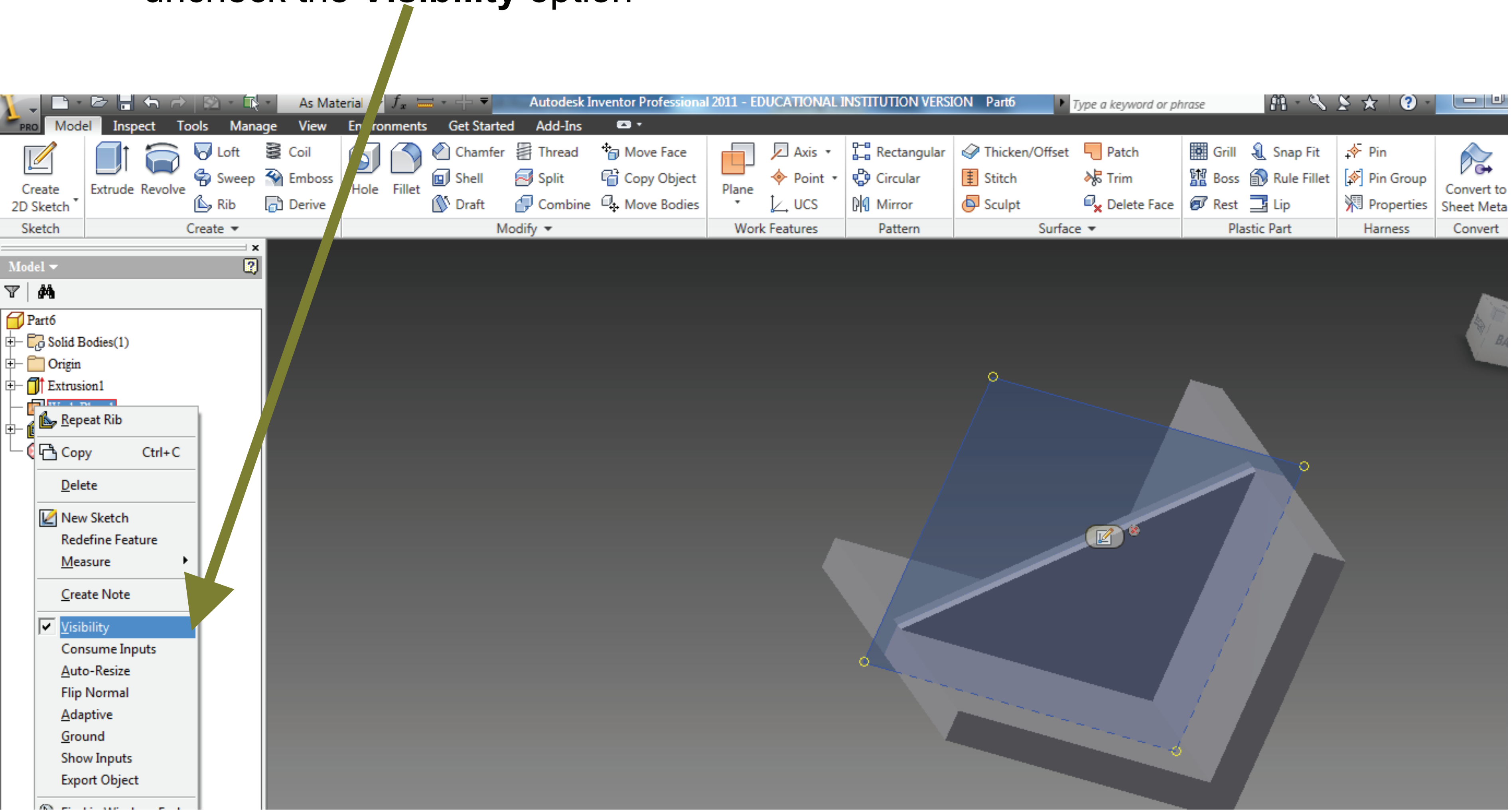
•Activate the **Rib** Command. Enter the rib thickness, select the direction by moving the cursor in the mid plane and **click OK**



11. How to create a rib in a solid model ?

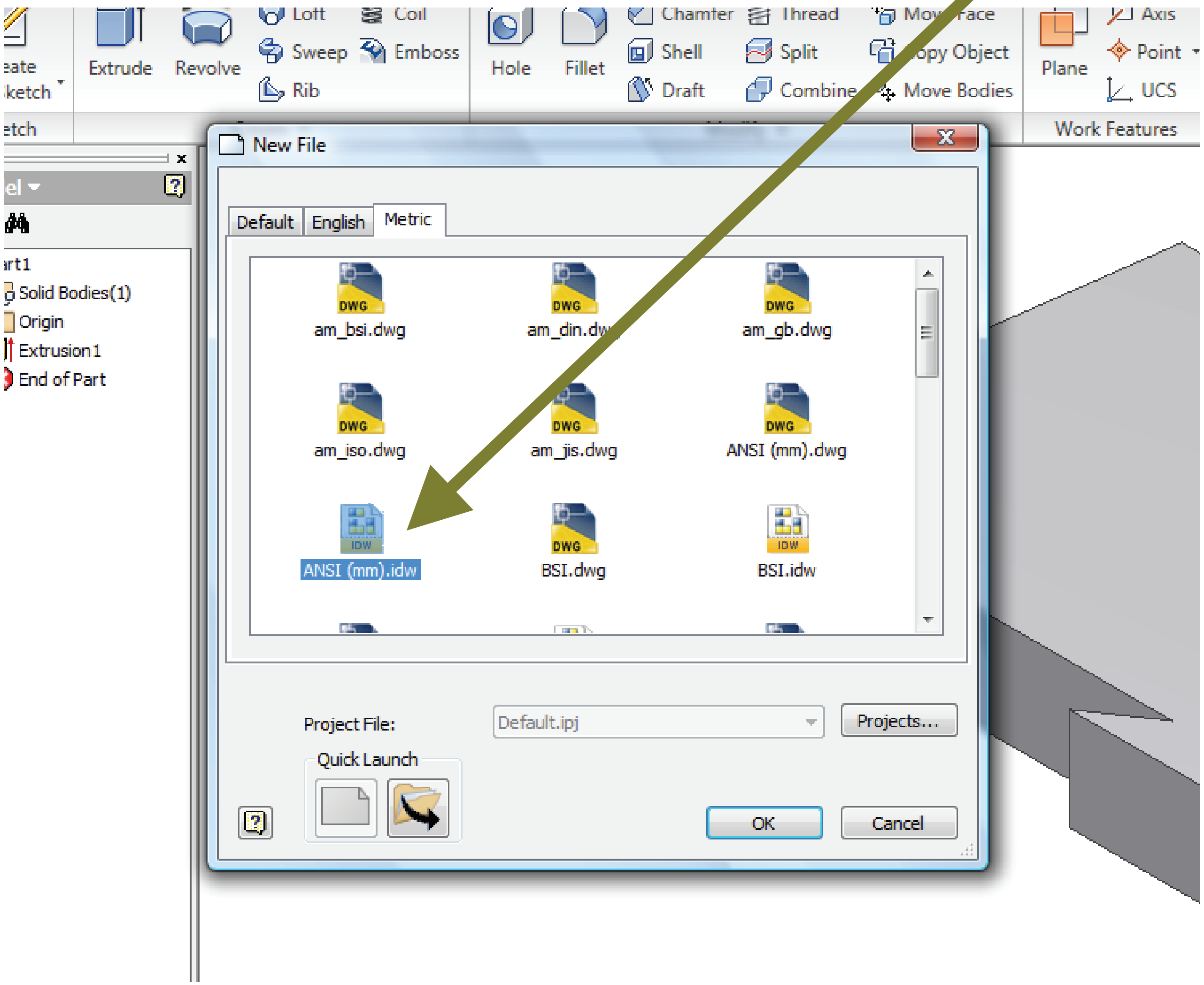
•This creates the required rib

▪To hide the mid plane, click the **Work plane** in the **browser window** and uncheck the **Visibility** option



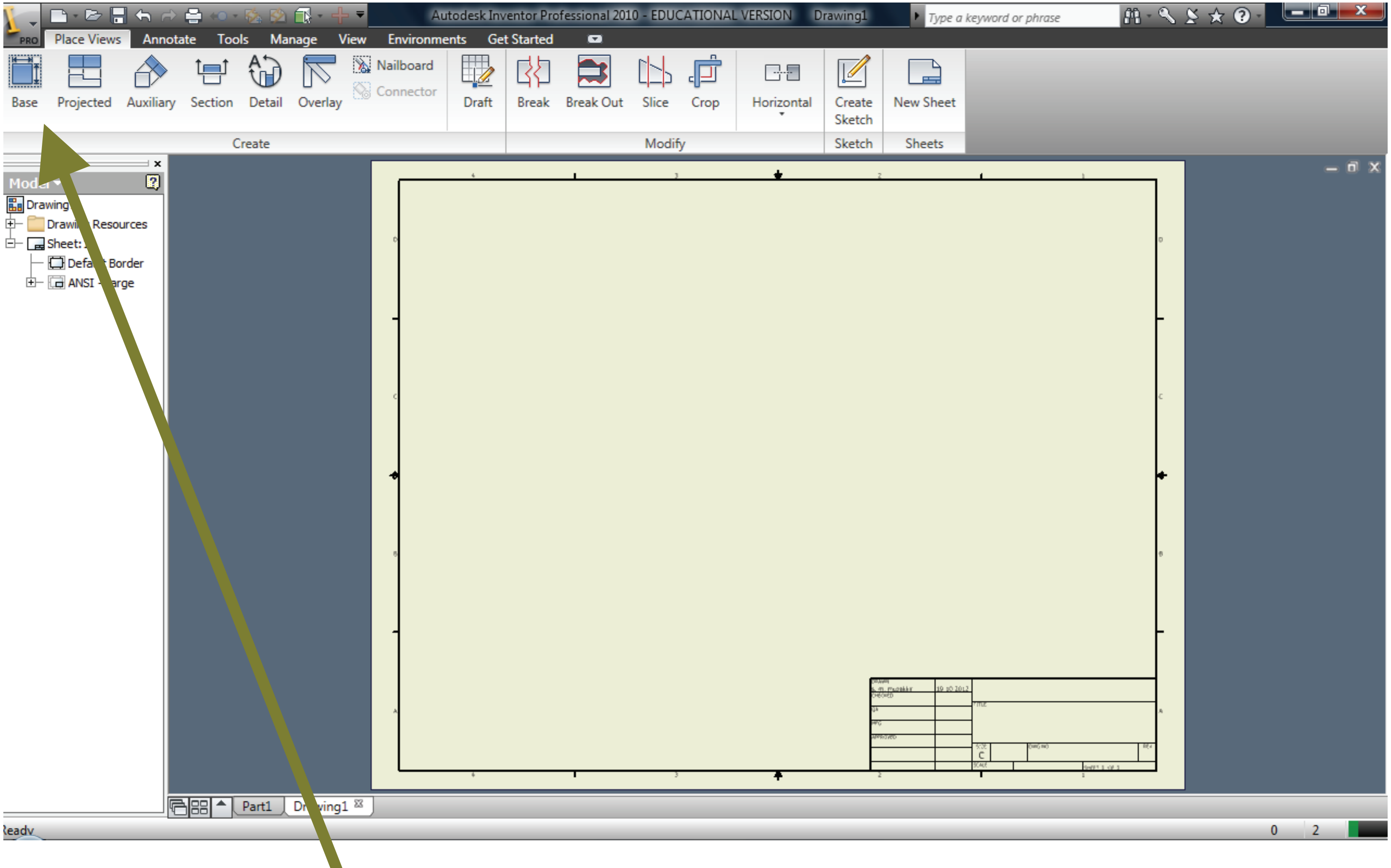
12. How to create orthographic projections of the solid model?

- In the '**Get Started**' tab, go to **new > Metric > ANSI(mm).idw** and click **OK**



12. How to create orthographic projections of the solid model?

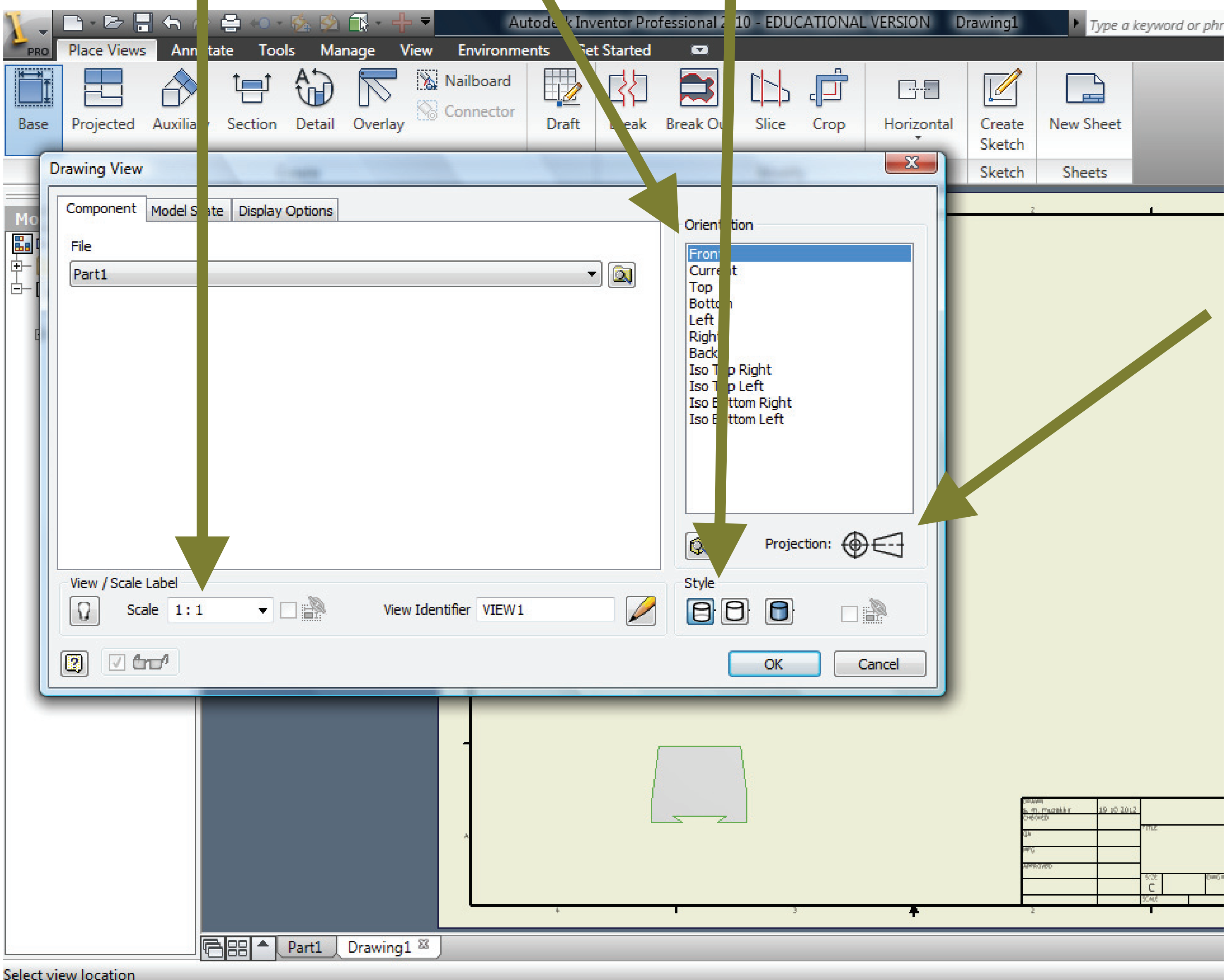
A new drawing sheet will open with a default name **Drawing1**



Now activate the **Base** command to insert the solid model whose orthographic projections is required.

12. How to create orthographic projections of the solid model?

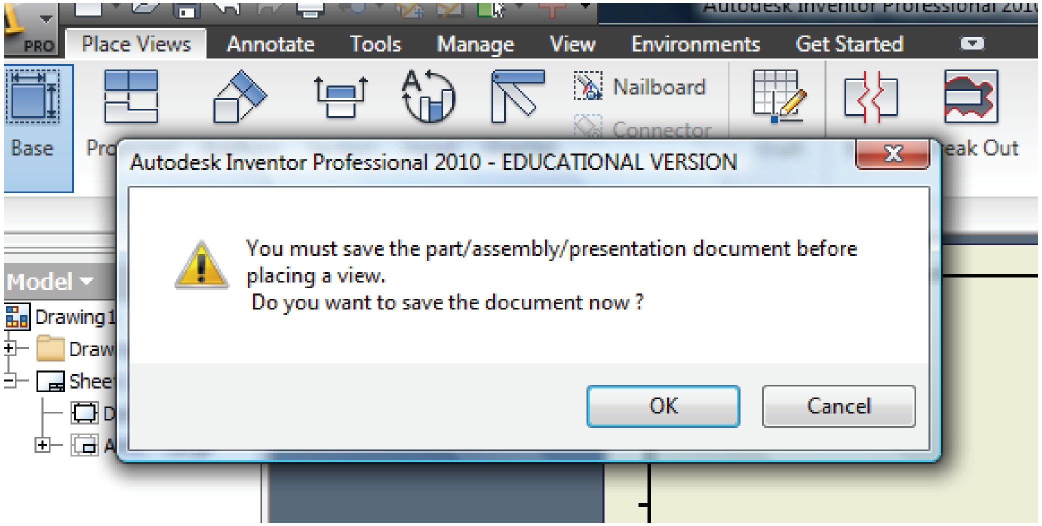
Select proper scaling factor, orientation and style ***(with or without hidden lines)*** of the drawing and click **OK**.



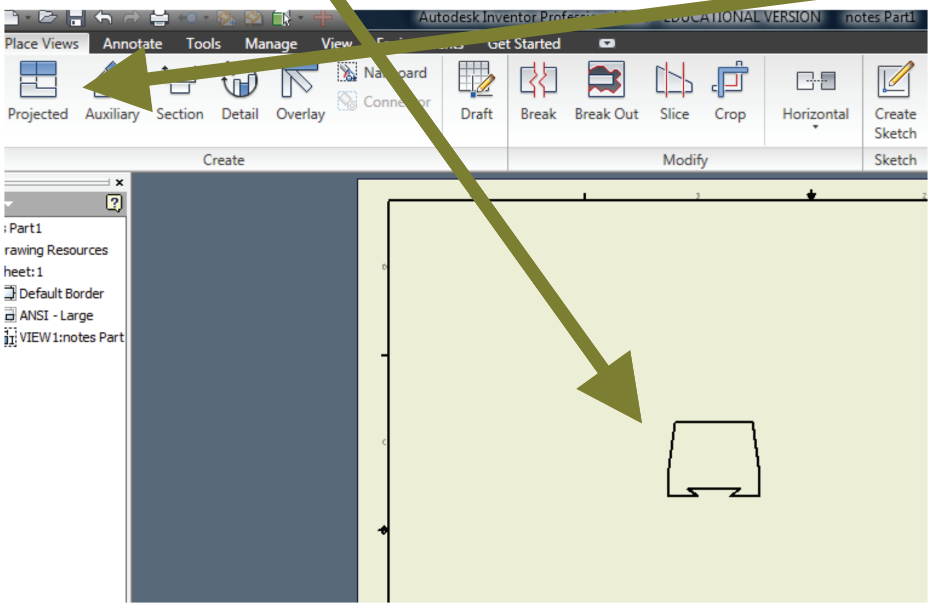
Note the III angle projection method, default for ANSI drawing

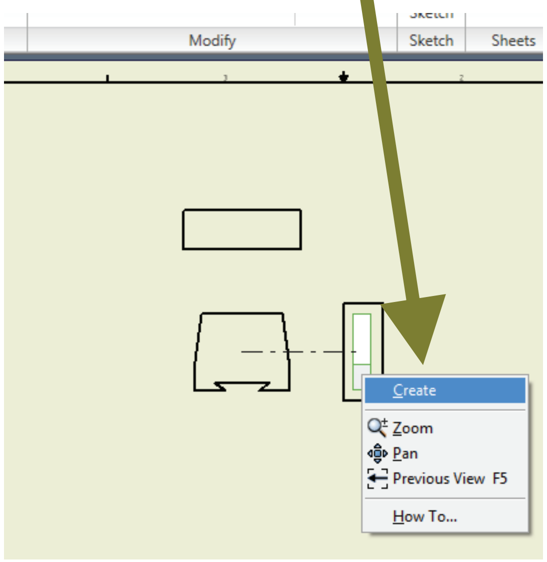
12. How to create orthographic projections of the solid model?

You will be prompt to save the part file if not done earlier. Save the part file.



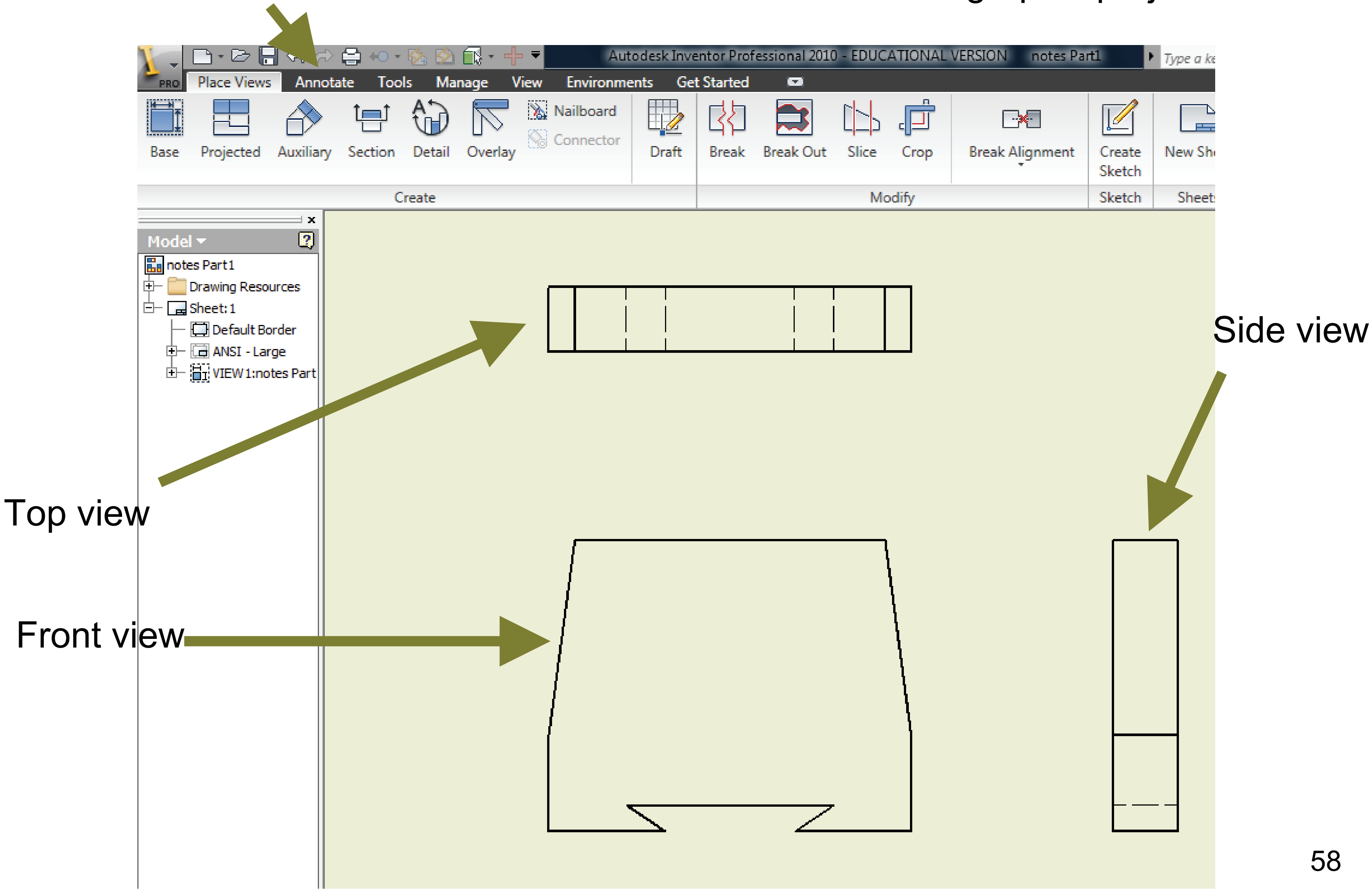
The front view of the solid model will be created. Now use **Projected** command to create the necessary views (top view, side view etc) and click **create**





12. How to create orthographic projections of the solid model?

The front view, top view and side view of the solid model will be created. Now use **Annotate** command to add dimensions to the orthographic projections

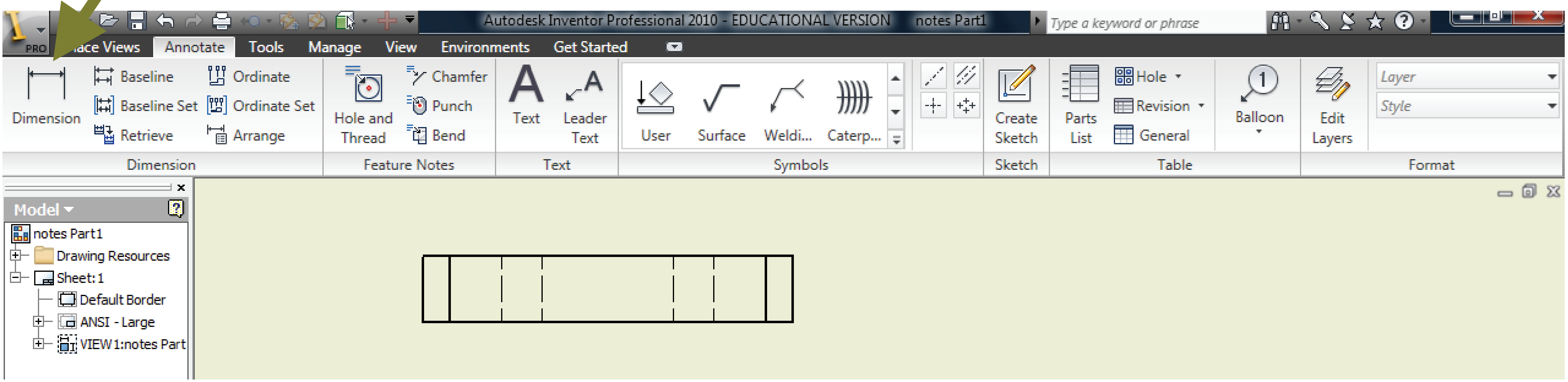


12. How to create orthographic projections of the solid model?

Click on **Dimension** command to add dimensions to the orthographic projections.

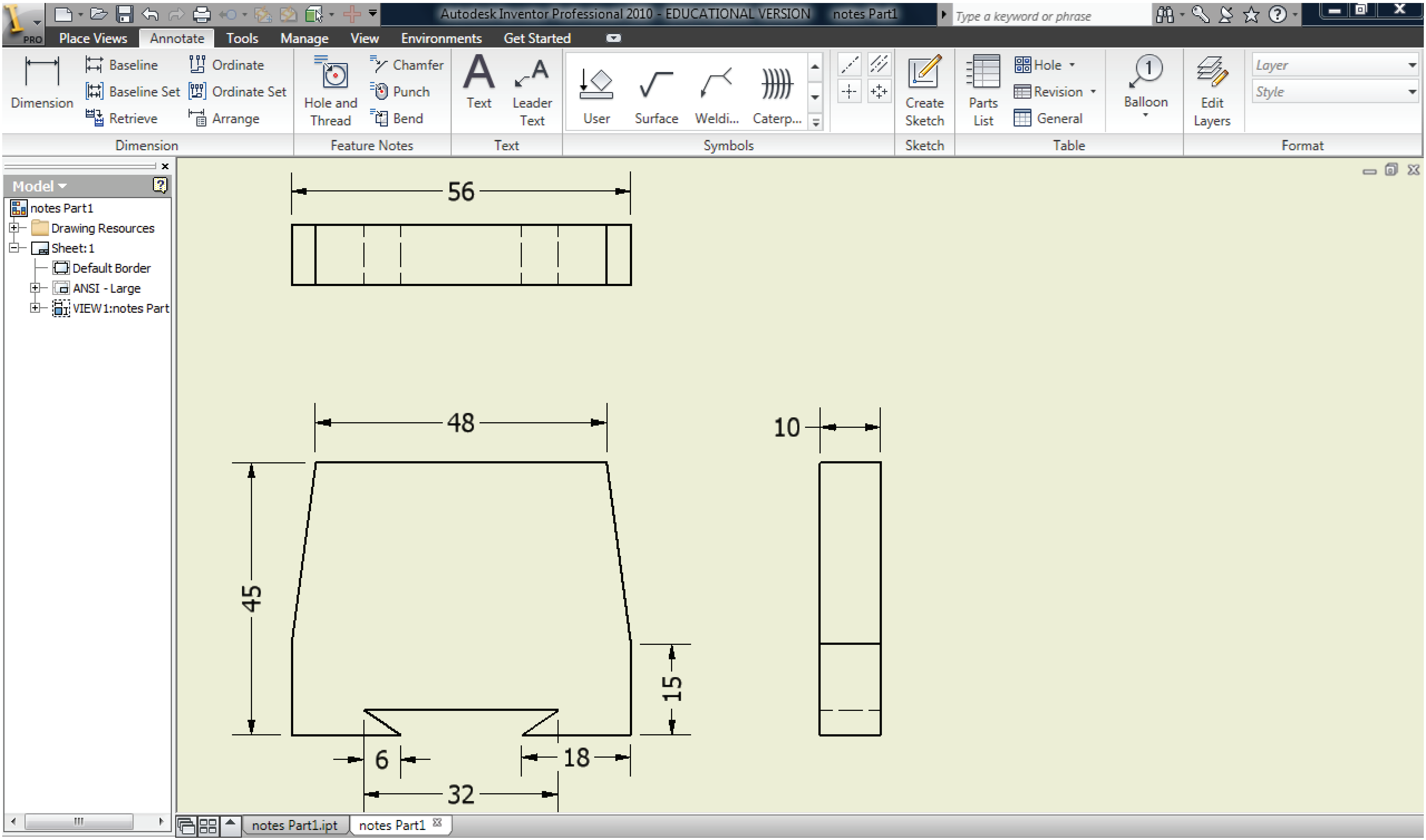
Add dimension to the orthographic projections by clicking the line/feature (that need to be dimensioned) and then dragging the mouse to a suitable distance to place the dimension line

Click right mouse button and select ‘**Done**’



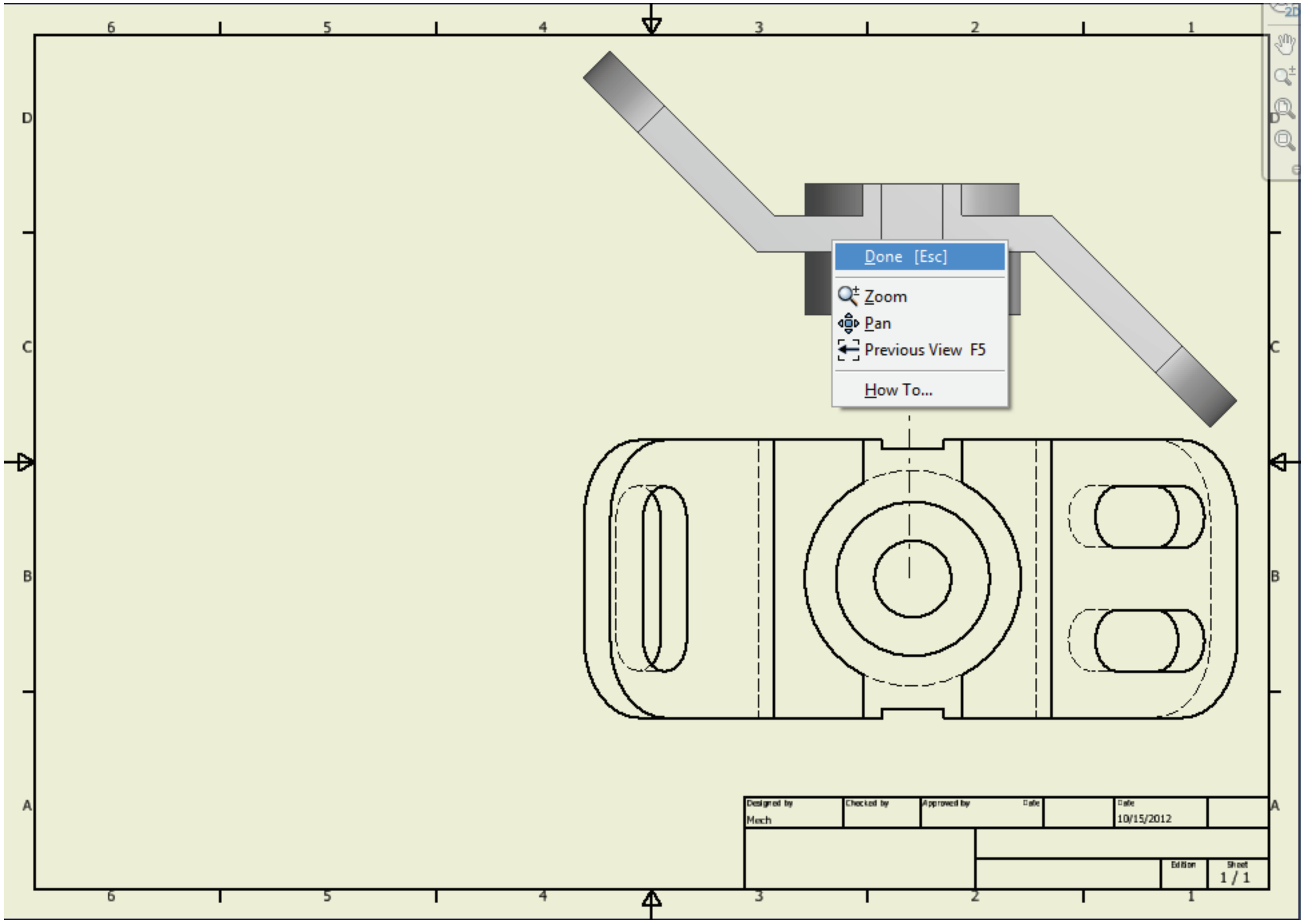
12. How to create orthographic projections of the solid model?

The complete orthographic projections of the solid model are thus created



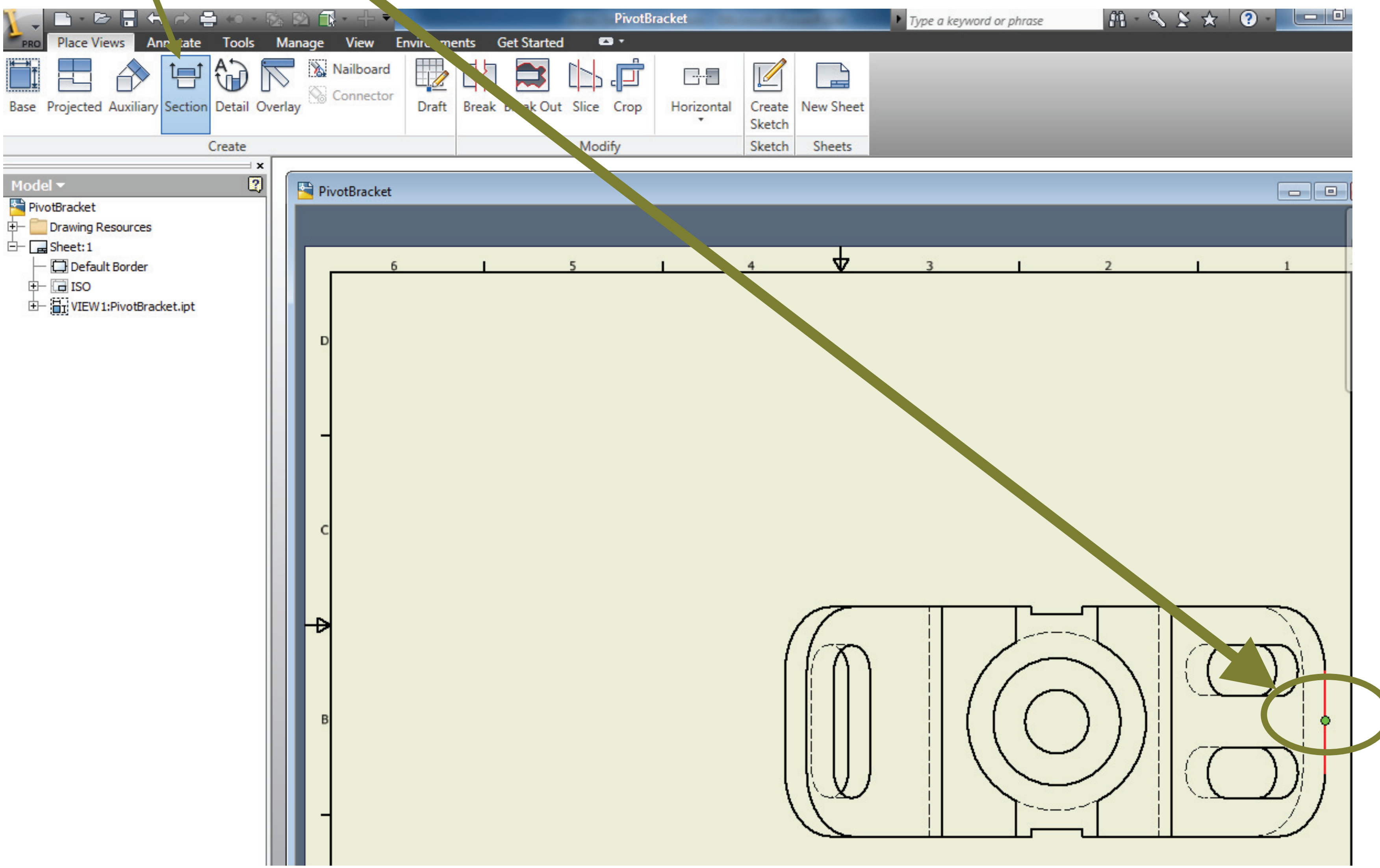
13. How to create Sectional Views of the solid model?

- Open the drawing file



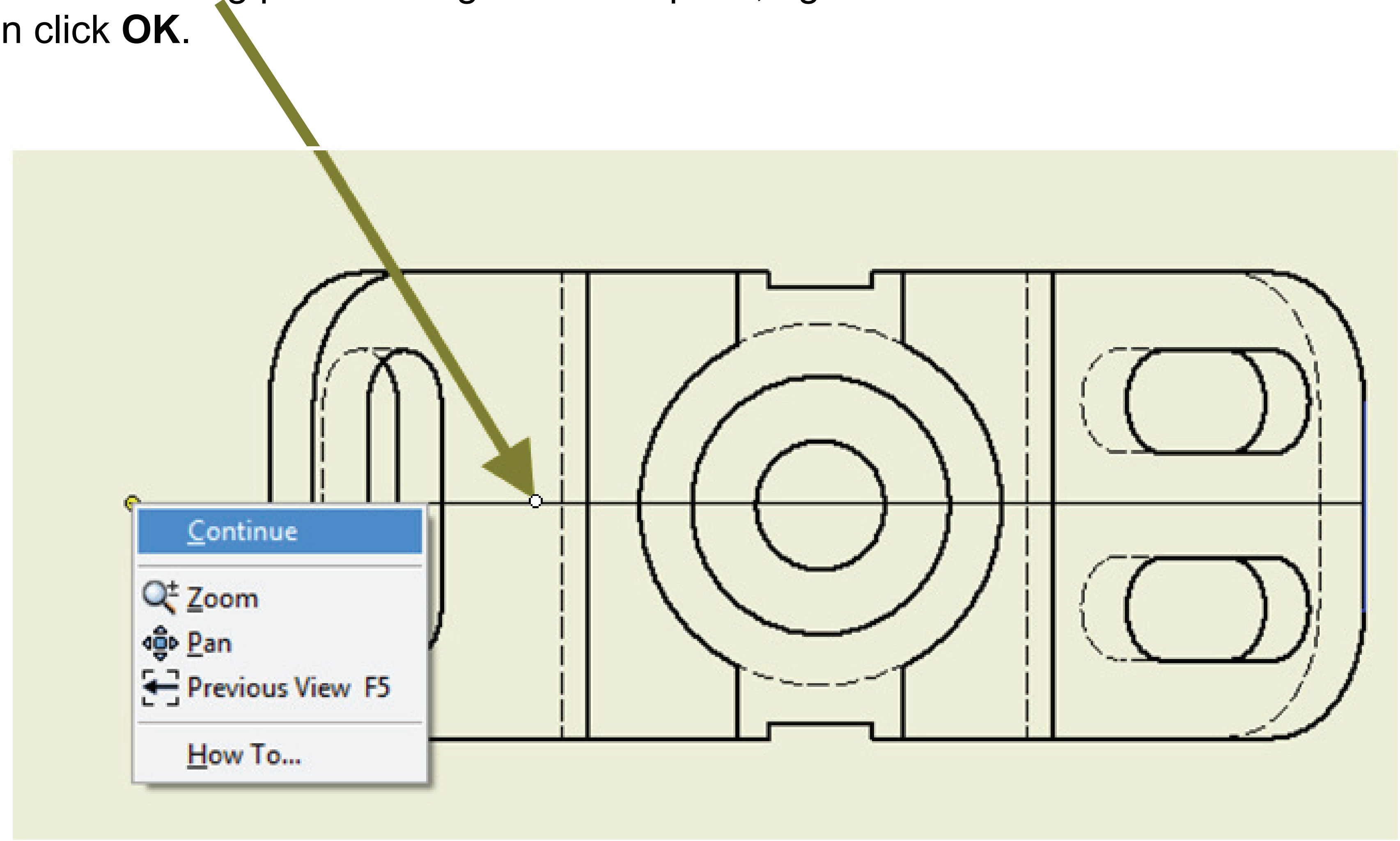
13. How to create Sectional Views of the solid model?

▪Click **'Section'** and draw a cutting plane line passing through the centre of the top view by selecting the mid point of the right-most vertical edge



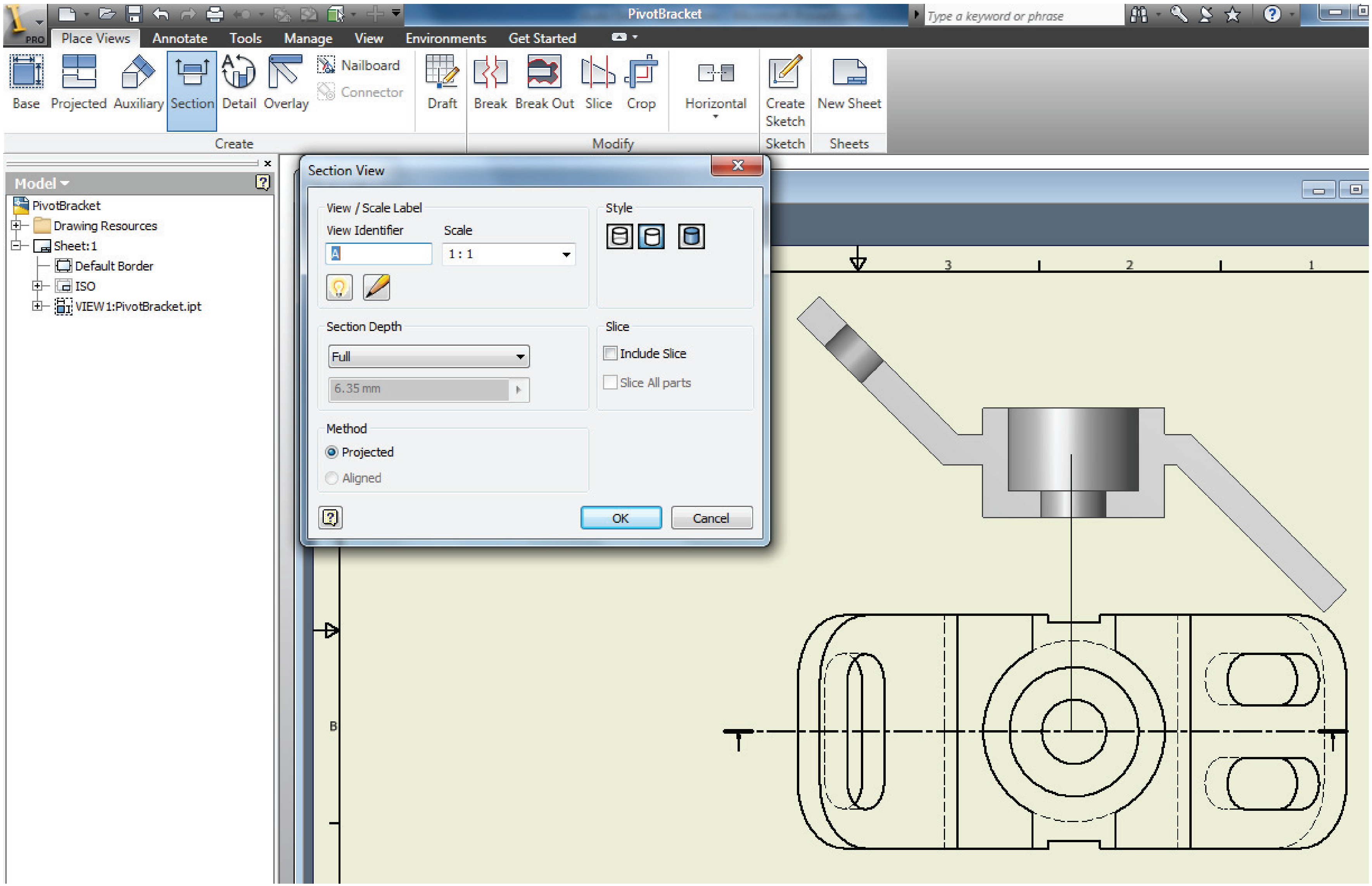
13. How to create Sectional Views of the solid model?

▪Draw the cutting plane through selected point, right click and click **continue** and then click **OK**.



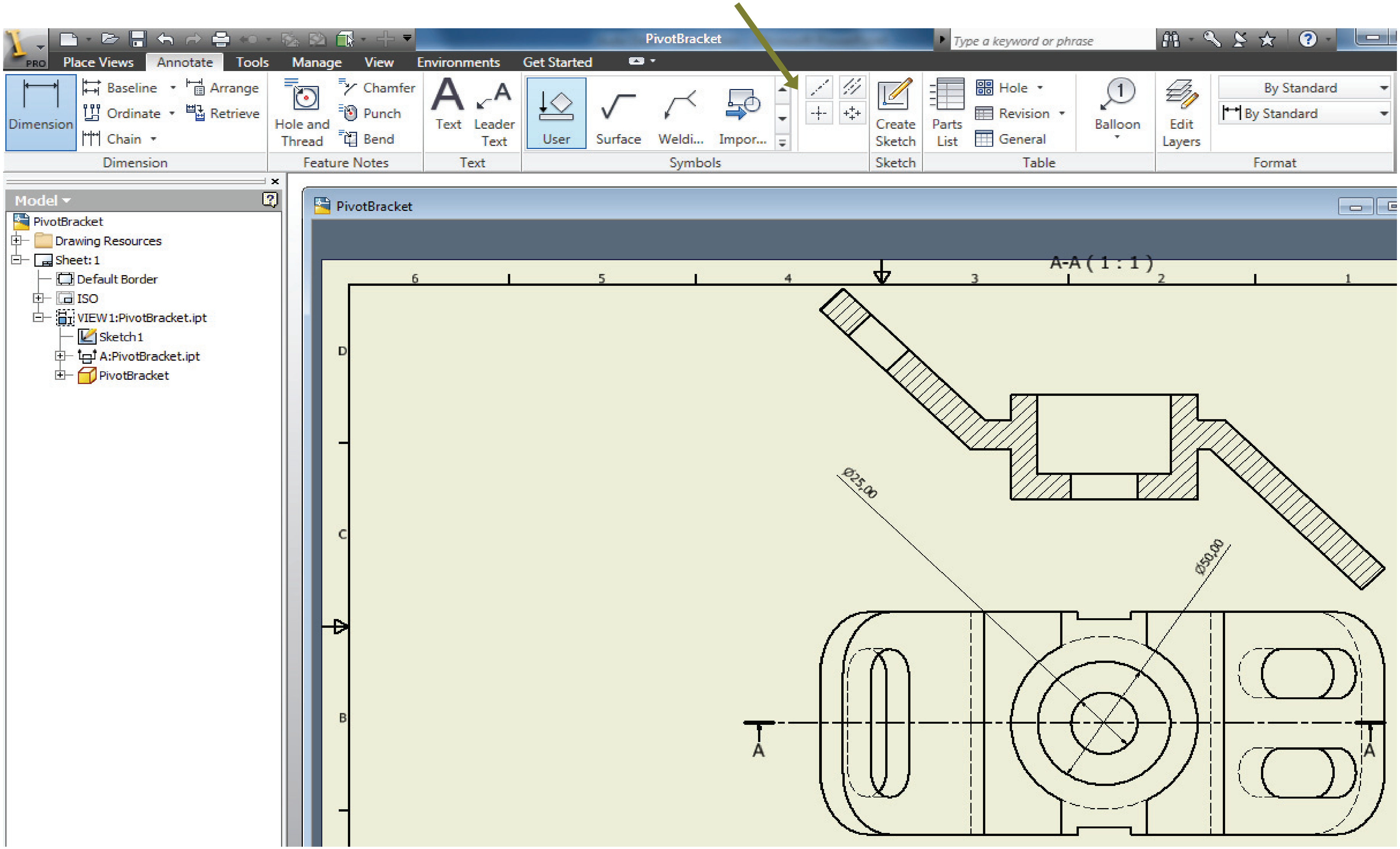
13. How to create Sectional Views of the solid model?

This window will then open, click **OK** and the sectional view will be created



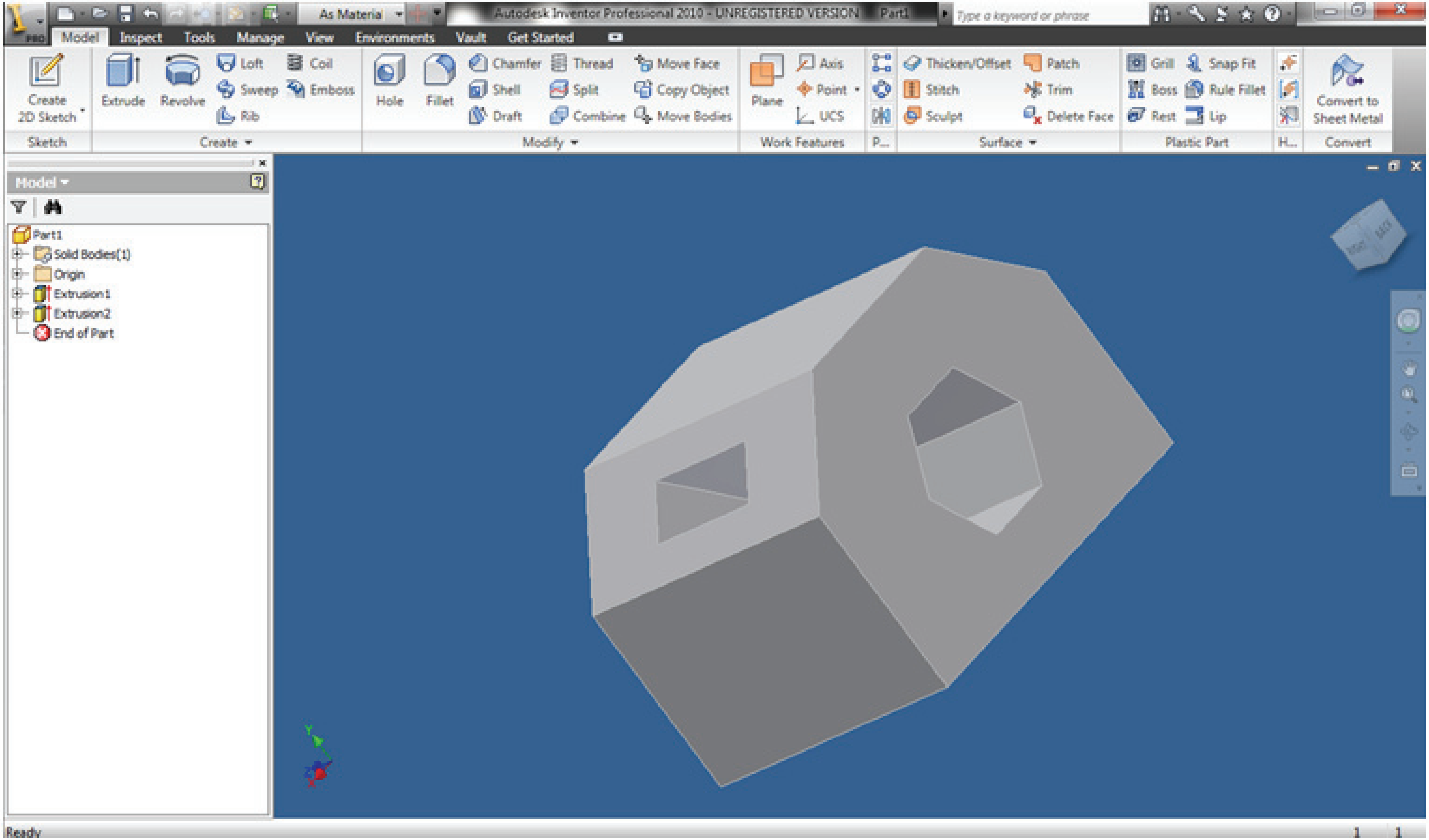
13. How to create Sectional Views of the solid model?

- After the sectional view is created, add center lines, axes etc to the view



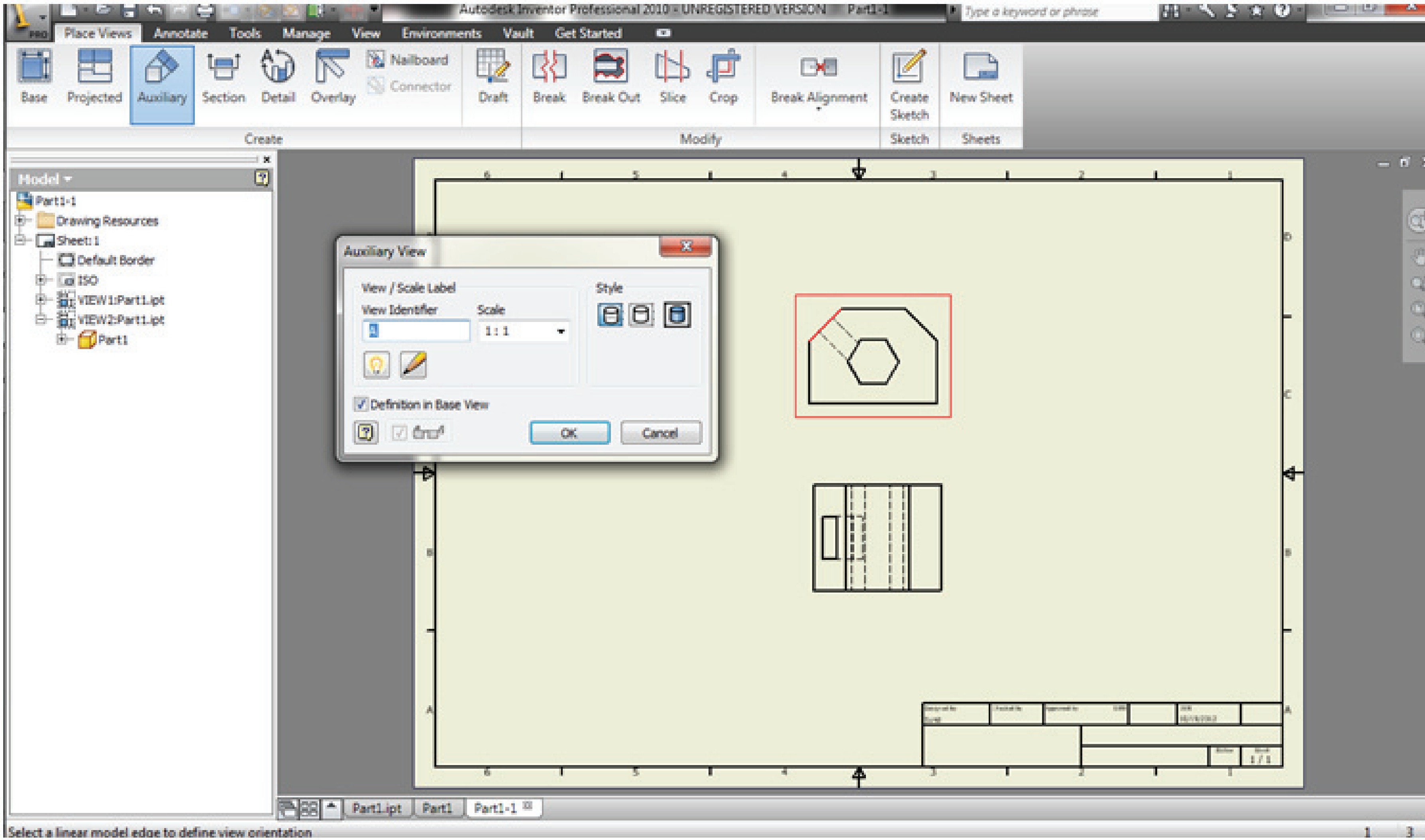
14. How to create auxiliary views of the solid model?

Create a solid model as shown below and create its orthographic projections



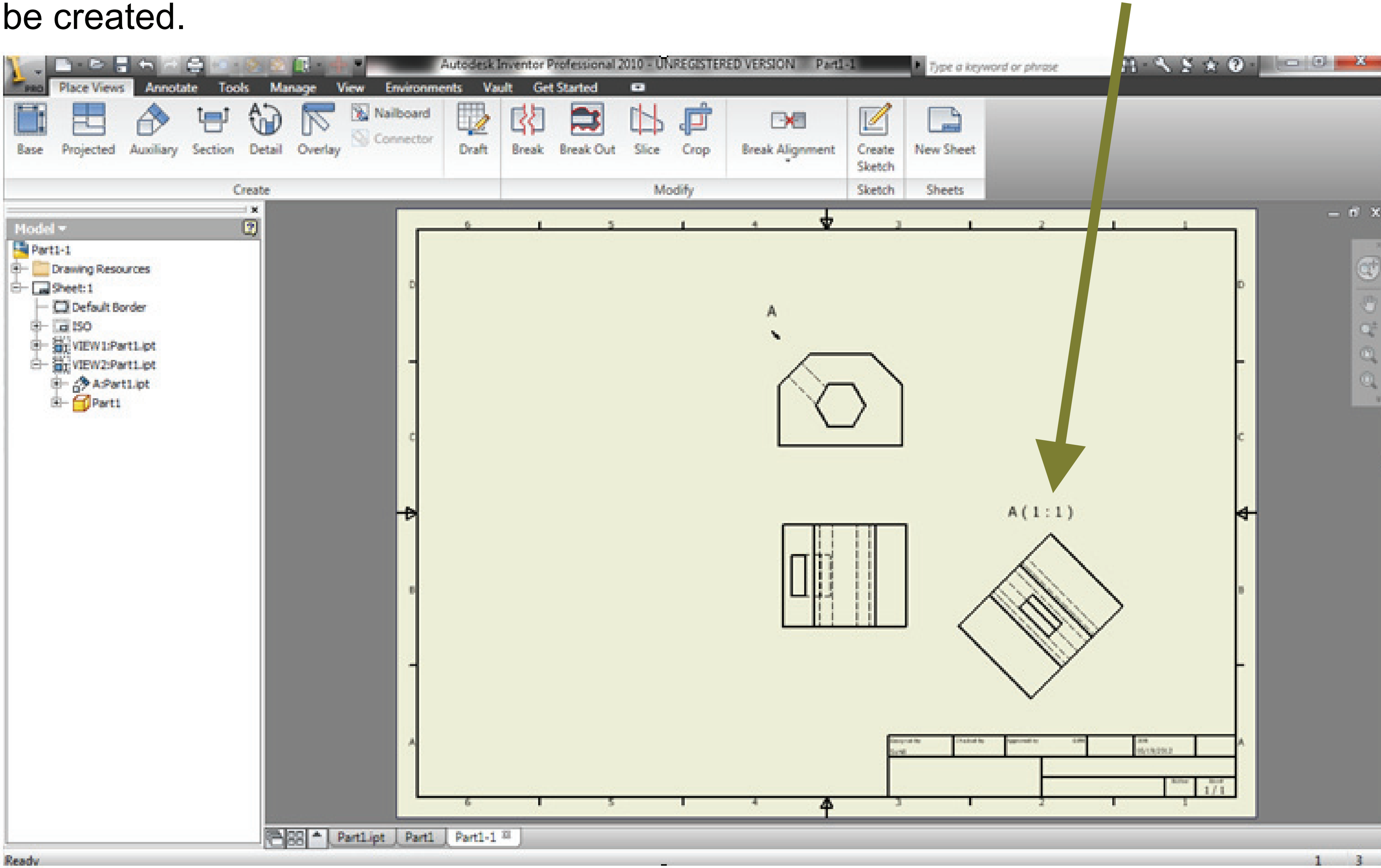
14. How to create auxiliary views of the solid model?

▪Now in **‘Place Views’** tab, click **‘Auxiliary’** and select the surface, perpendicular to which the auxiliary view is to be drawn.

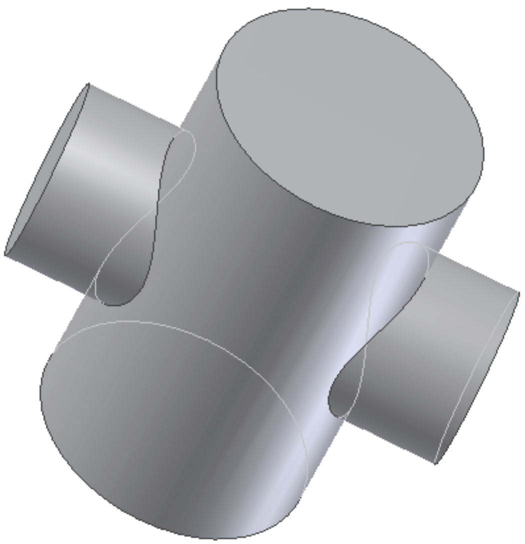


14. How to create auxiliary views of the solid model?

- Click the side on which the auxiliary view is to be drawn and the auxiliary view will be created.



15. How to create curves of intersection of two solids?



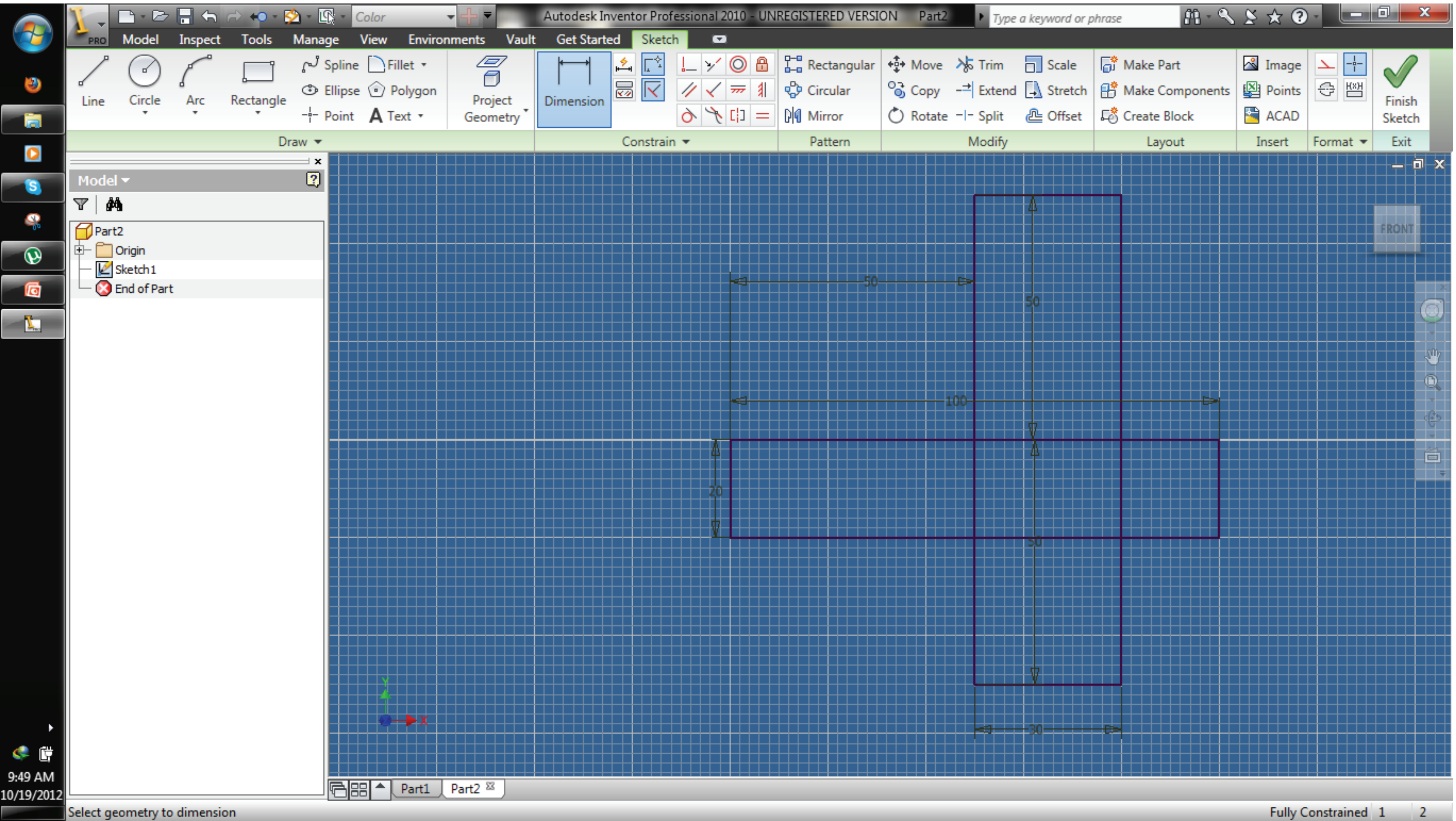
- A vertical cylinder of 60 mm diameter and 100 mm length is completely penetrated by a horizontal cylinder (40 mm diameter and 100 mm length)
- Axes of both cylinders bisect each other
- Create the orthographic projections of intersecting solids and obtain the curves of intersection

Procedure:

Create the solid models of two cylinders using the "Revolve" command

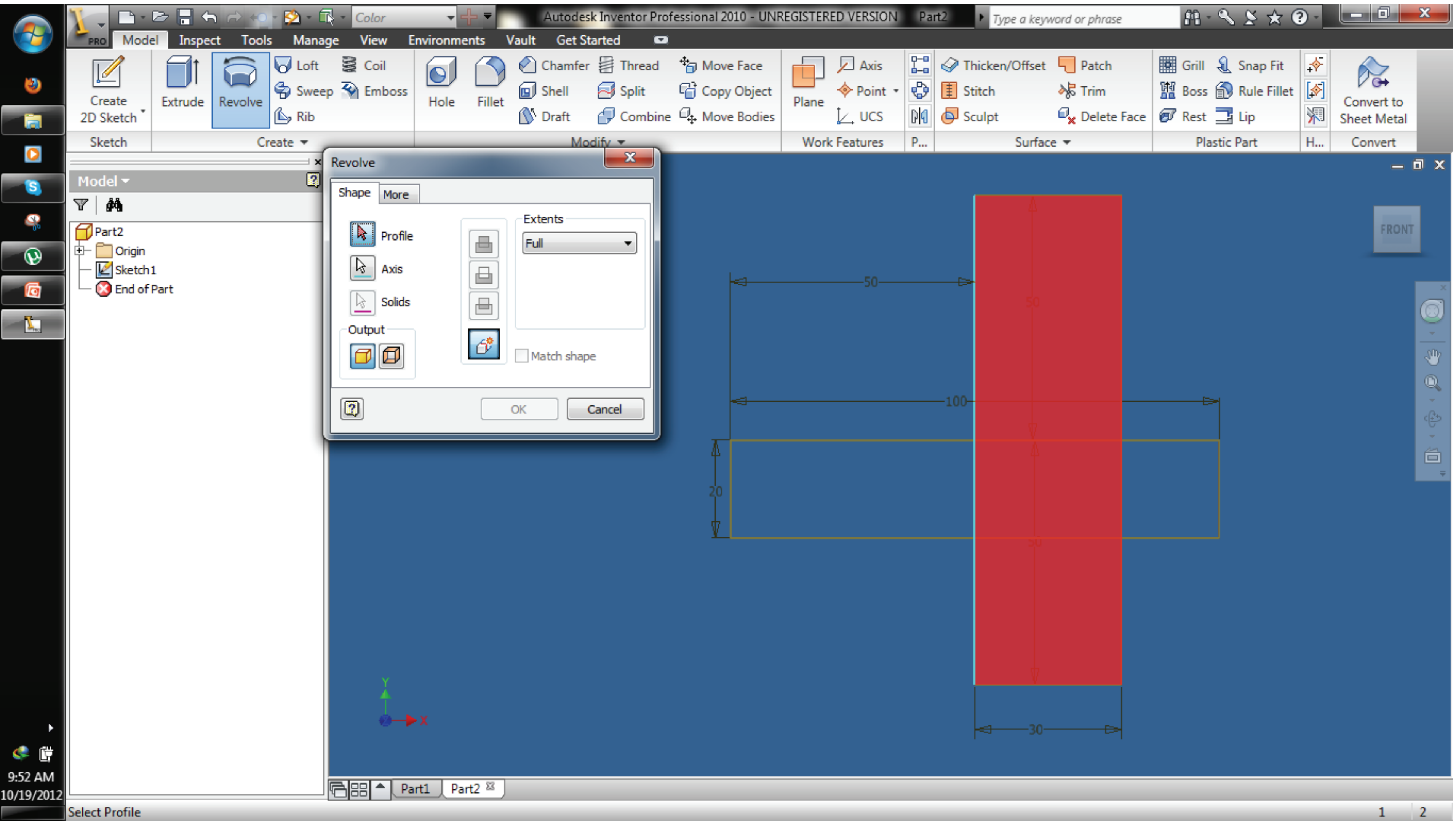
15. How to create curves of intersection of two solids?

In the Sketch mode, draw the two rectangles in such a way that, after revolution of these rectangles, the required cylinders are obtained as per the given dimensions



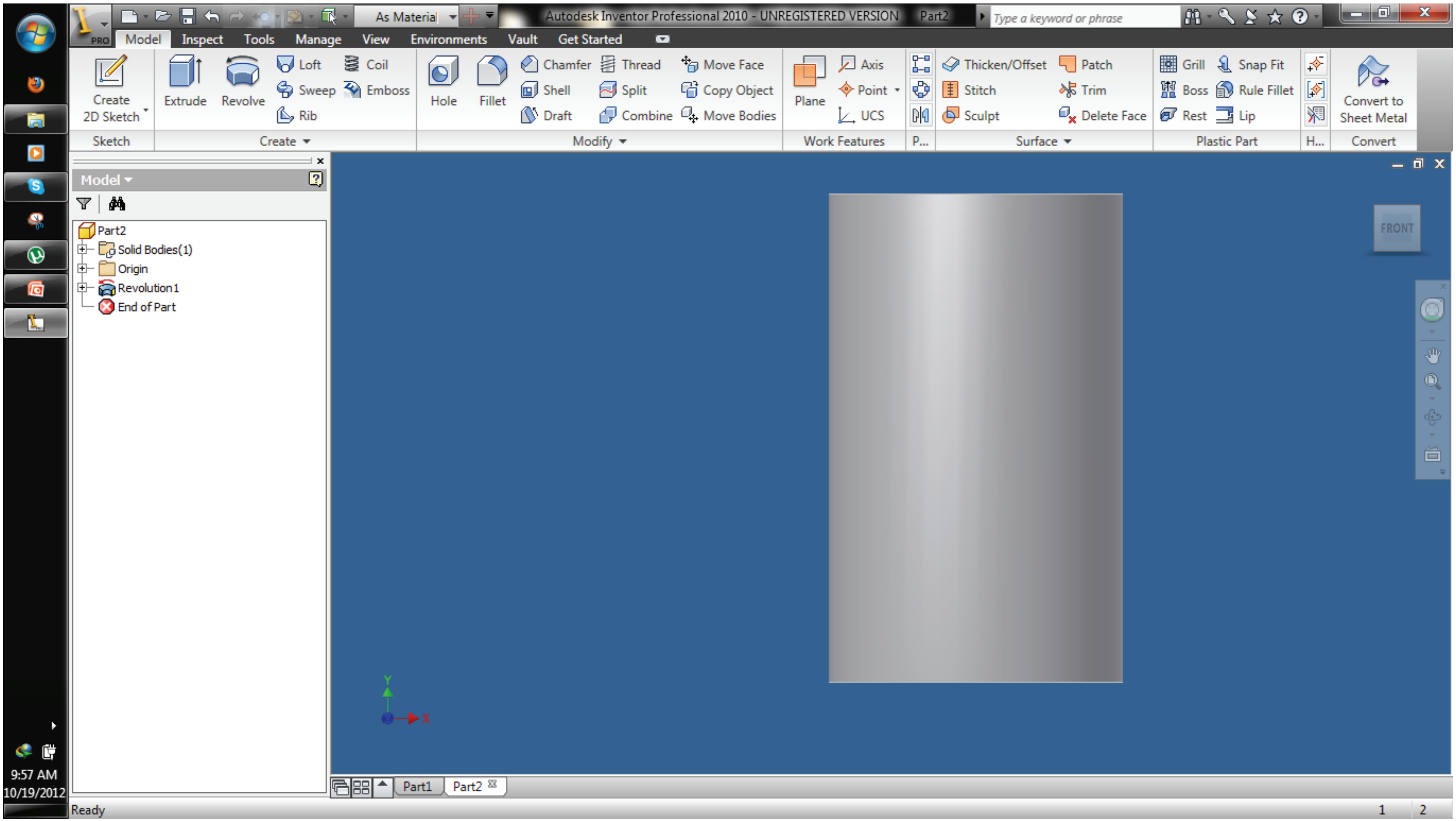
15. How to create curves of intersection of two solids?

Create the vertical cylinder by selecting the 'profile' and by selecting the left side vertical edge as an 'axis' in 'Revolution' tab and click 'OK' to complete the solid.



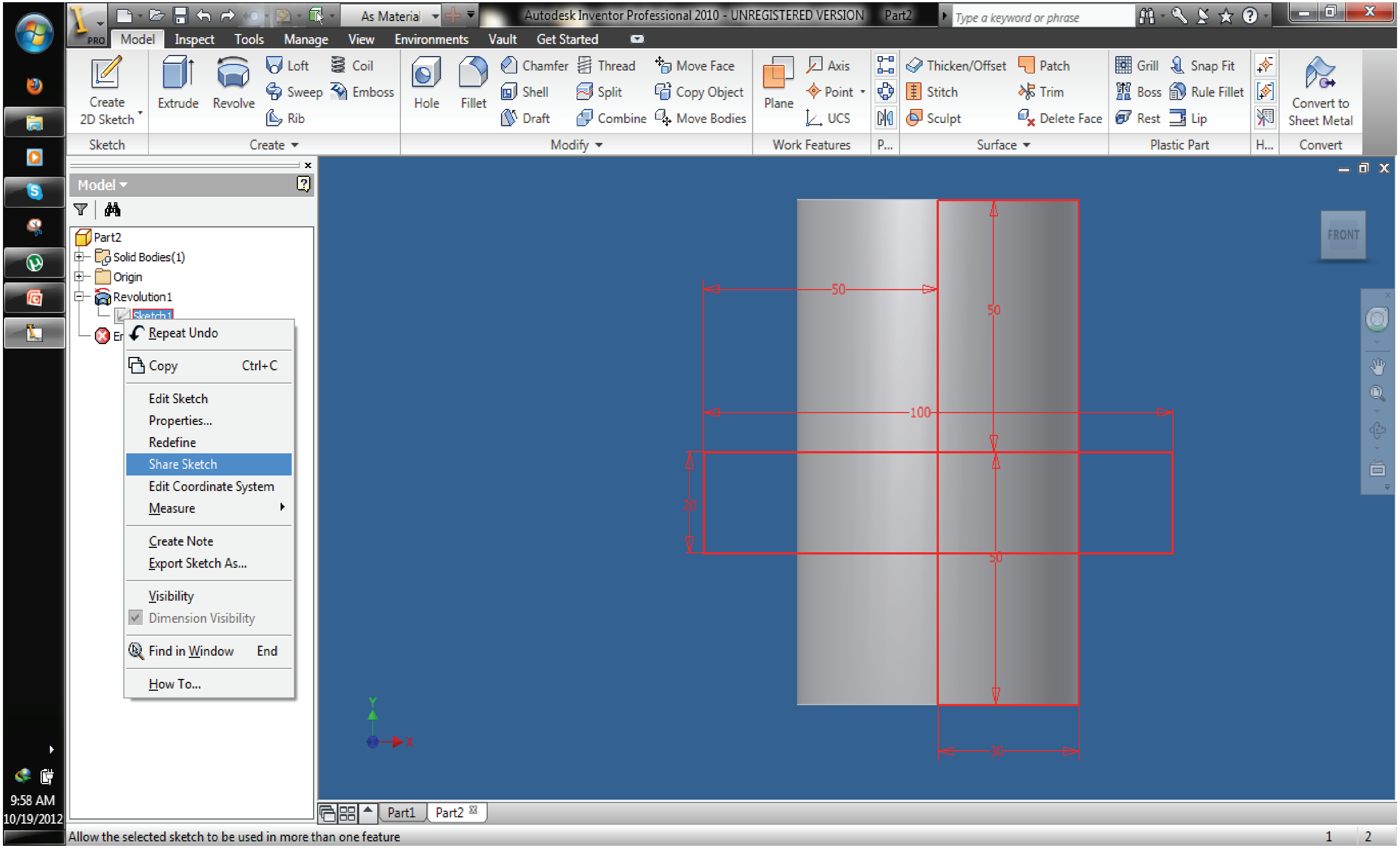
15. How to create curves of intersection of two solids?

One cylinder is thus created



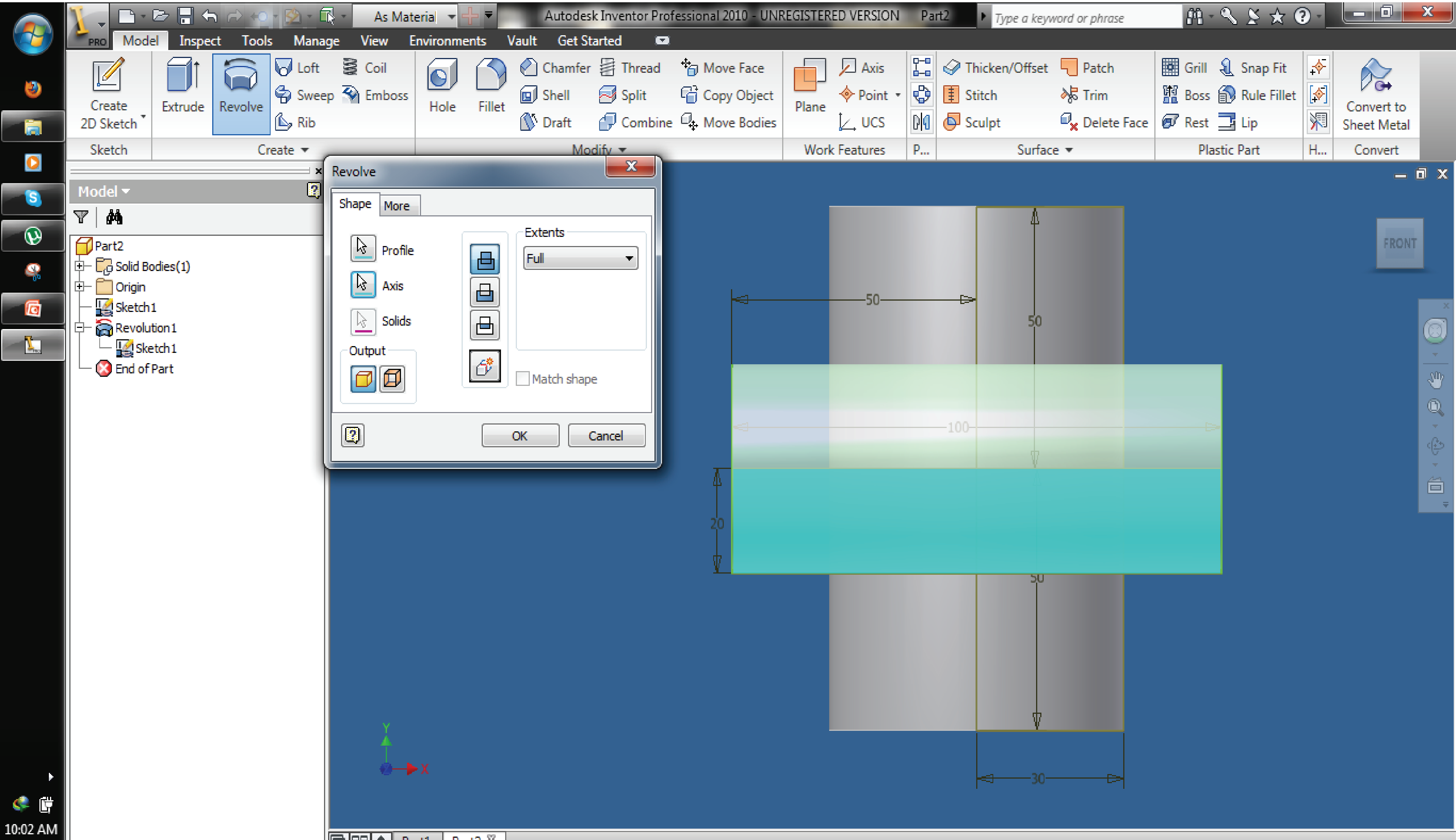
15. How to create curves of intersection of two solids?

For drawing the other cylinder, click revolution1 (in the browser window) then right click Sketch1/ Share Sketch



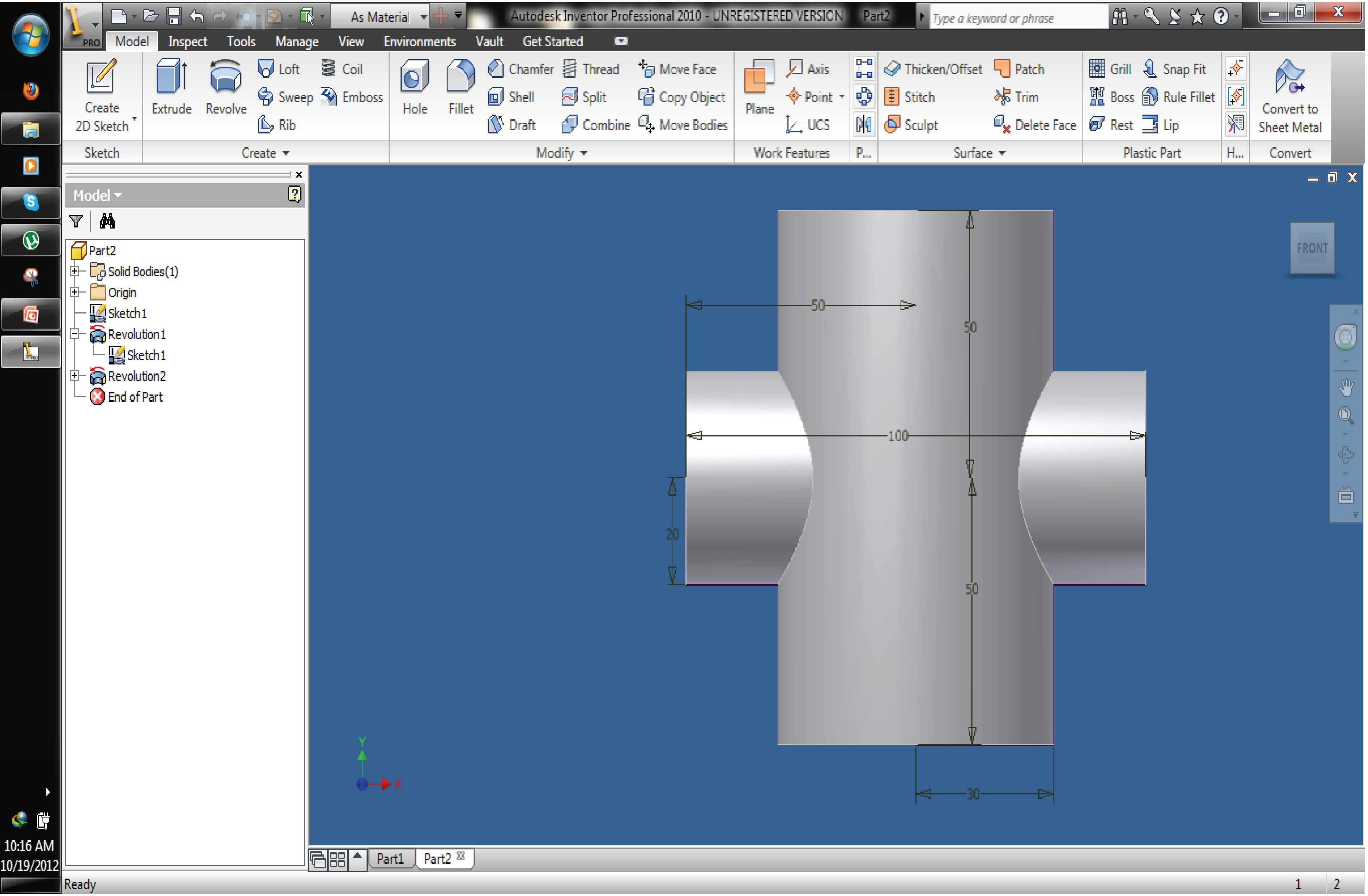
15. How to create curves of intersection of two solids?

Create the horizontal cylinder by selecting this ‘profile’ and by selecting the upper horizontal edge of the rectangle as an ‘axis’ in ‘Revolution’ tab and click OK to complete the solid.



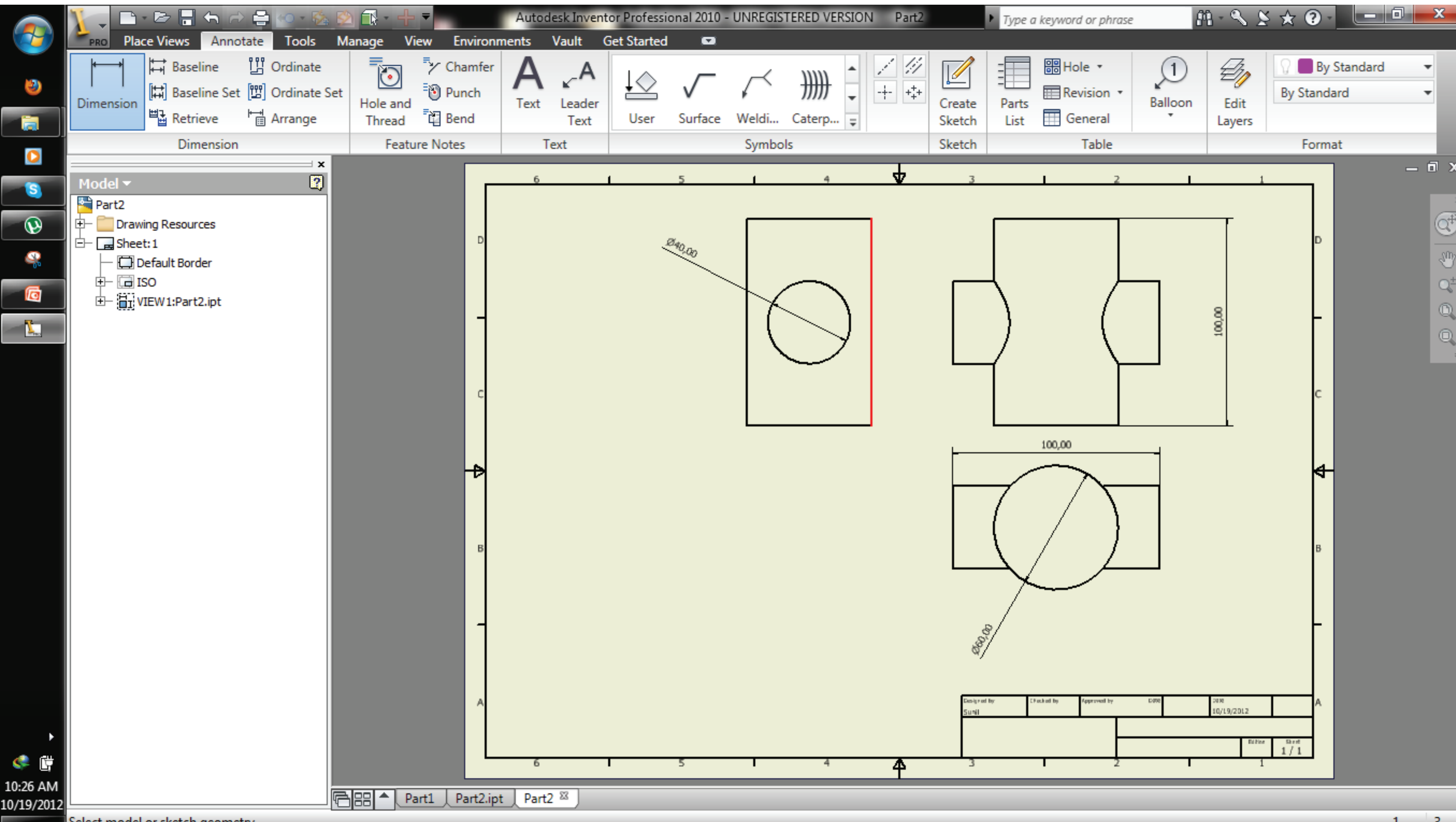
15. How to create curves of intersection of two solids?

The two cylinders are thus created, intersecting with each other



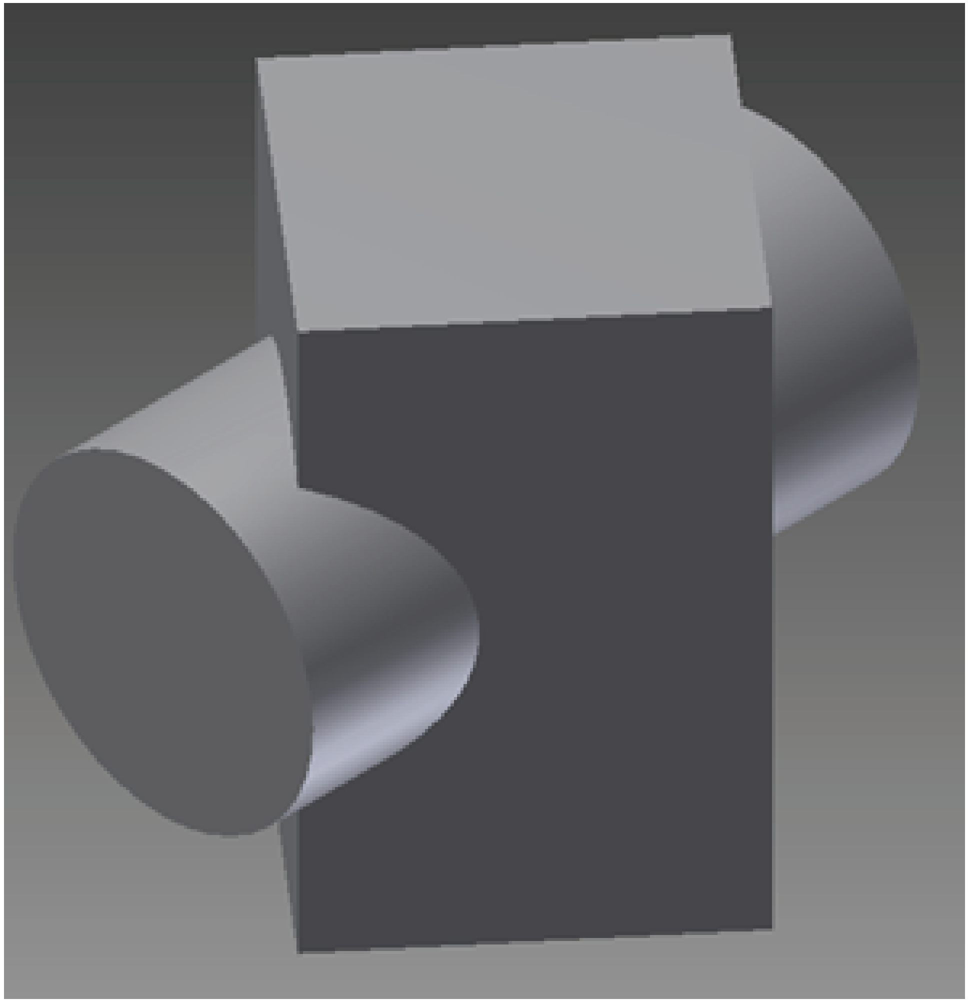
15. How to create curves of intersection of two solids?

In "Get started" tab, open a new ANSI.idw file and create orthographic views of the solid model showing the curves of intersection.
Add important dimensions



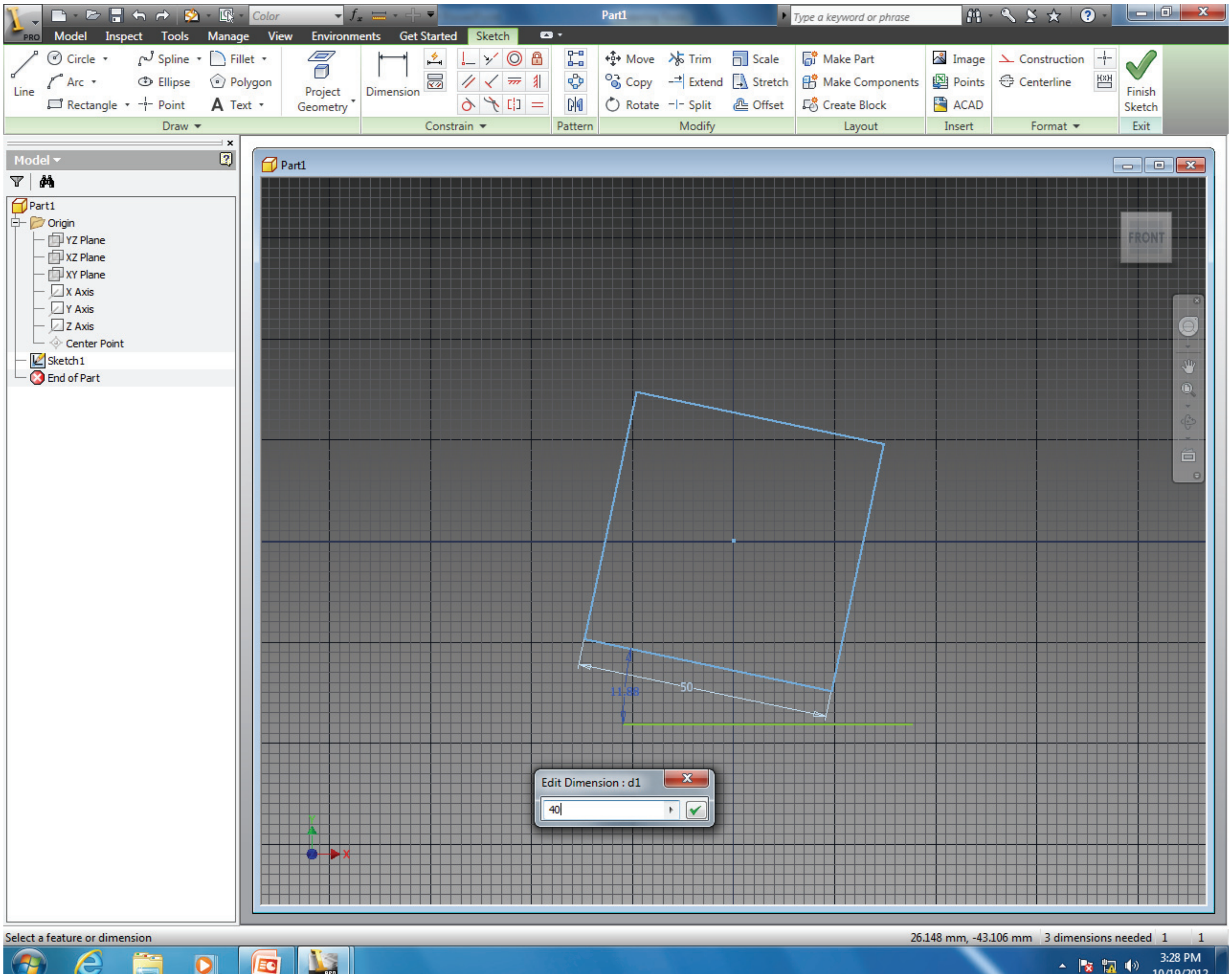
15. How to create curves of intersection of two solids?

- A square prism of base 50 mm and height 80mm is resting on one of its base edges inclined at 40 degrees to VP. A horizontal cylinder of diameter 30 mm and length 90mm, having its axis parallel to both HP and VP, intersects the prism bisecting its axis.
- Create the solid model and obtain curves of intersection



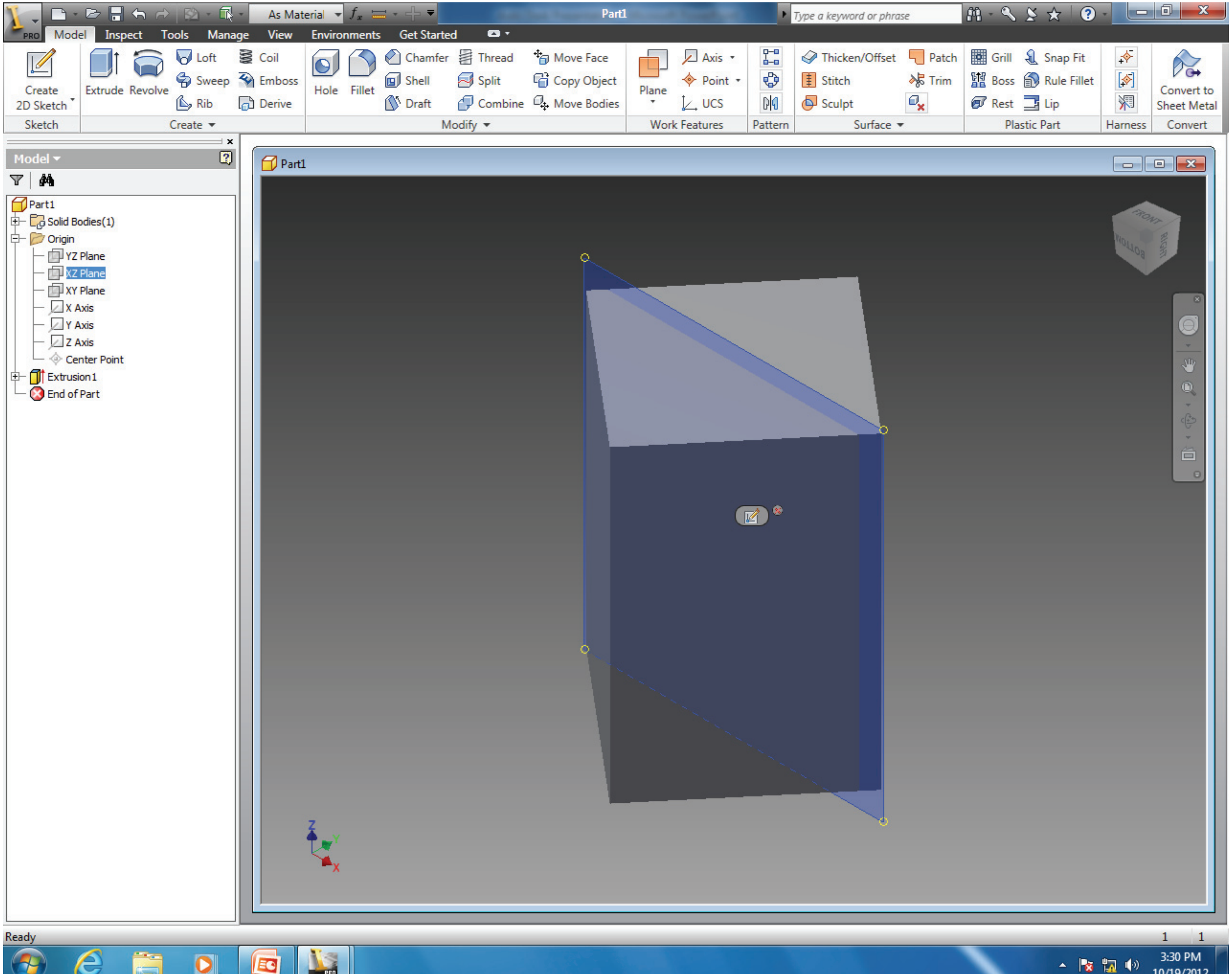
15. How to create curves of intersection of two solids?

Create square of 50 mm side, with one of its edge inclined at 40 degrees to horizontal reference line



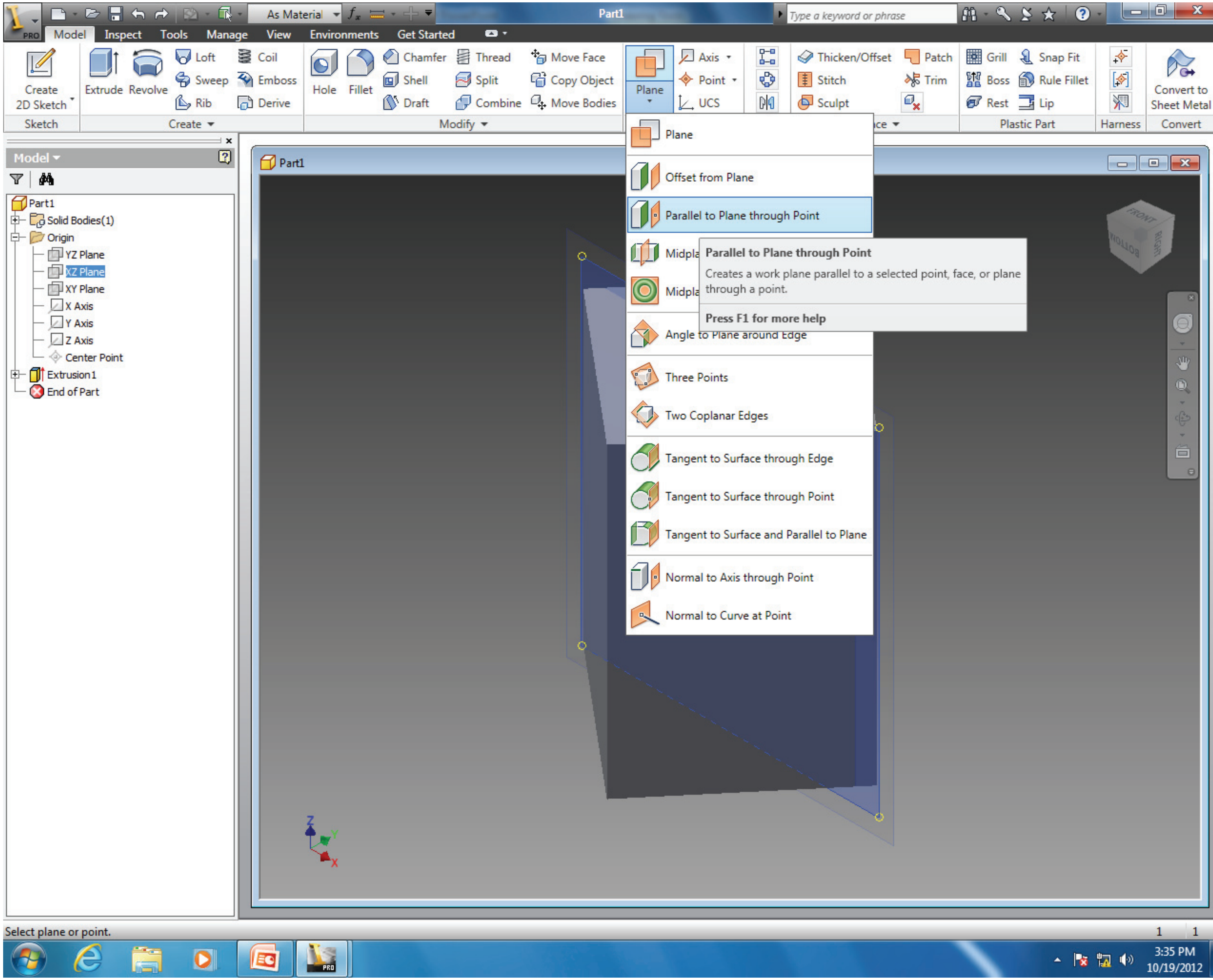
15. How to create curves of intersection of two solids?

Create the prism using Extrude command and make X-Z plane visible



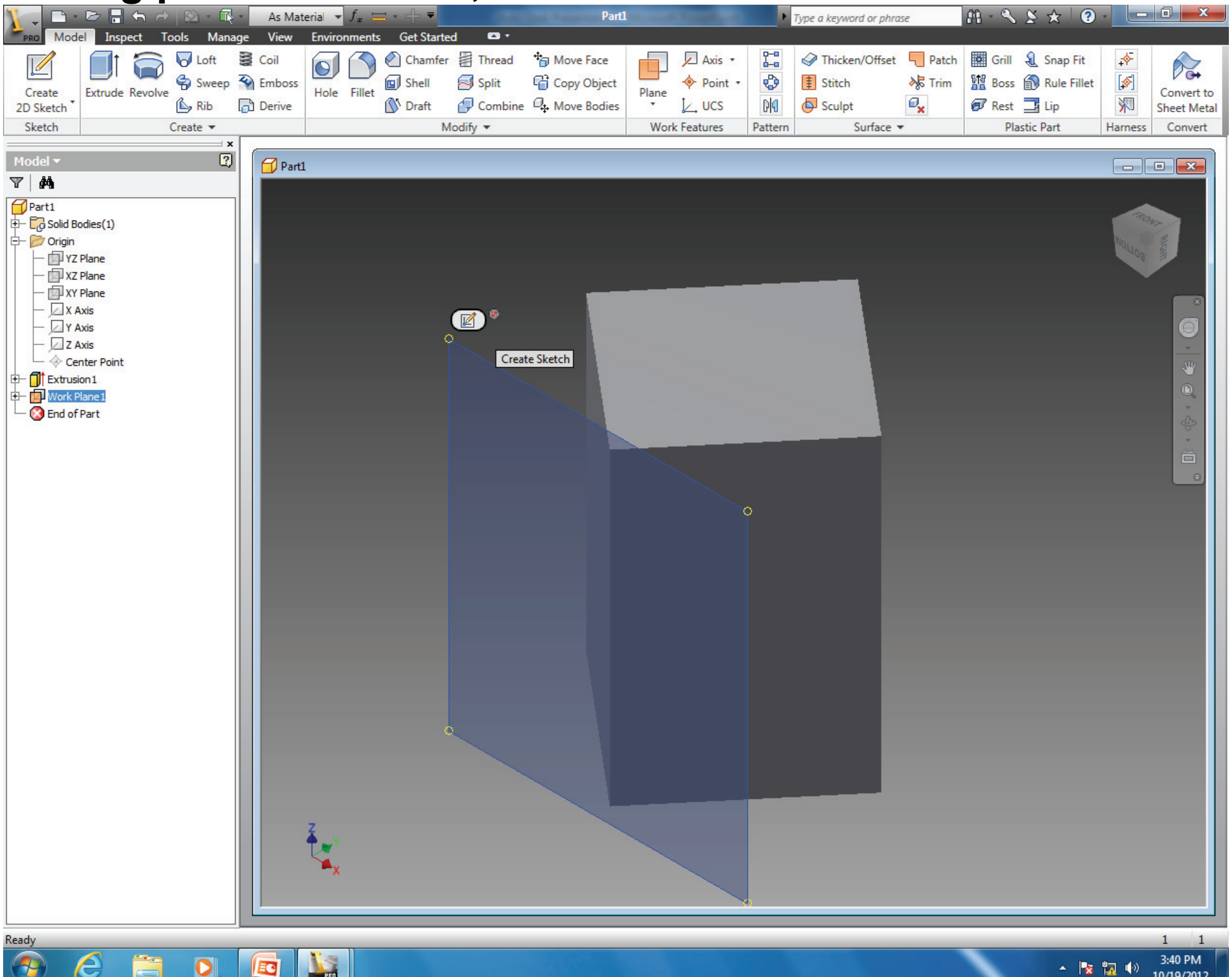
15. How to create curves of intersection of two solids?

Create a plane parallel to X-Z plane through one of the corners away from X-Z plane



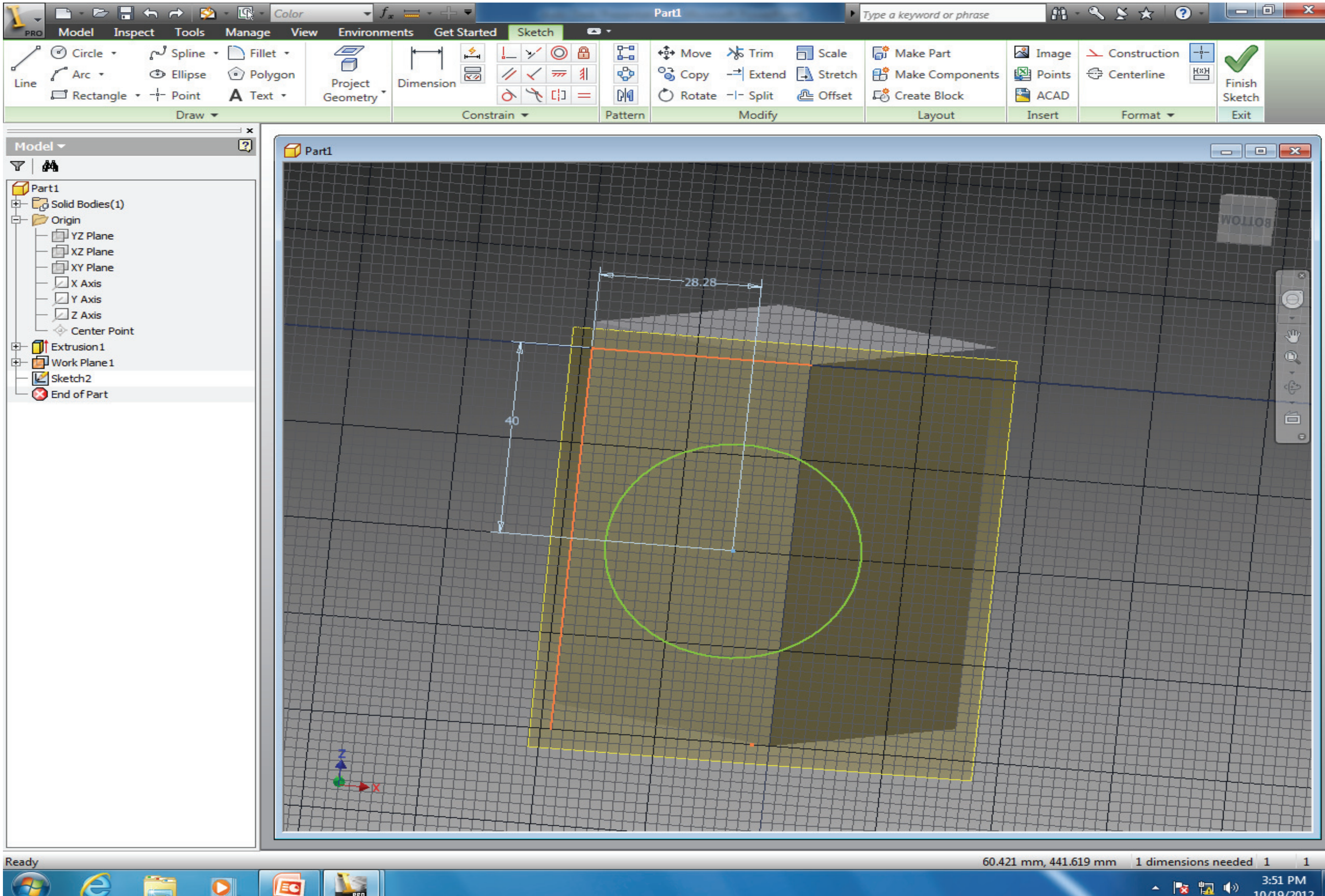
15. How to create curves of intersection of two solids?

A working plane is created, select it to create a sketch



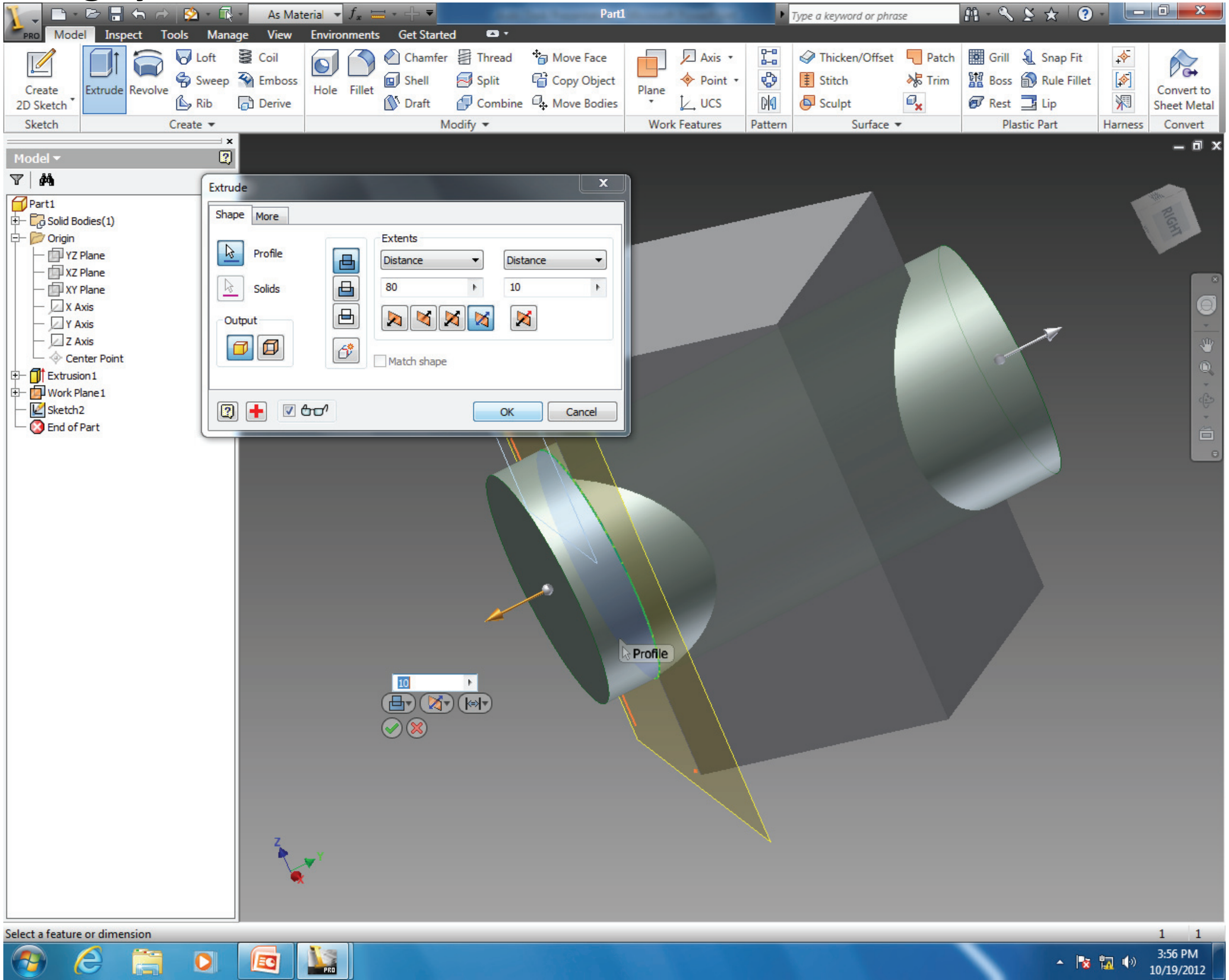
15. How to create curves of intersection of two solids?

Create a circle at required position on this working plane.



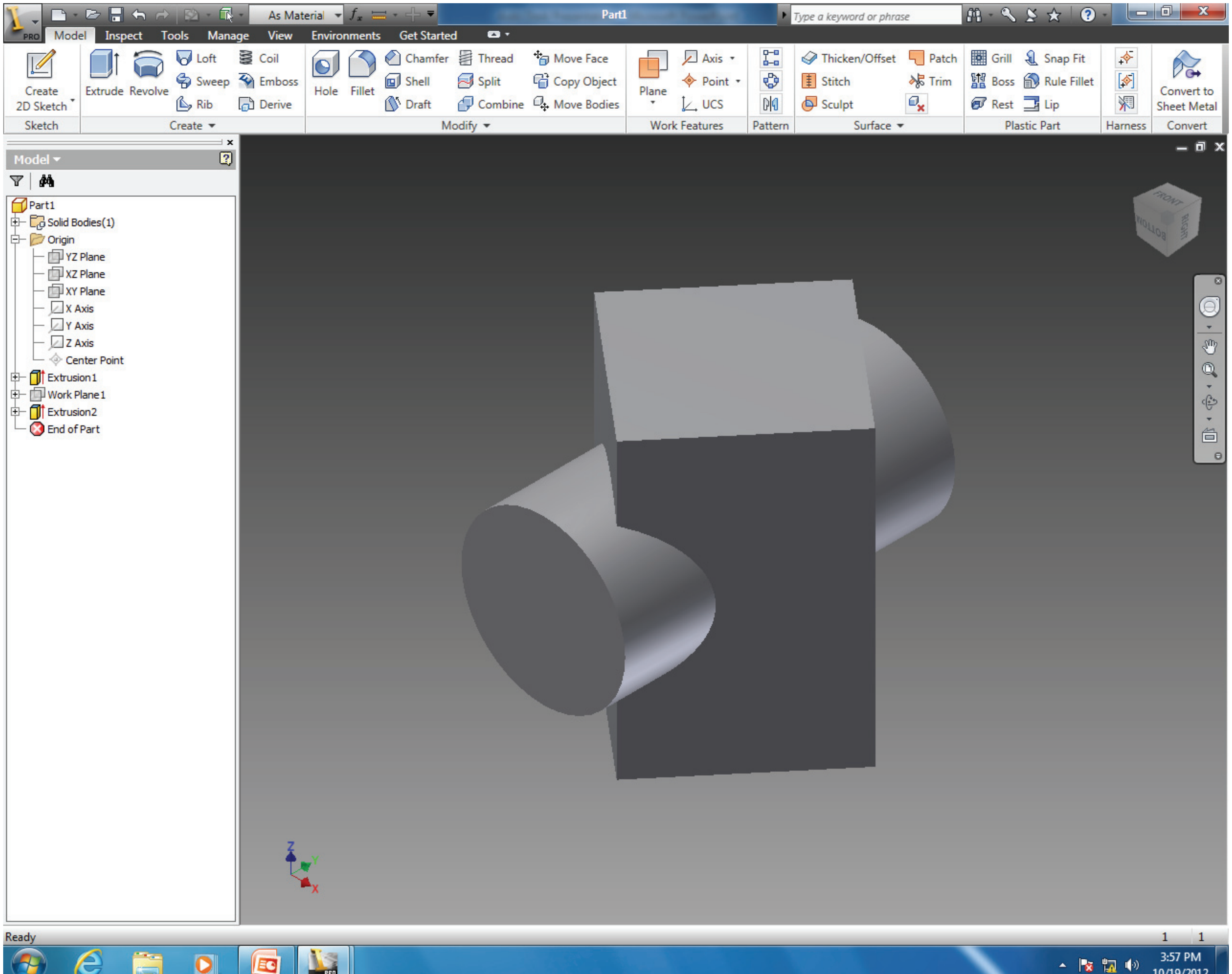
15. How to create curves of intersection of two solids?

Extrude the circle asymmetrically on both sides of the plane to create the penetrating cylinder



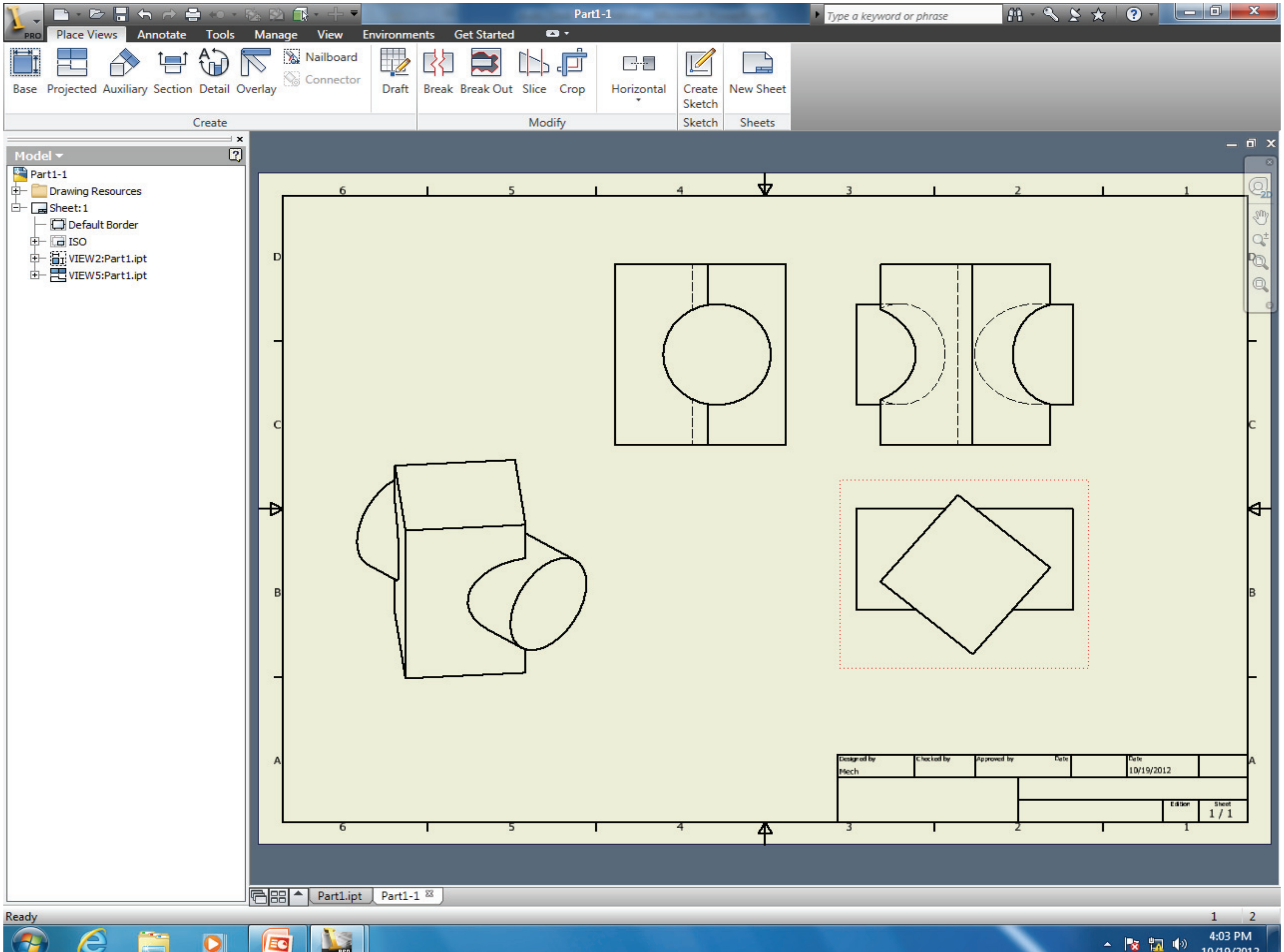
15. How to create curves of intersection of two solids?

Solid model is created, uncheck the visibility of the working plane



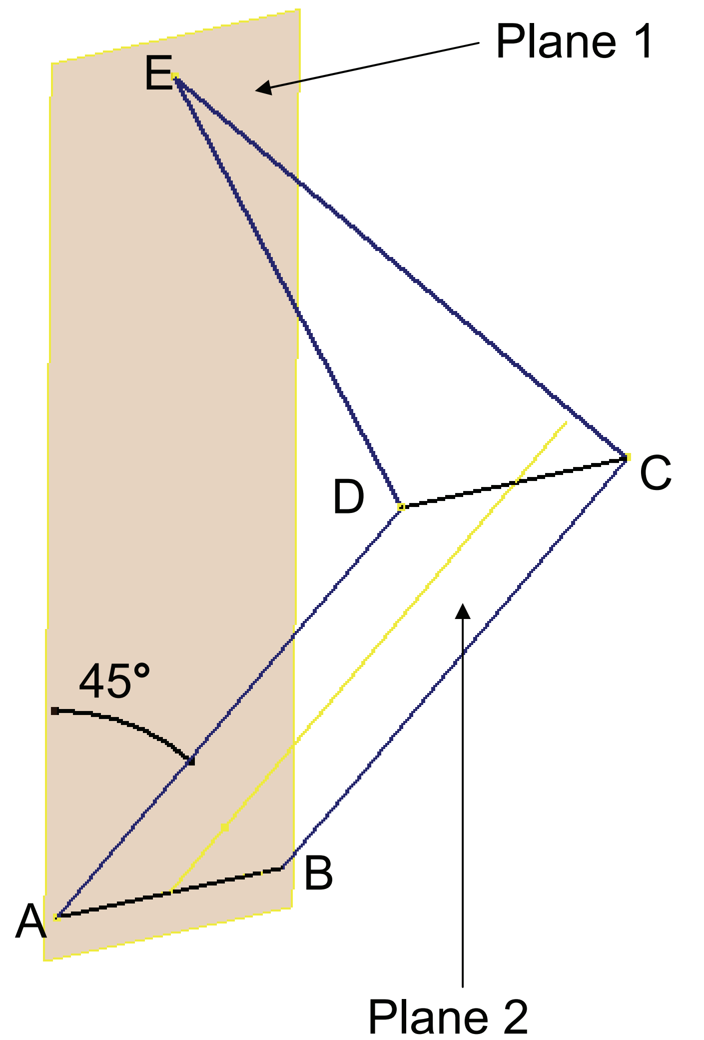
15. How to create curves of intersection of two solids?

Open a new ANSI(mm).idw file and create the orthographic projections of the intersecting solids to obtain the curves of intersection



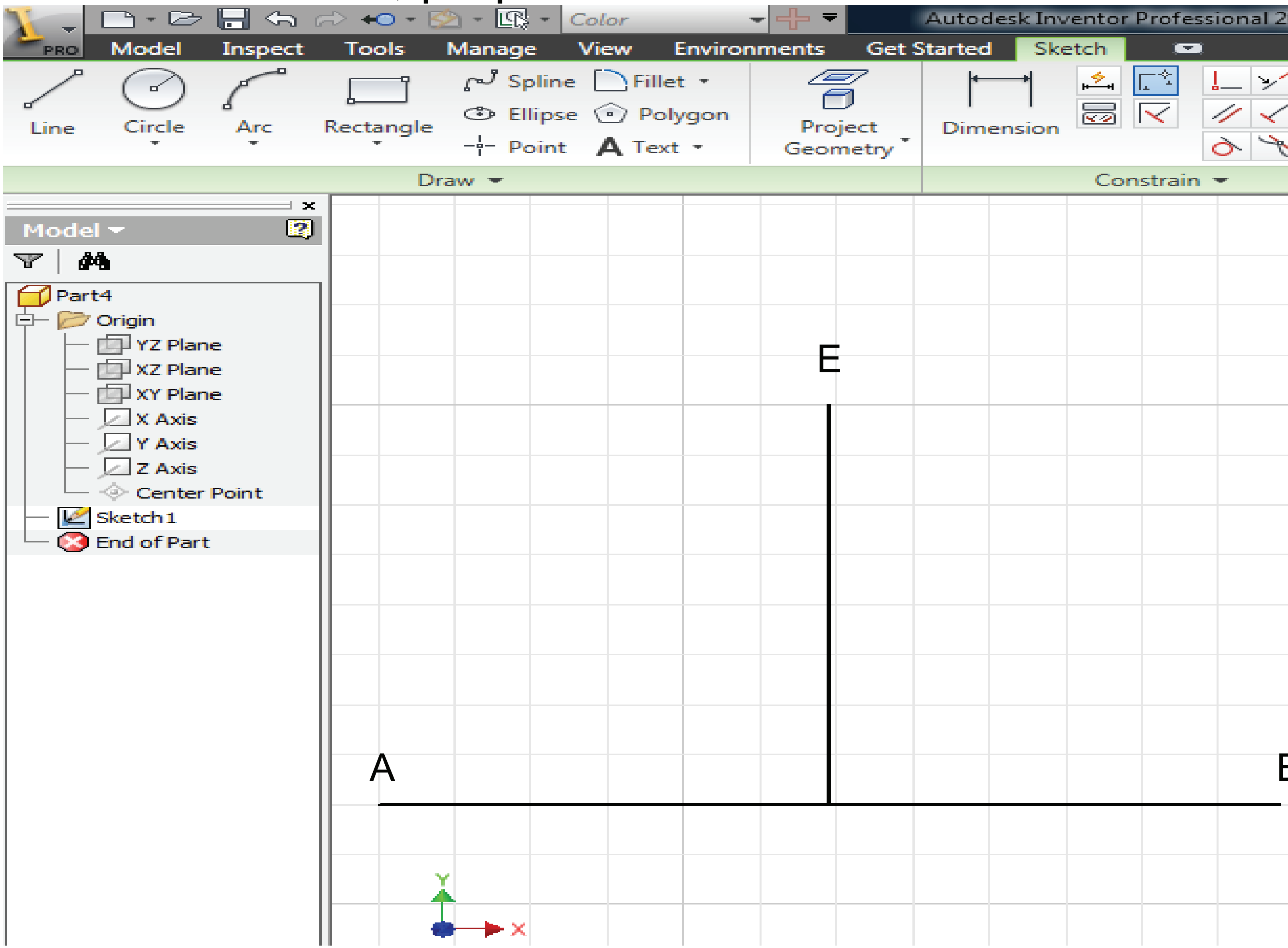
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Rectangle ABCD of given dimensions inclined at an angle of 45° with the vertical plane (plane 1)
- Distance between line AB and point E is given
- Measure length of line ED and DC
- Measure angle DEC using Inspect tool



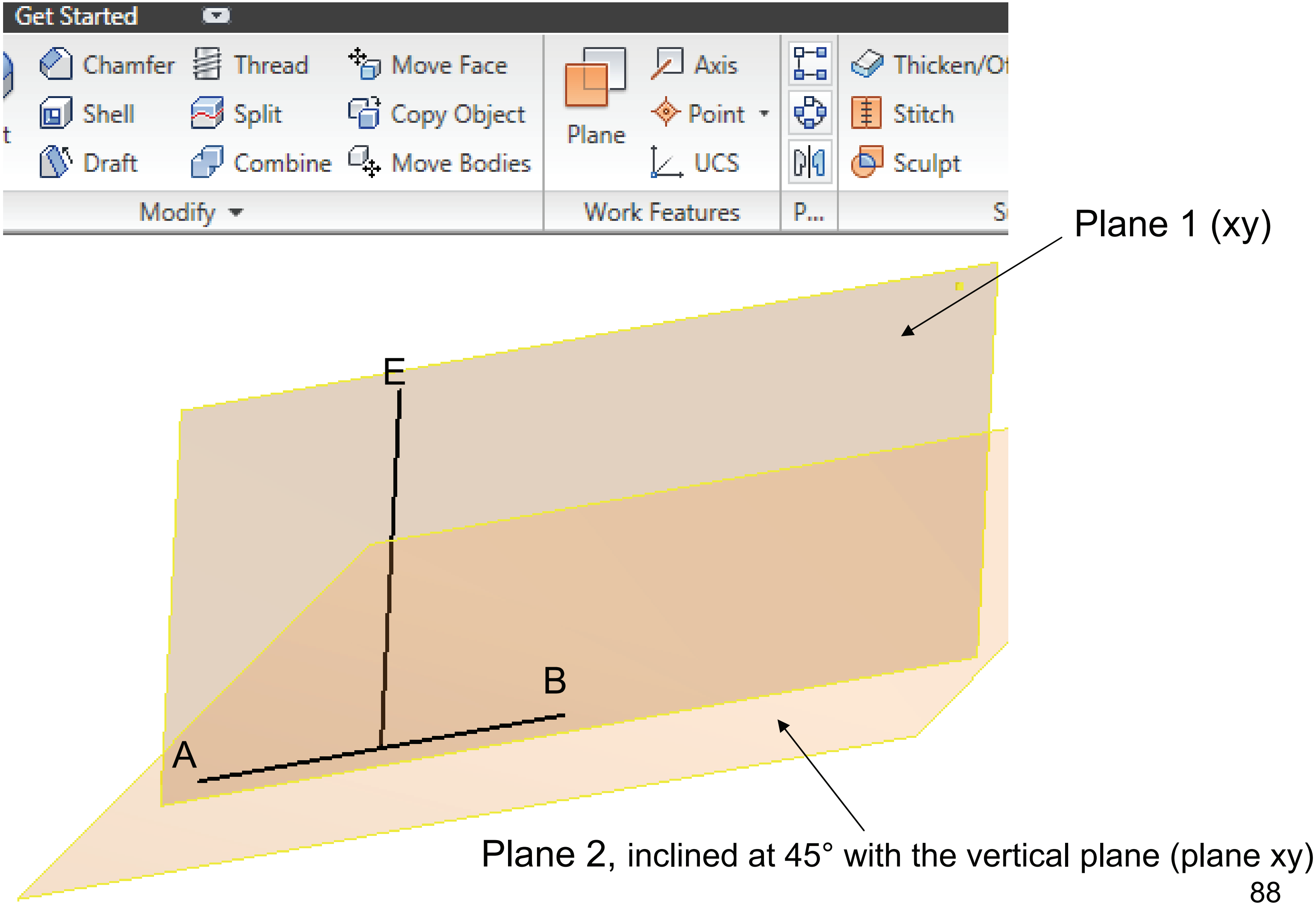
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Create line AB of given dimensions on plane xy
- Create vertical line, perpendicular to line AB



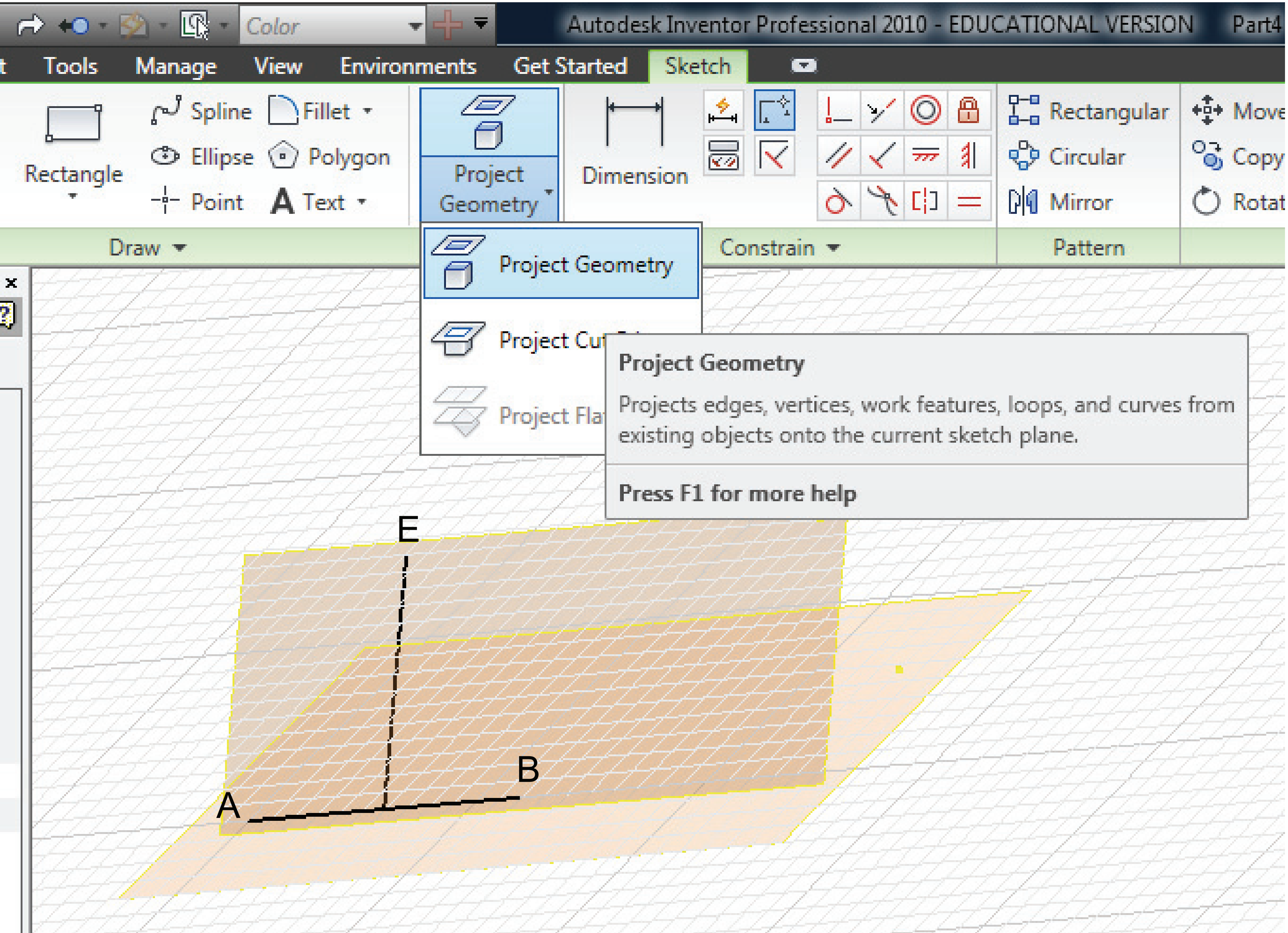
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Create a working plane 2 inclined at 45° with the vertical plane (plane xy)



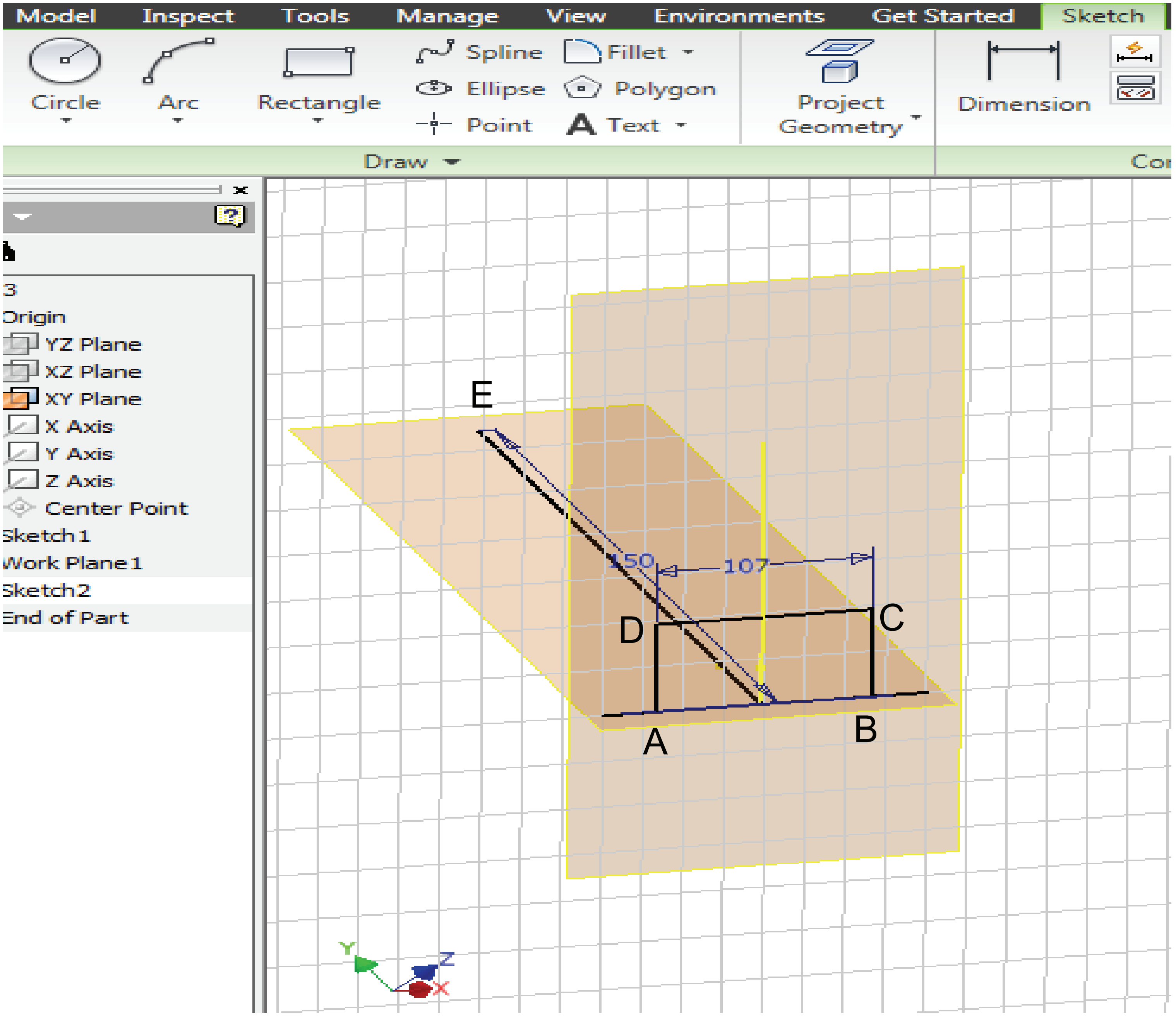
16. How to use inspect tool to measure dimensions of a 3D sketch?

•Project geometry on this working plane (plane 2)



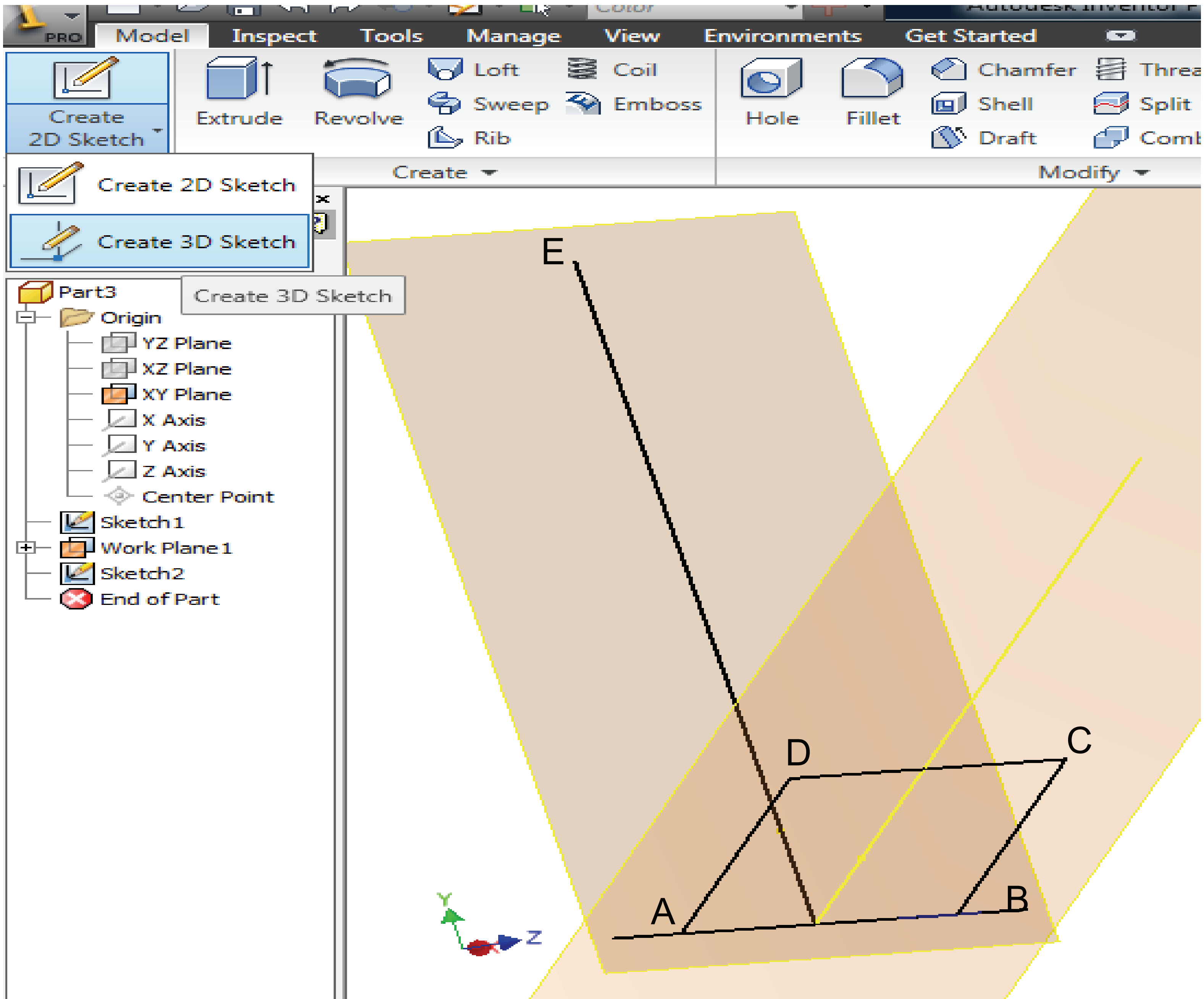
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Create rectangle ABCD of given dimensions on this working plane (plane 2) and finish the sketch



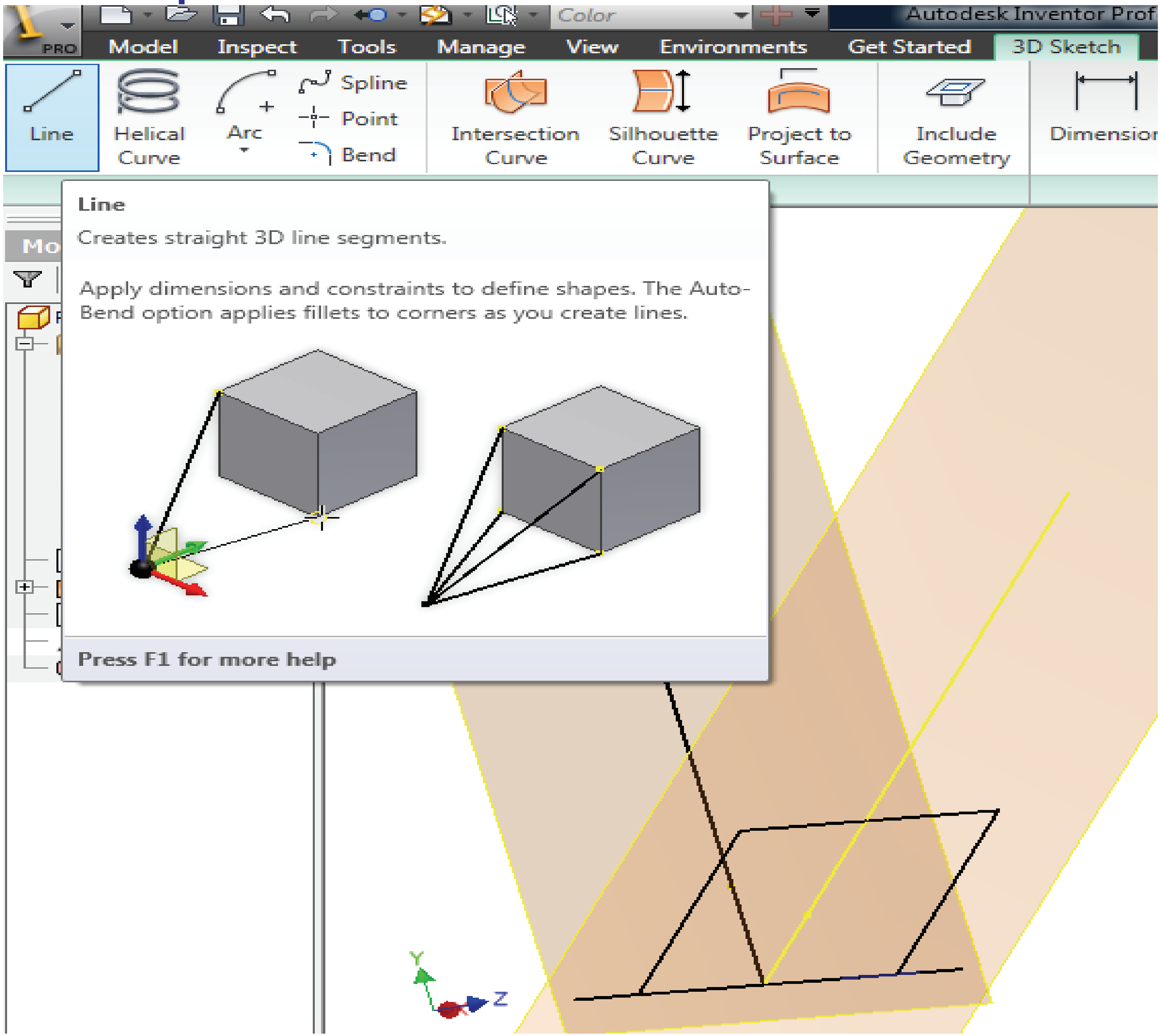
16. How to use inspect tool to measure dimensions of a 3D sketch?

•Create a 3D sketch



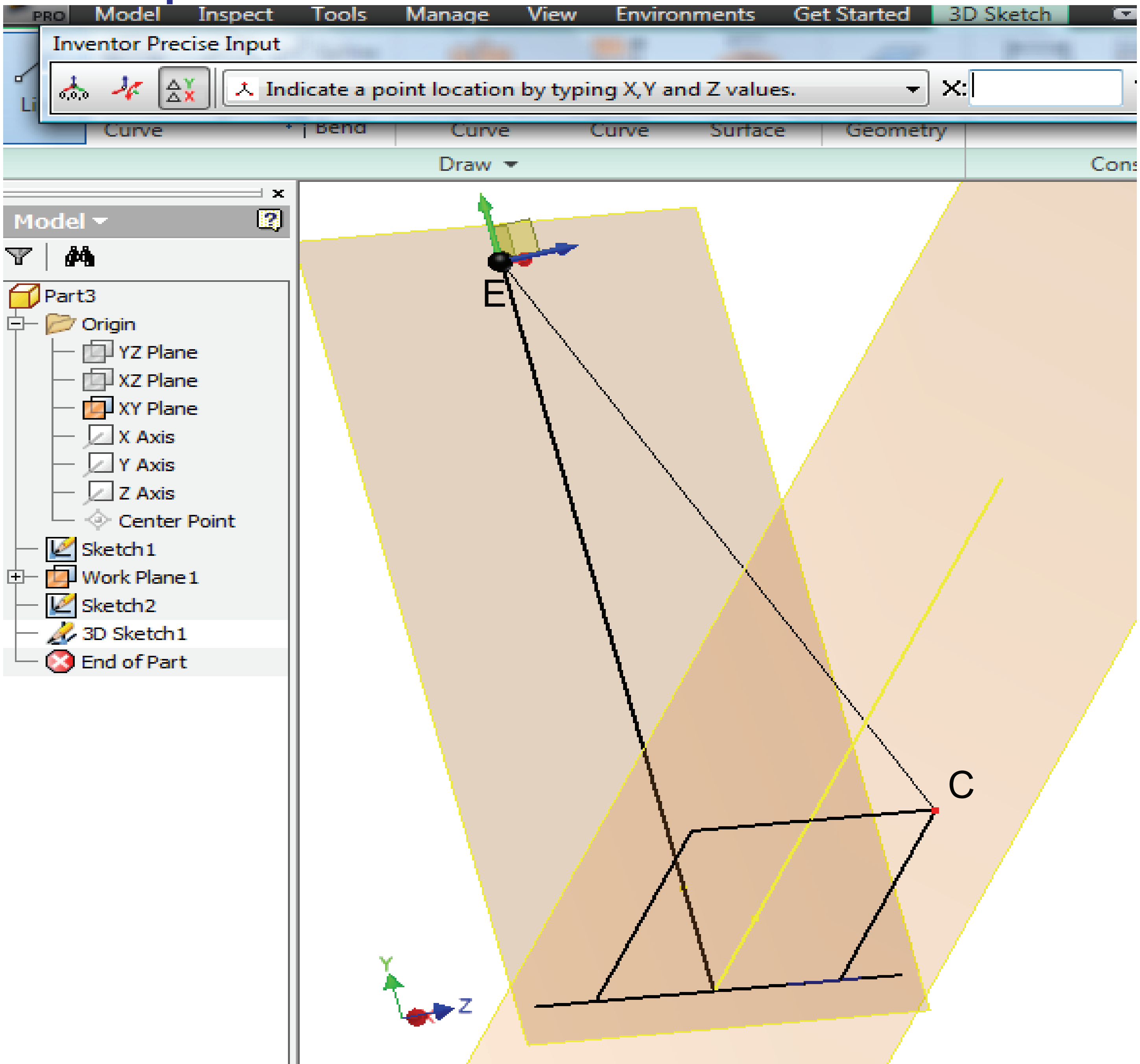
16. How to use inspect tool to measure dimensions of a 3D sketch?

•Create a 3D line



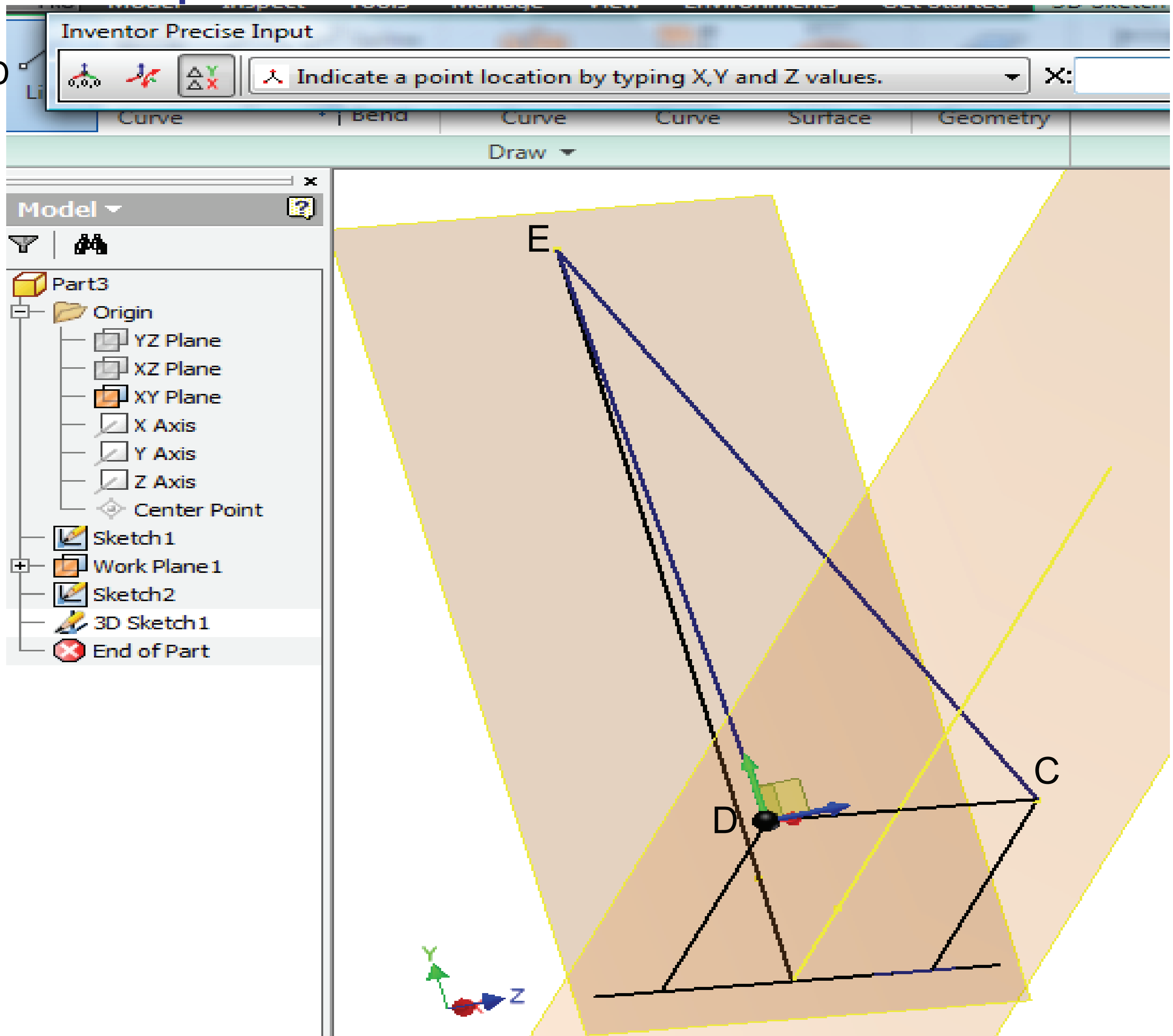
16. How to use inspect tool to measure dimensions of a 3D sketch?

•Create a 3D line, Join point E with C



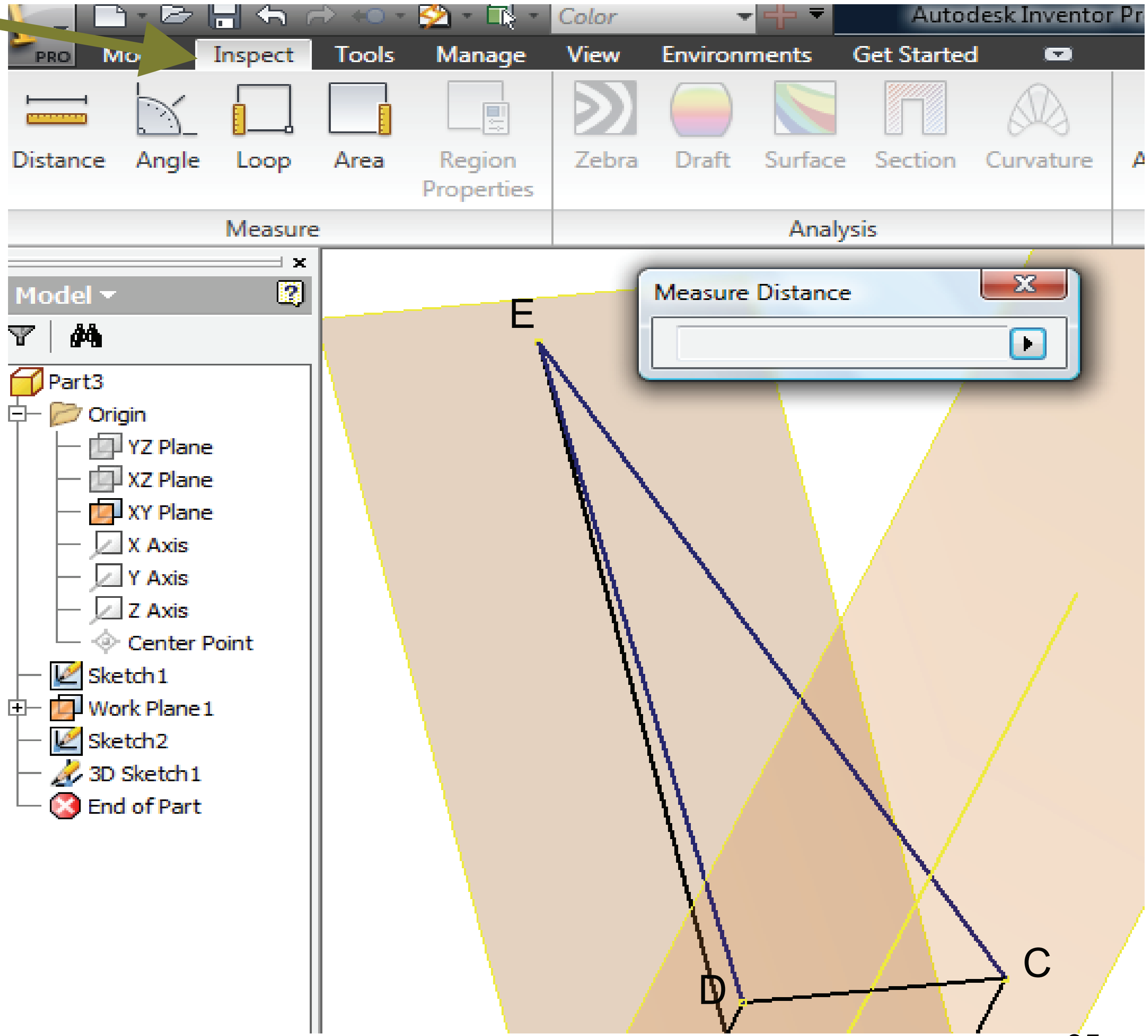
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Create a 3D line, Join point E with D And finish the sketch

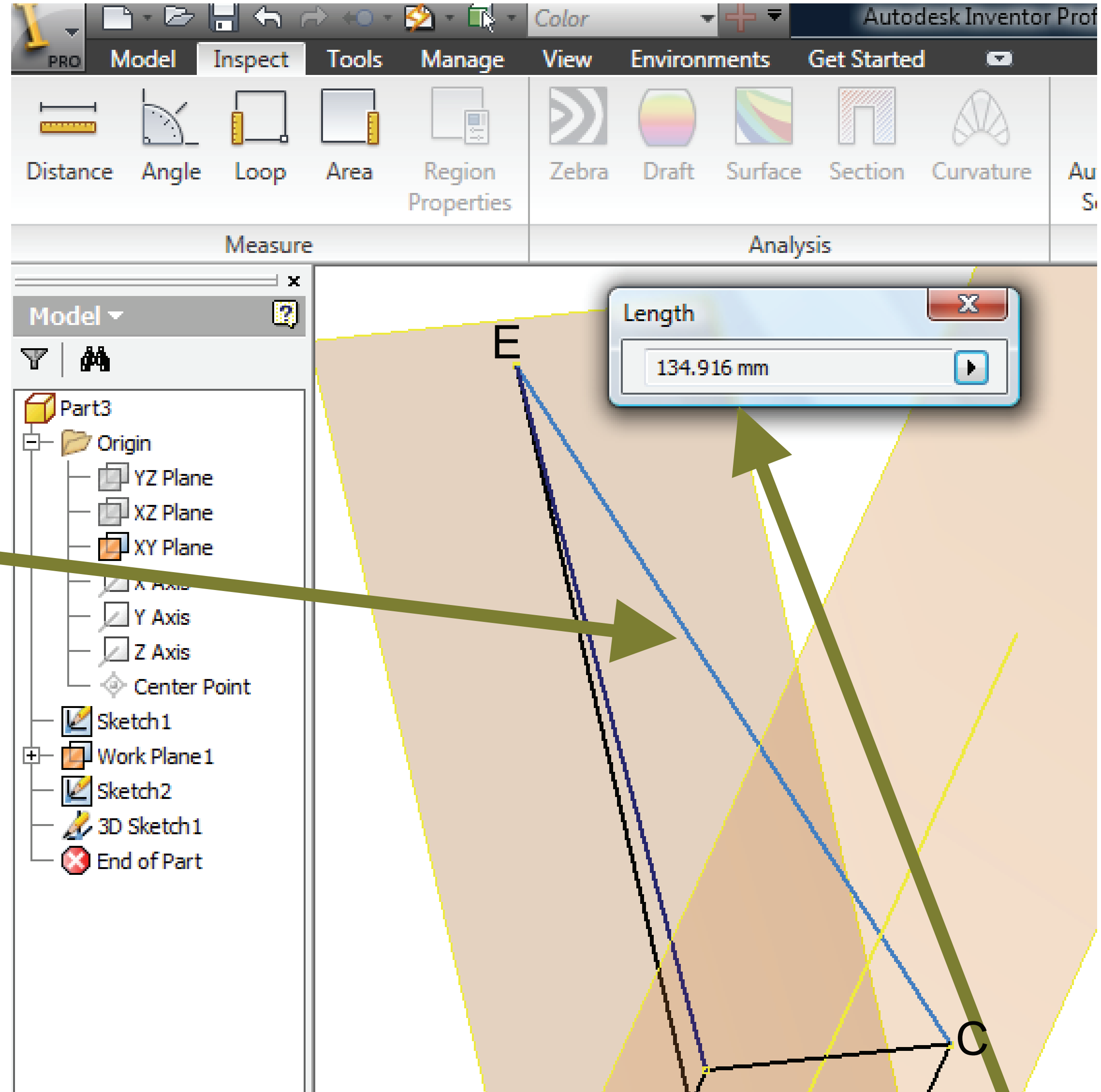


16. How to use inspect tool to measure dimensions of a 3D sketch?

- Click on Inspect menu to activate it
- Use Distance command to measure length of line ED and EC



16. How to use inspect tool to measure dimensions of a 3D sketch?

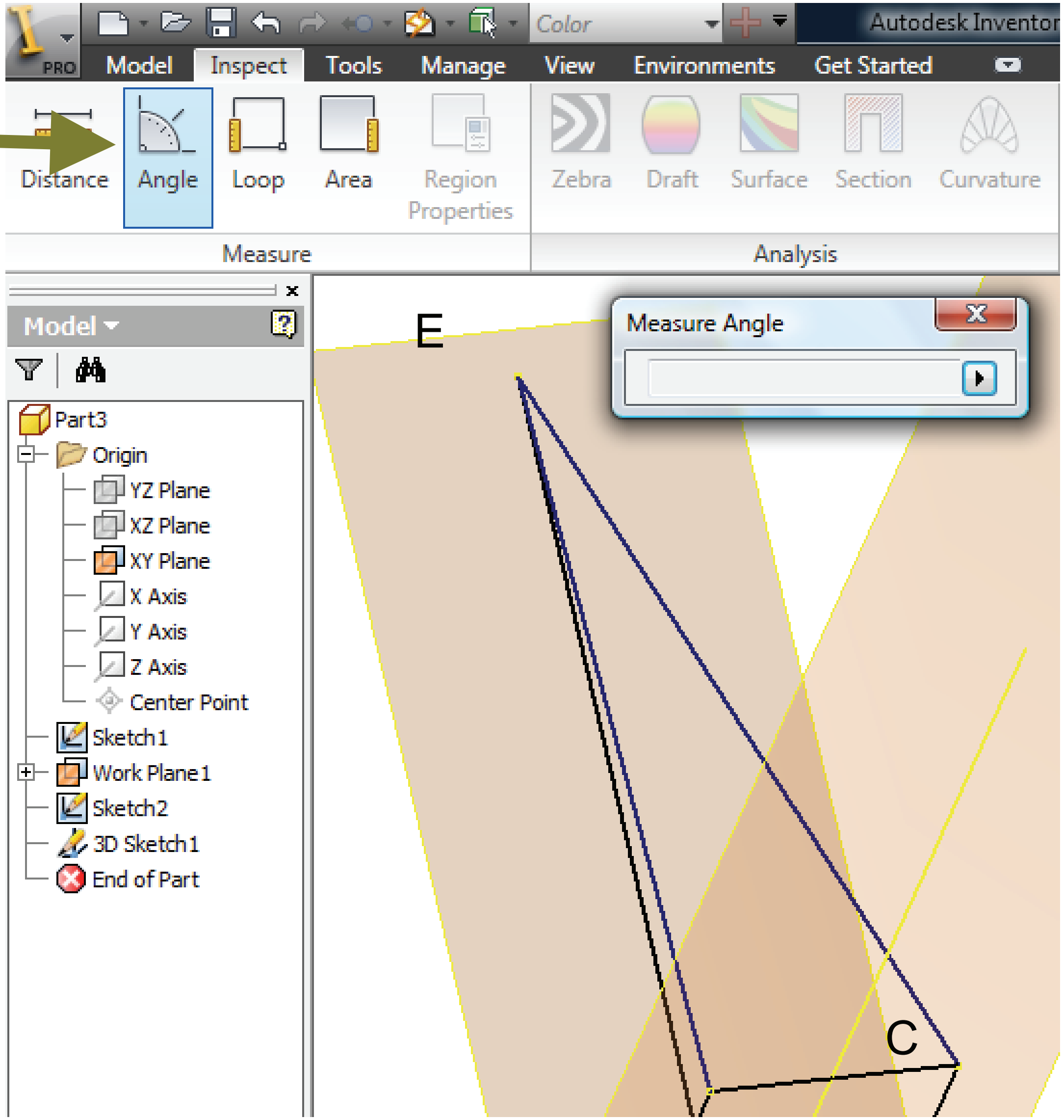


•Select line EC

The length of the line will be indicated

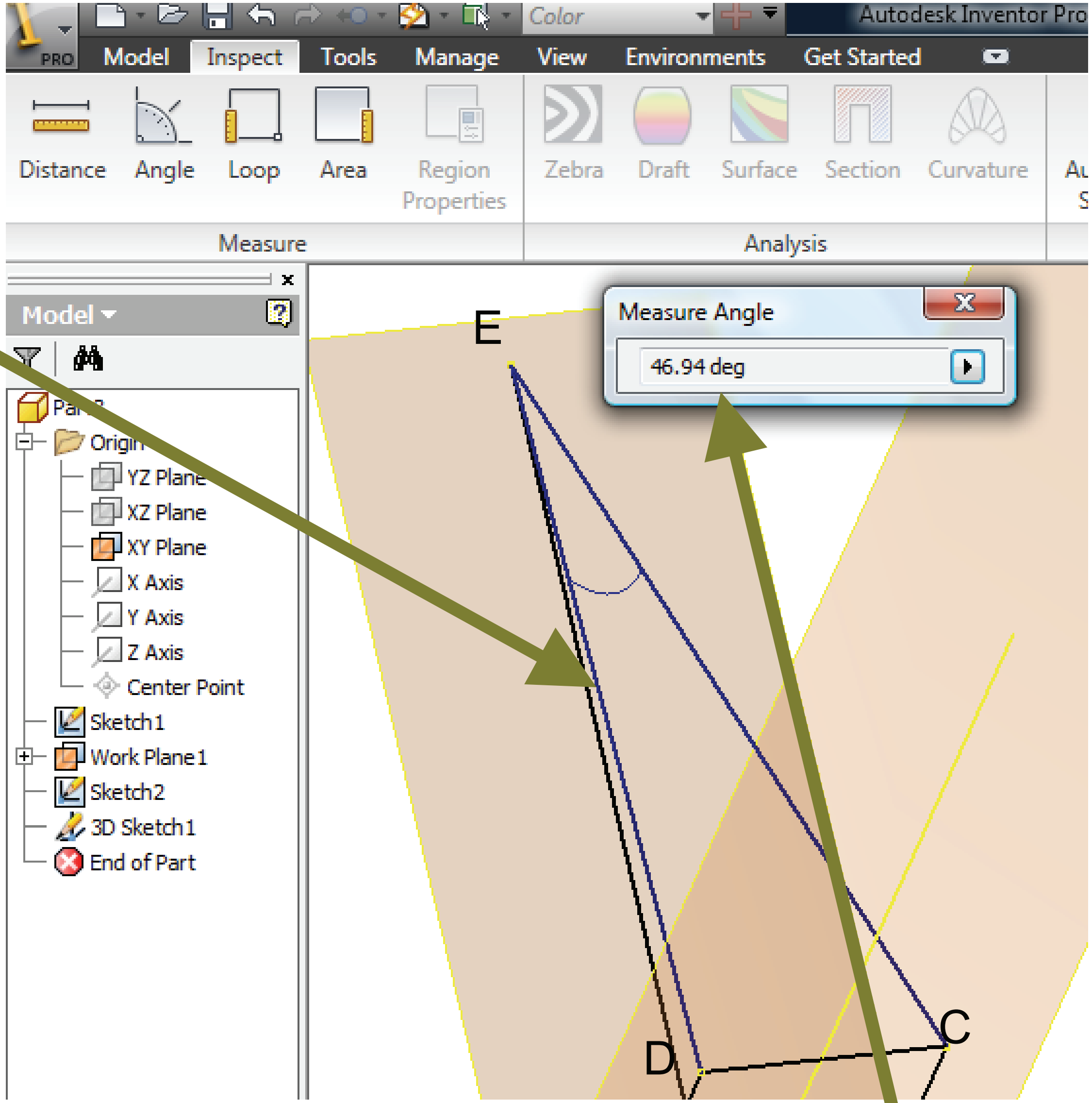
16. How to use inspect tool to measure dimensions of a 3D sketch?

- Use Angle command to measure angle between line ED and EC



16. How to use inspect tool to measure dimensions of a 3D sketch?

•Select lines ED and EC



The angle between the two lines will be indicated