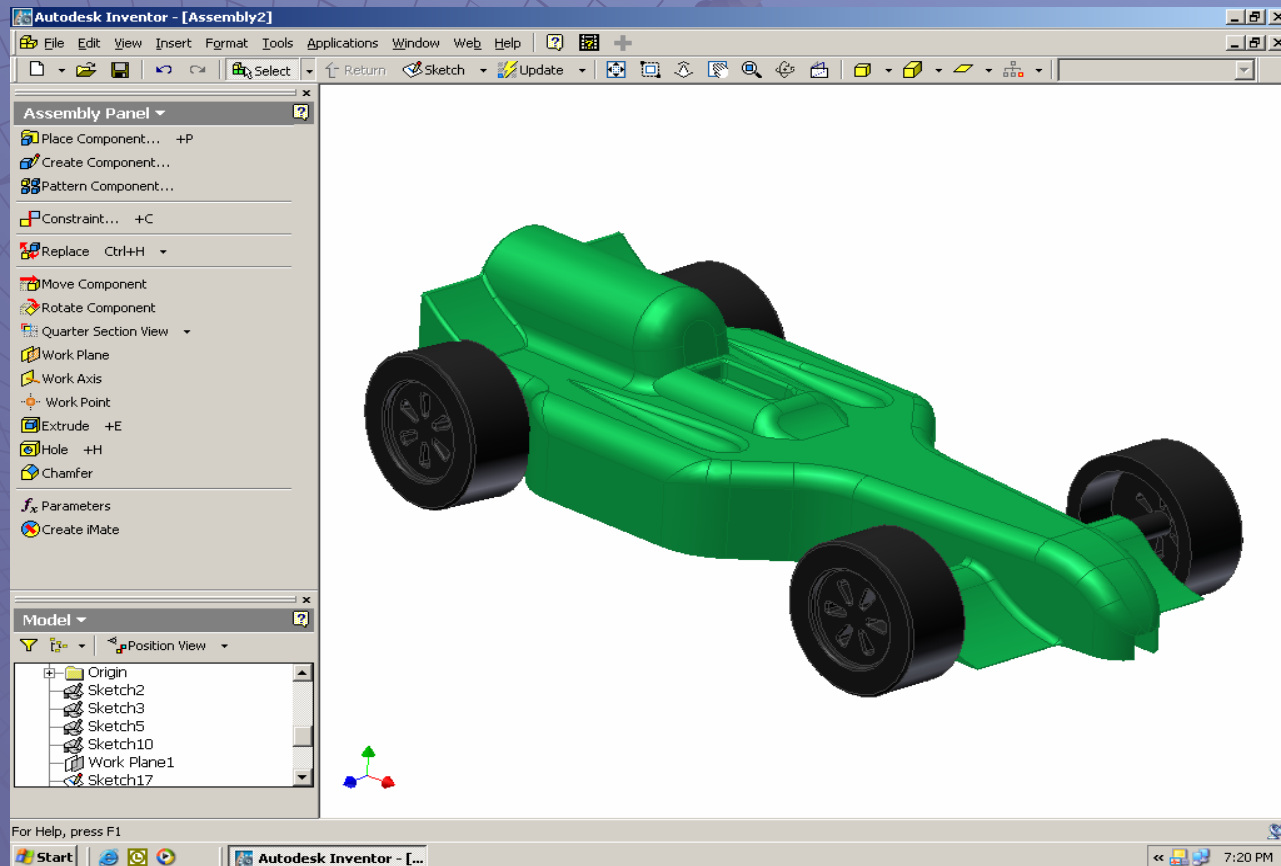


# Autodesk Inventor

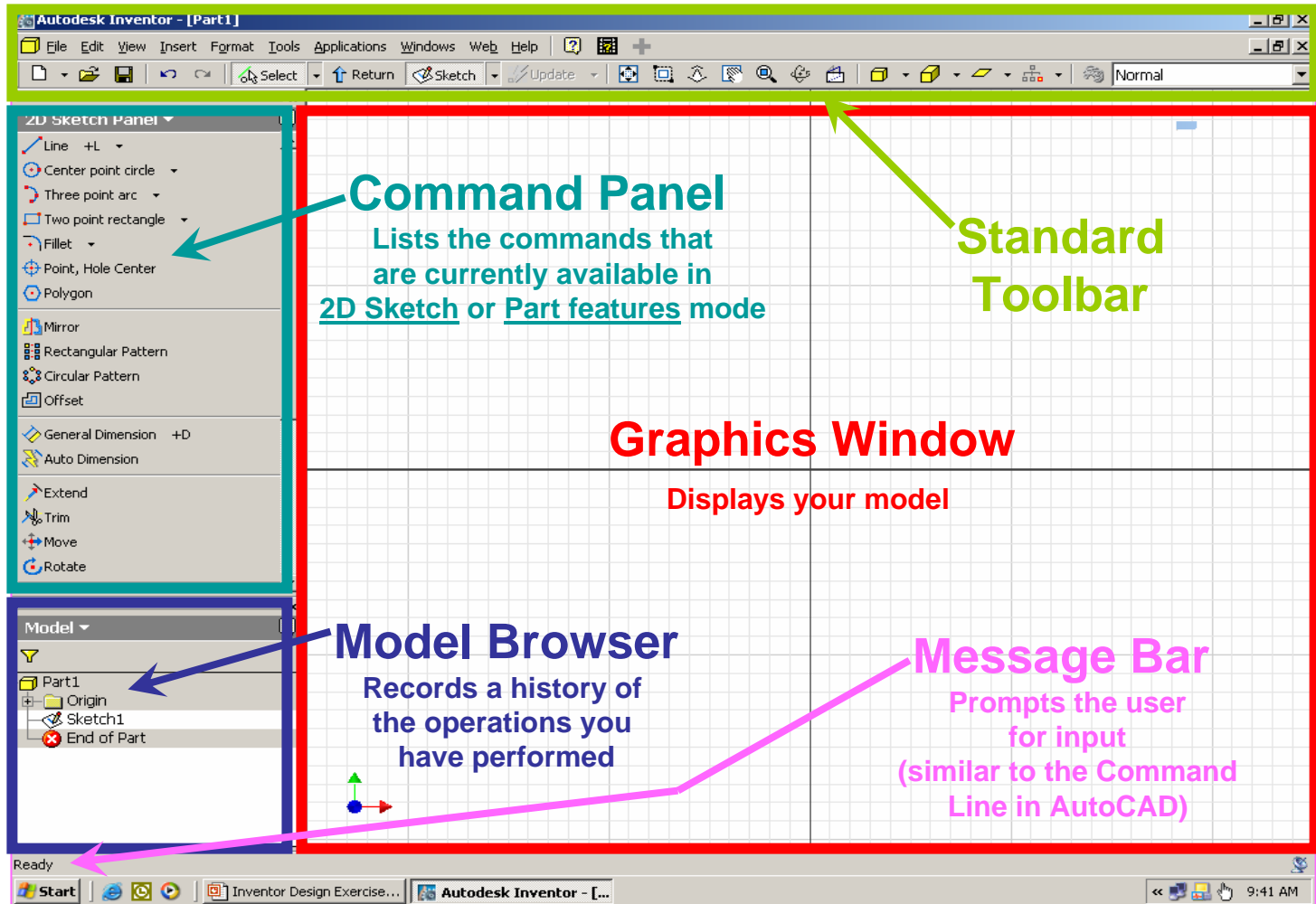
## Design Exercise 2:

# F1 Team Challenge Car

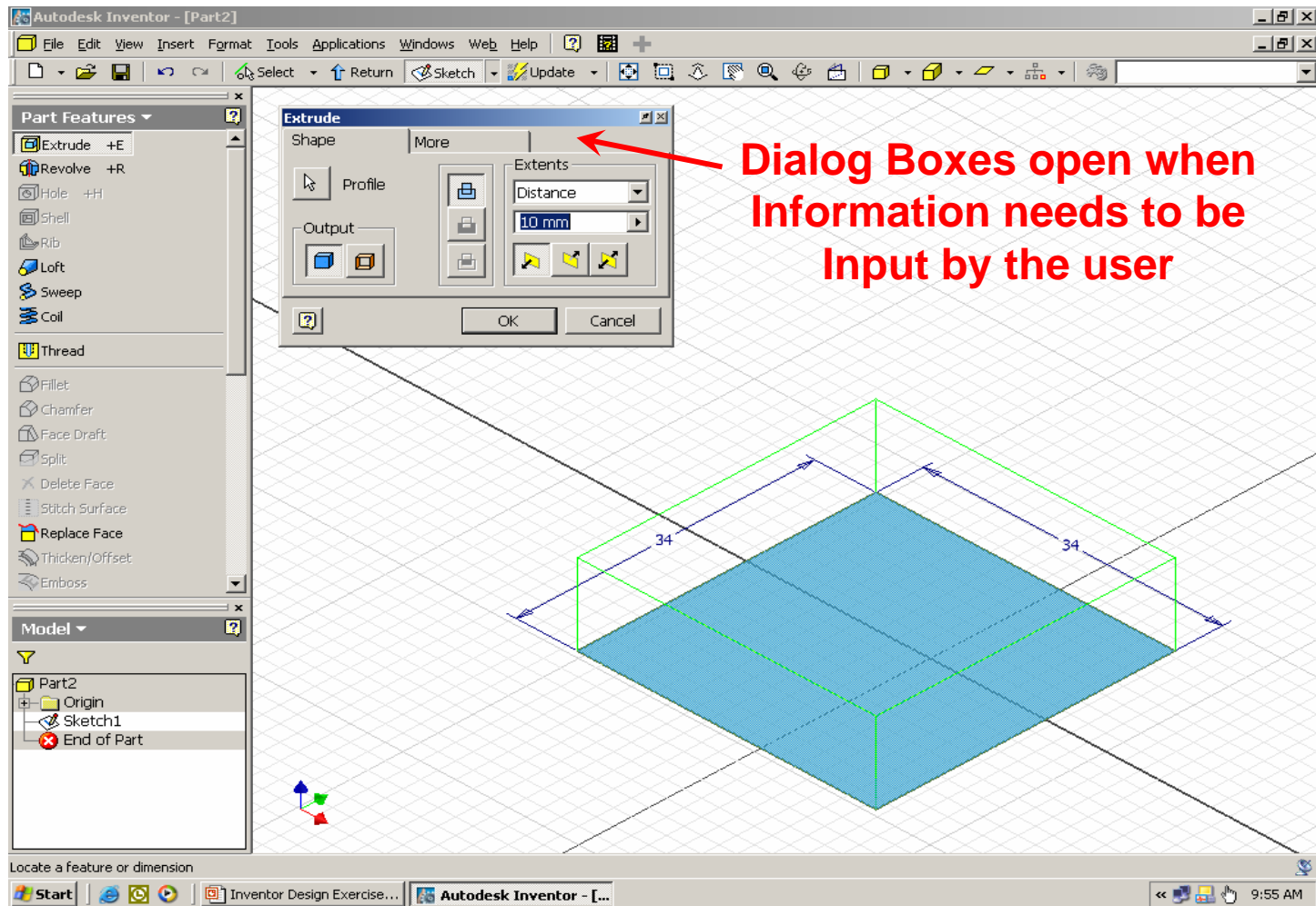
Developed by Tim Varner – Synergis Technologies



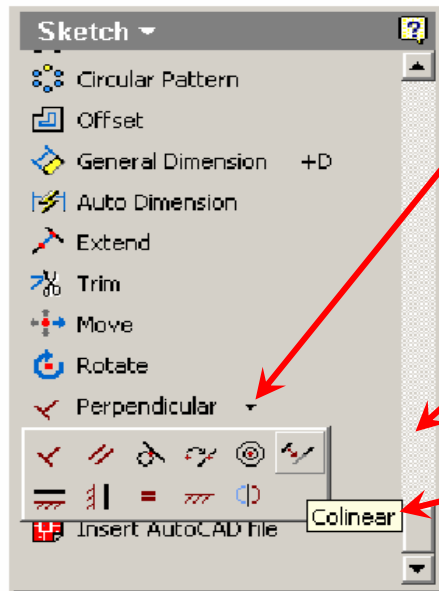
# The Inventor User Interface



# The Inventor User Interface



# More about the User Interface



Left clicking on a down arrow next to a command will open a pull-down menu with additional options

Slider bar – used for scrolling up and down

Fly out tabs show the user the command before selection

# Using the Mouse

The screenshot shows the Autodesk Inventor (tm) - [Part1] window. The interface includes a menu bar (File, Edit, View, Insert, Format, Tools, Applications, Windows, Help), a toolbar with various icons, and a command bar with 'Finish 2D Sketch'. On the left, there are two panels: 'Sketch' and 'Model'. The 'Sketch' panel lists tools like Circular Pattern, Offset, General Dimension, Auto Dimension, Extend, Trim, Move, Rotate, Perpendicular, Show Constraints, Project Geometry, and Insert AutoCAD file. The 'Model' panel shows a tree view with 'Part1' containing 'Origin', 'Sketch1', and 'End of Part'. A 'Finish 2D Sketch' context menu is open in the center, listing options such as Snap to Grid, Create Line (+L), Create Dimension (+D), Create Constraint, Create Feature, Slice Graphics, Measure Distance, Measure Angle, Measure Loop, Measure Area, Create iMate, Previous View (+F5), Isometric View, and How To... A red arrow points from the text 'A click with the right (R) mouse button invokes "pop-up" menus' to the 'Finish 2D Sketch' menu. Another red arrow points from the text 'The left (L) mouse button selects commands and objects' to the 'Finish 2D Sketch' menu. A third red arrow points from the text 'The mouse wheel is used to Zoom and Pan' to the 'Finish 2D Sketch' menu. A fourth red arrow points from the text 'The Escape key ends commands' to a small red dot with a green arrow pointing up and a blue arrow pointing right, representing the escape key.

**A click with the right (R) mouse button invokes "pop-up" menus**

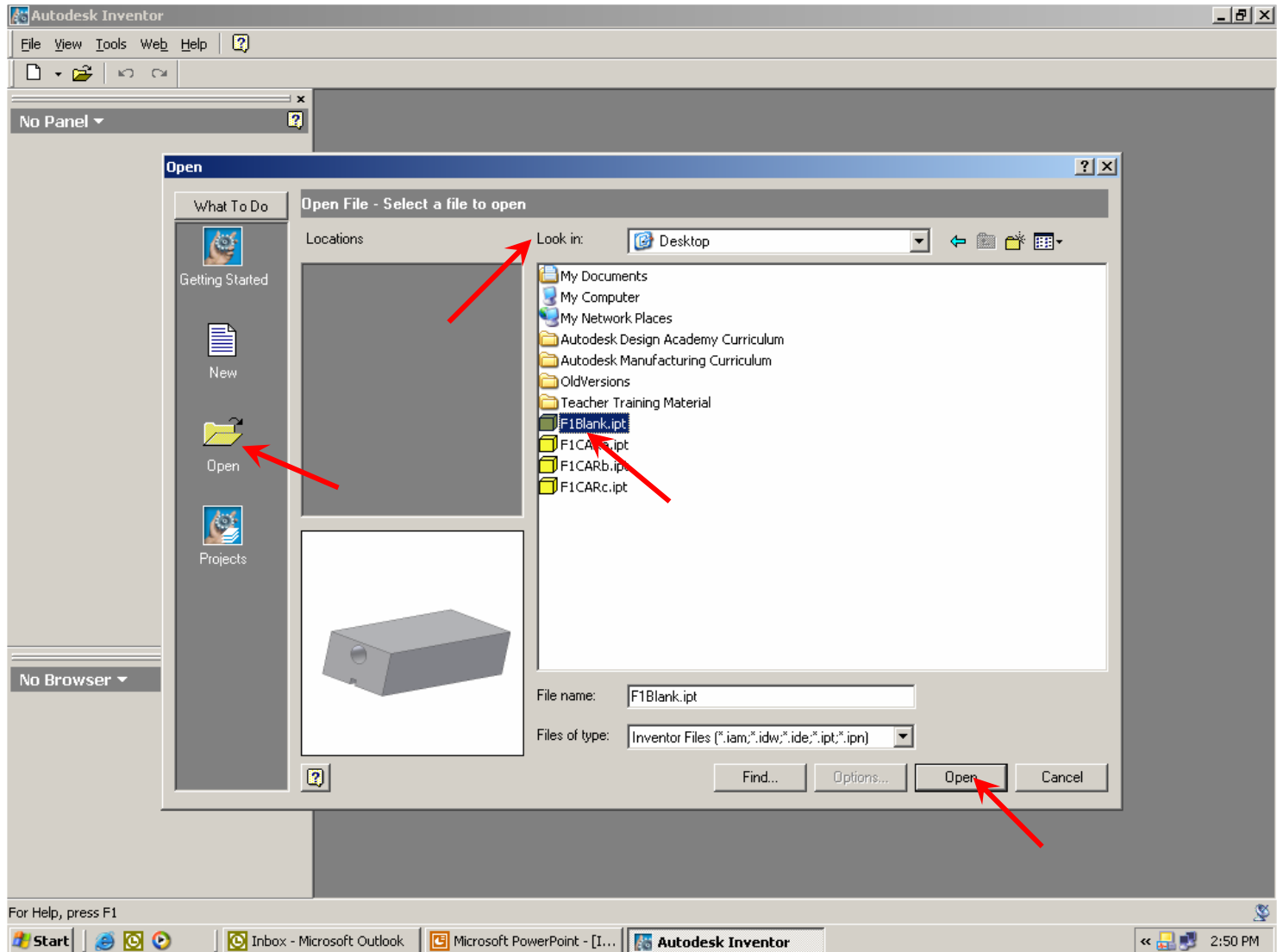
**The left (L) mouse button selects commands and objects**

**The mouse wheel is used to Zoom and Pan**

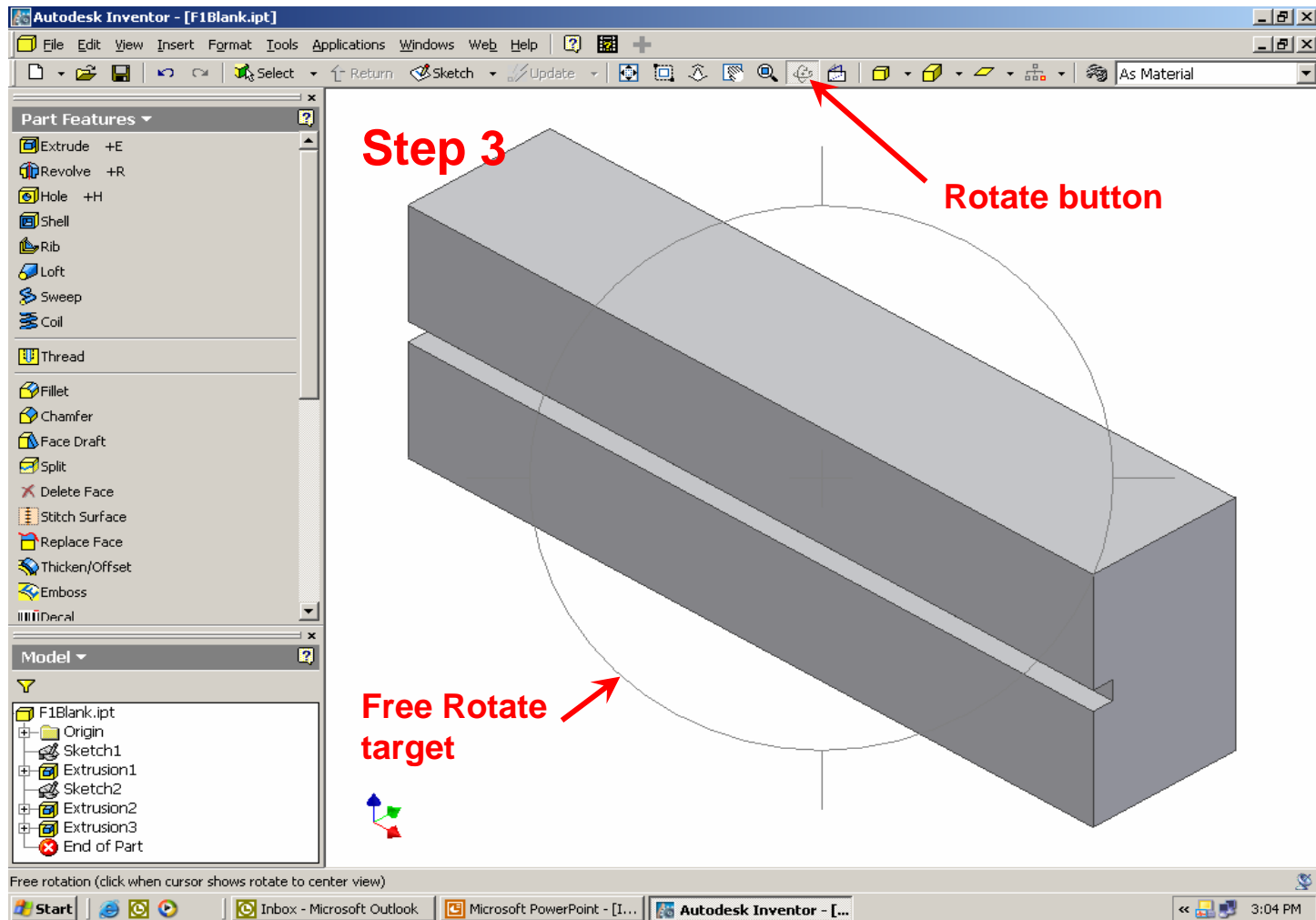
**The Escape key ends commands**

**Step 1.** – Launch Autodesk Inventor by double (L) clicking the **Inventor Icon** on the Windows desktop.

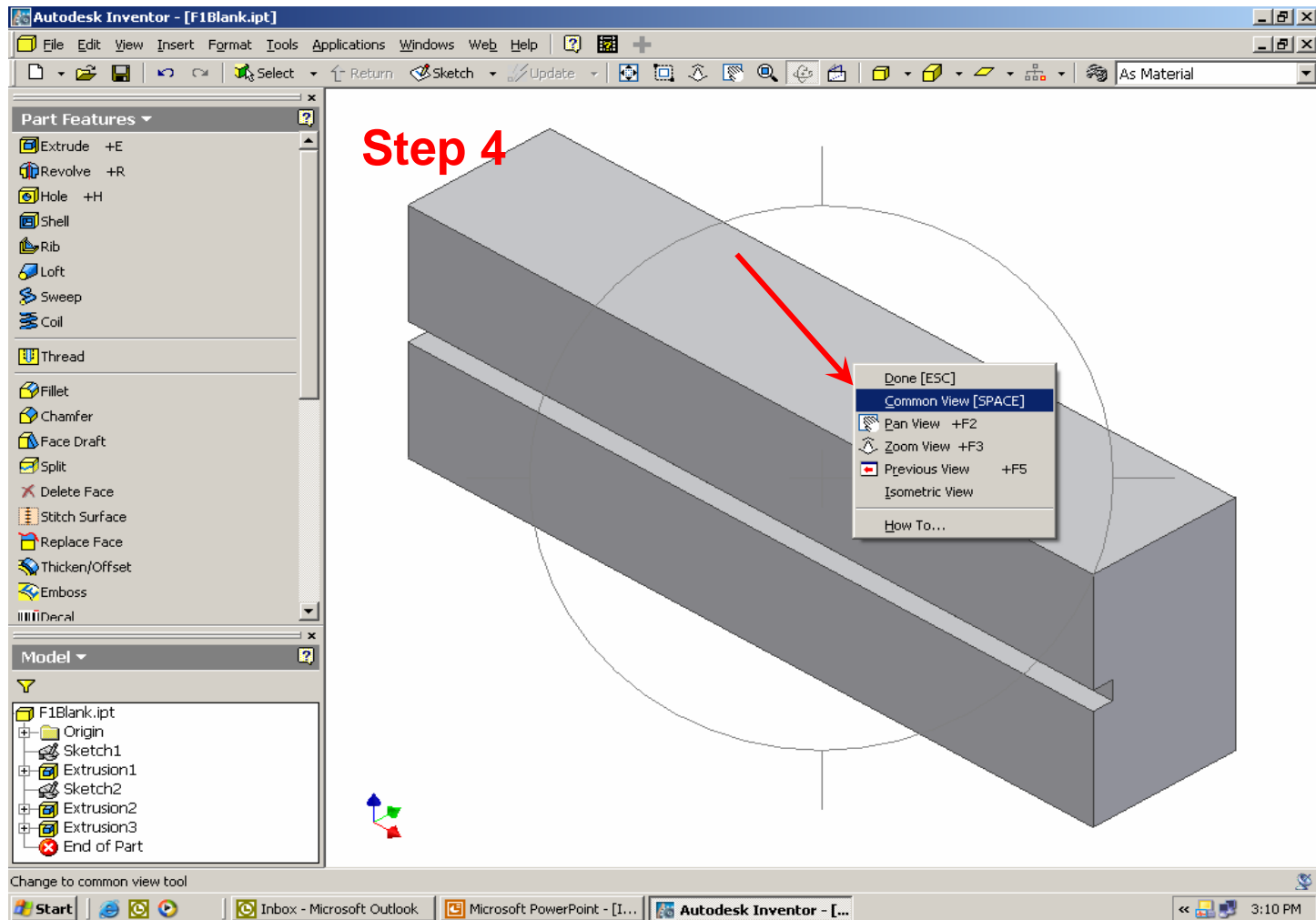
**Step 2.** – (L) click on **Open**. Your Instructor will tell you which directory to “**Look in**”. (L) click on the directory then (L) click on the file **F1Blank.ipt**. (L) click on **Open**.



**Step 3.** – (L) click on the **Rotate** button in the Standard Toolbar and a “free rotate target” will appear on the screen. Move the cursor inside the target, press and hold the left mouse button while moving the mouse. Notice how the view rotates. Next, move the cursor just outside the target; press and hold the left mouse button while moving the mouse. Notice how the view rotates.

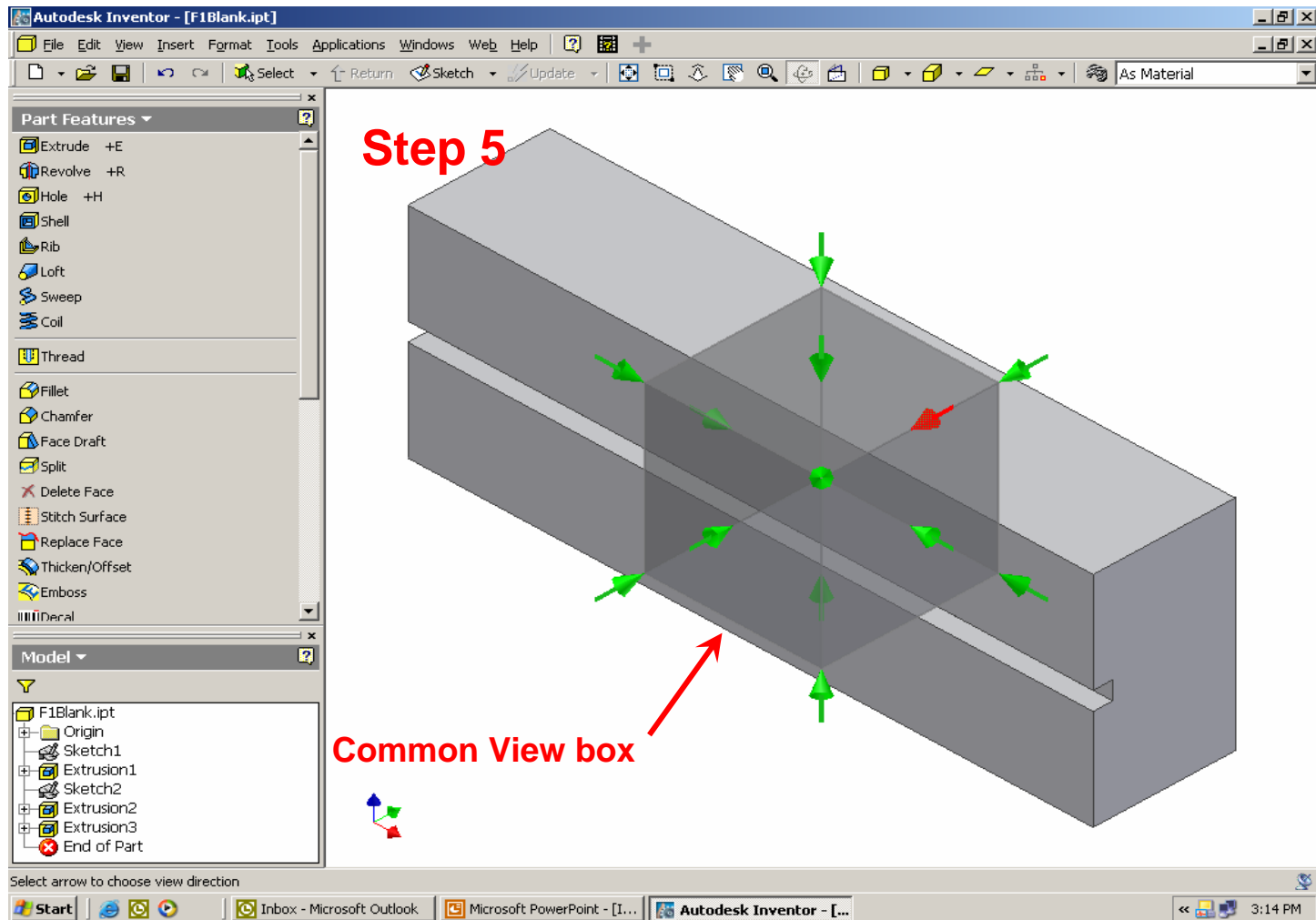


**Step 4.** – Practice with the **Free Rotate** command until you feel comfortable using it. Now move the cursor inside the **Free Rotate** target and single (R) click to open the dialog box shown below. Select the **Common View [SPACE]** option. **NOTE:** that pressing the **Space Bar** on the keyboard will toggle between the **Free Rotate** target and the **Common View** box.

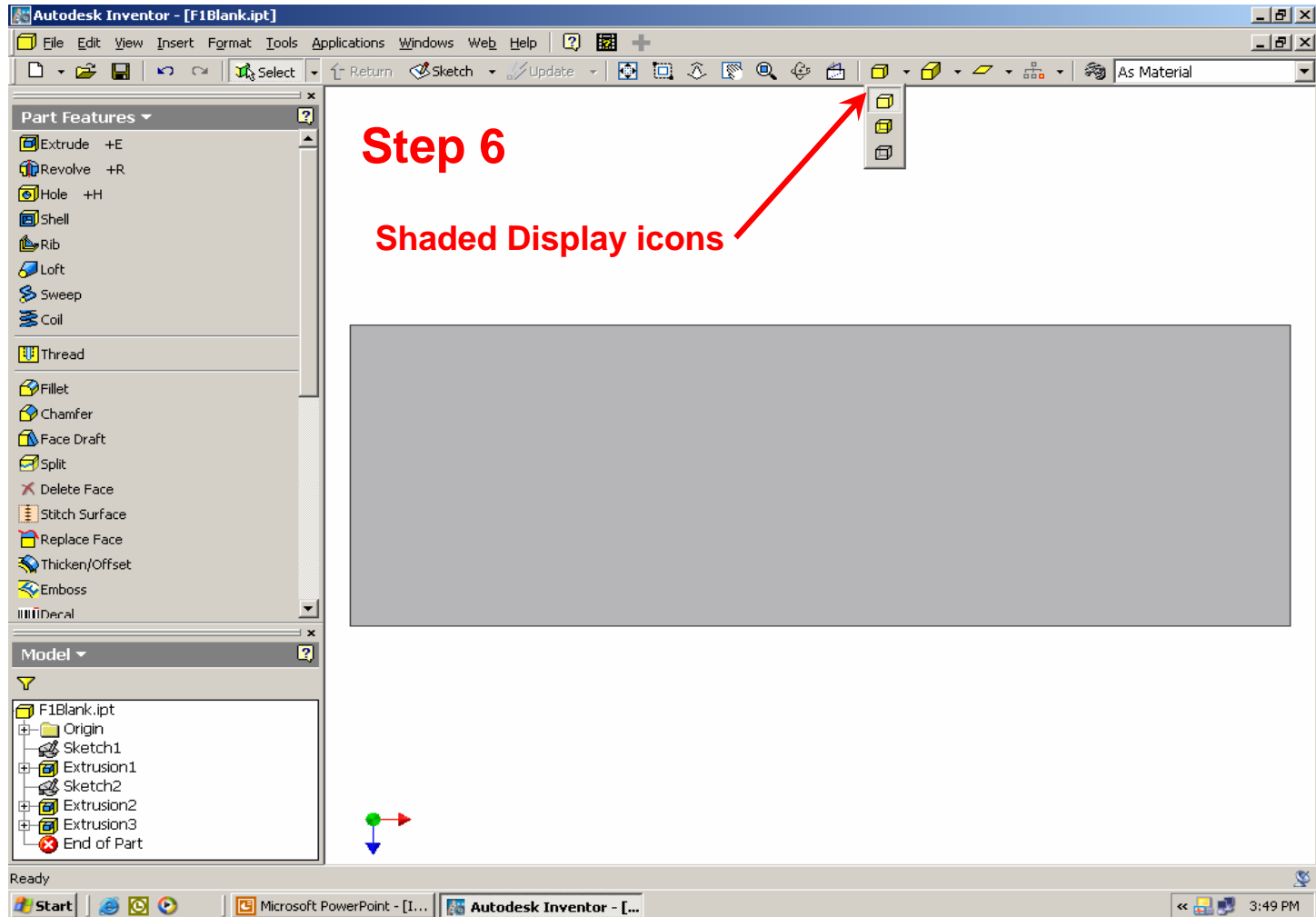




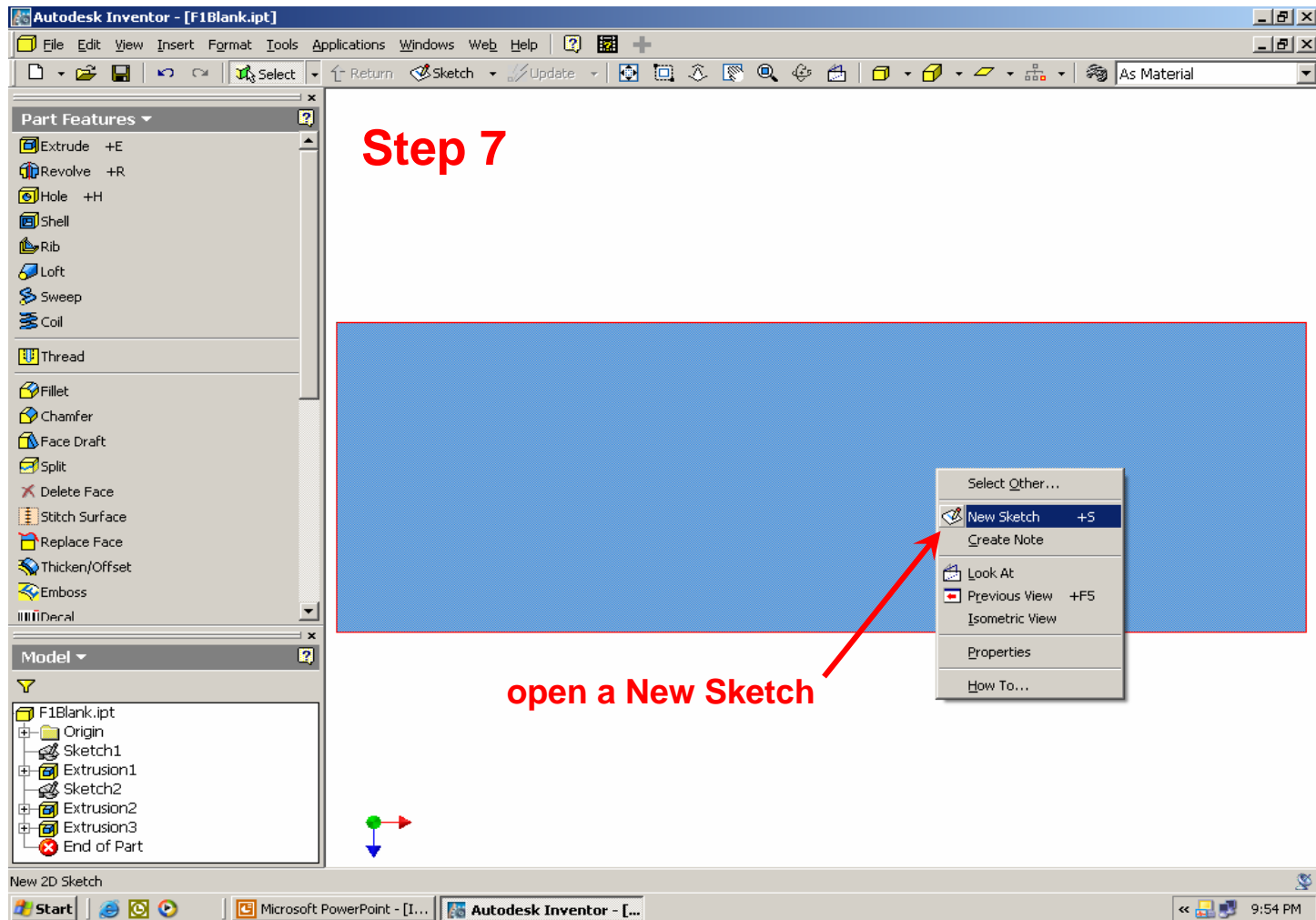
**Step 5.** – When the **Common View** command is selected, the **Free Rotate** target disappears and the **Common View** box is displayed. The green arrows indicate the direction from which your model will be viewed. As you move the cursor over the green arrows with the mouse they highlight. When the arrows are highlighted and selected with a (L) click, the view changes according to the direction the selected arrow was pointing. Practice with the **Common View** rotation command until you feel comfortable using it.



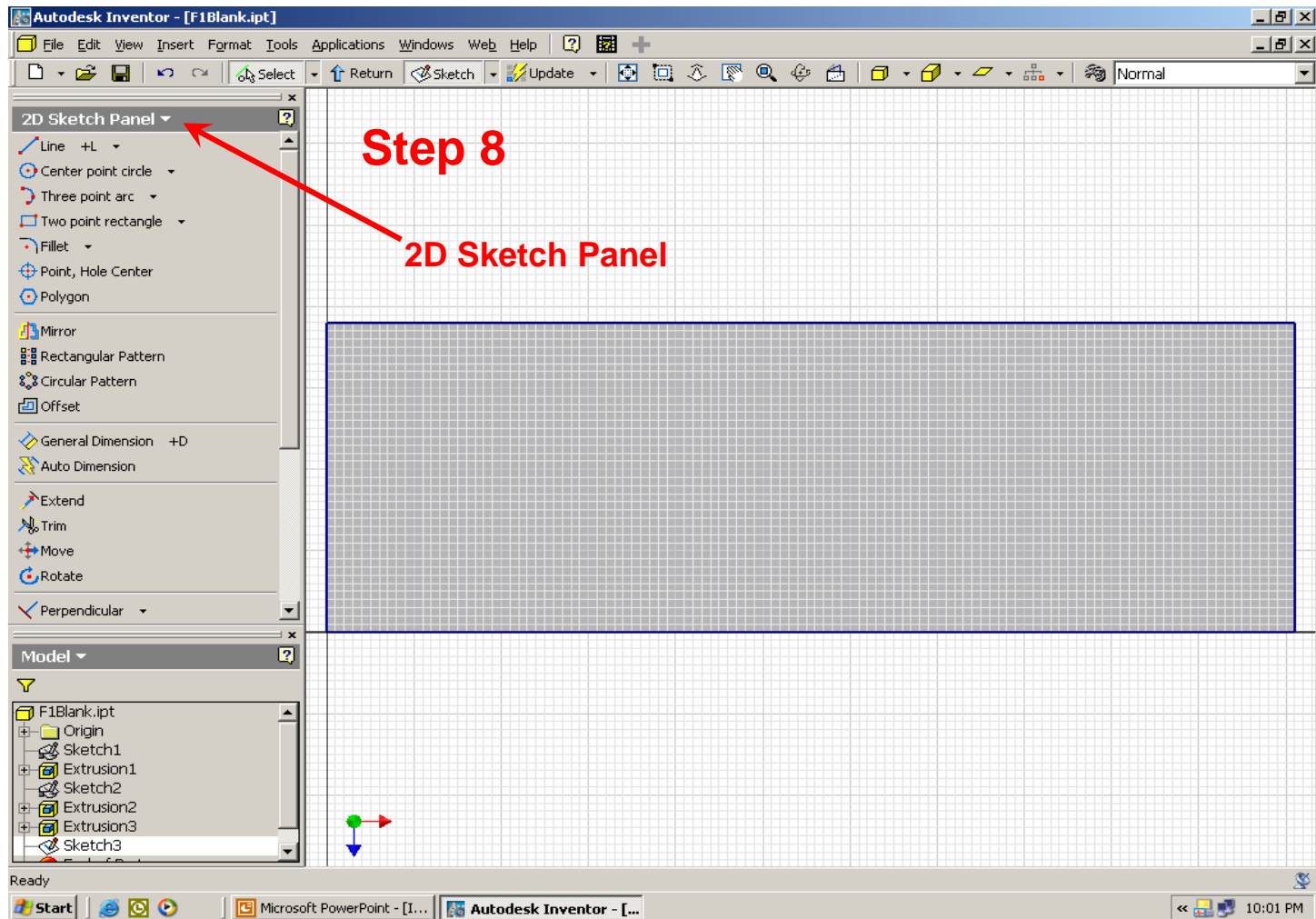
**Step 6.** – Using a combination of **Free Rotate** and **Common View**, Rotate the F1 Blank until you are looking directly at the TOP surface (as shown below) with the hole for the CO2 cartridge to the left. Check to be sure the hole is to the left by toggling the display between **Shaded Display** and **Wireframe Display** with the **Shaded Display** icons.



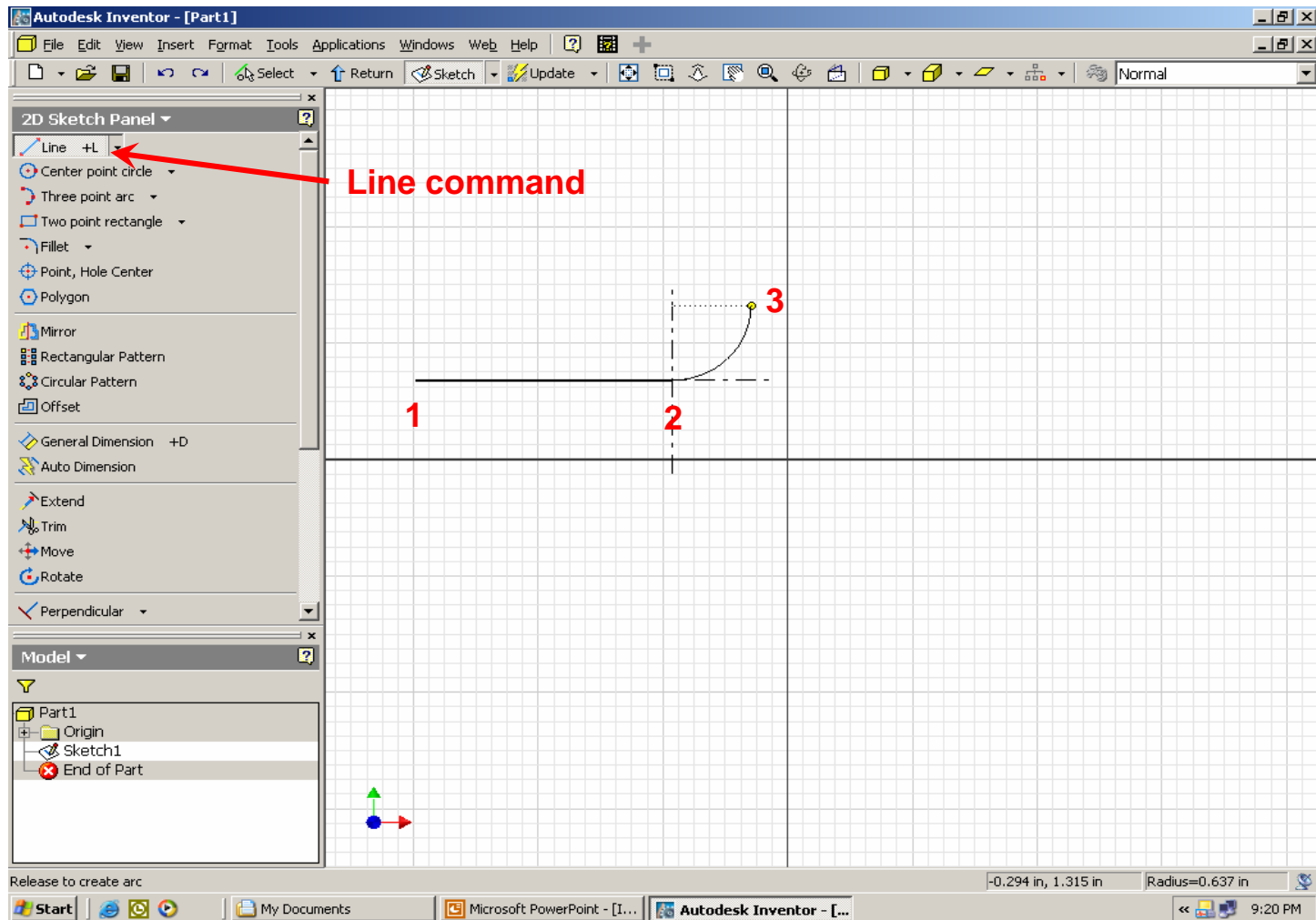
**Step 7.** – Move the cursor over the outline of the car blank and (R) click. The outline of the blank will turn blue and a pop-up dialog box opens. (L) click on **New Sketch**.



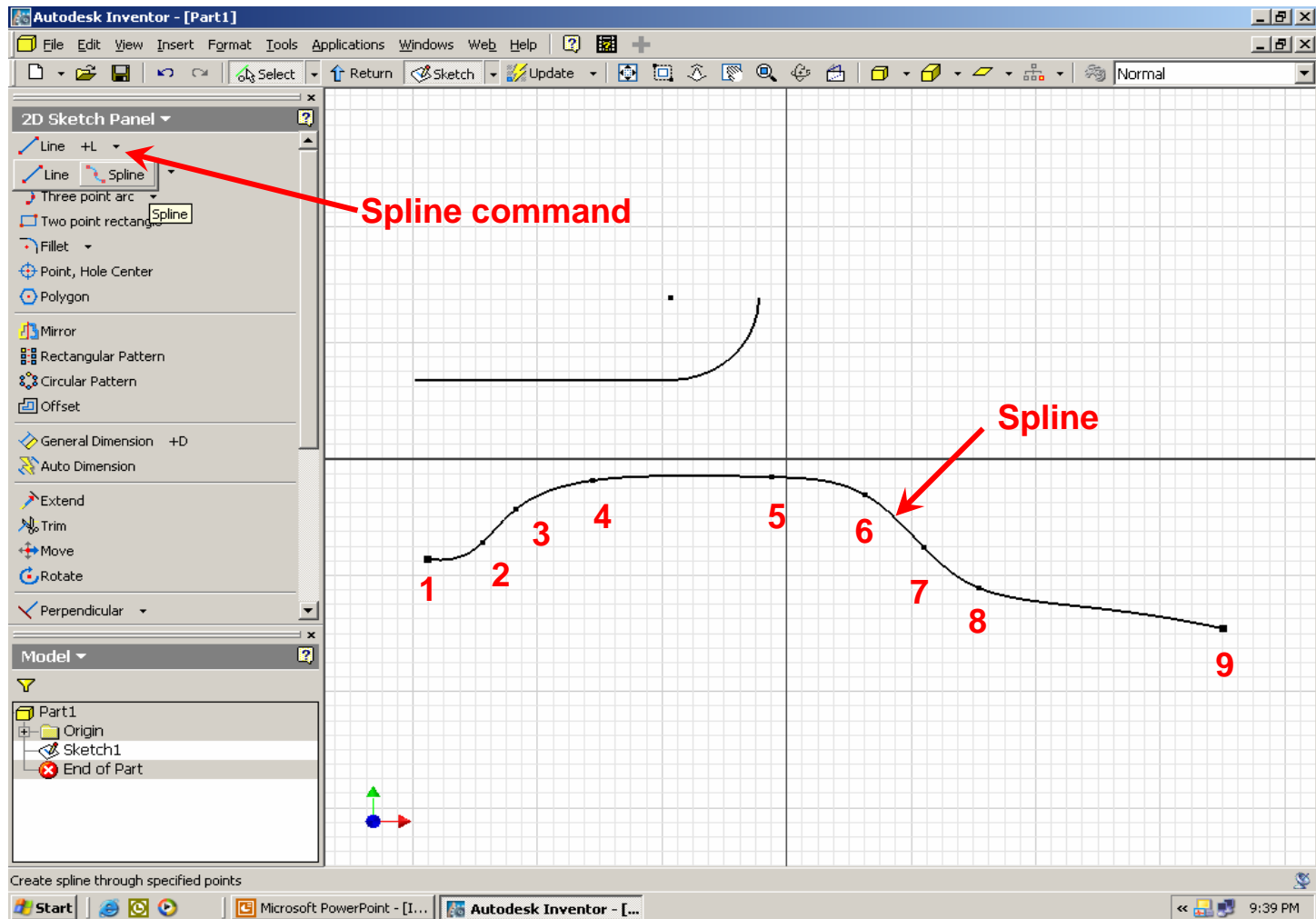
**Step 8.** – Notice that a grid appears over the Graphics Window and the Command Panel changes from the **Part Features Panel** to the **2D Sketch Panel**. You are now ready to begin sketching the top profile of your F1 car design.



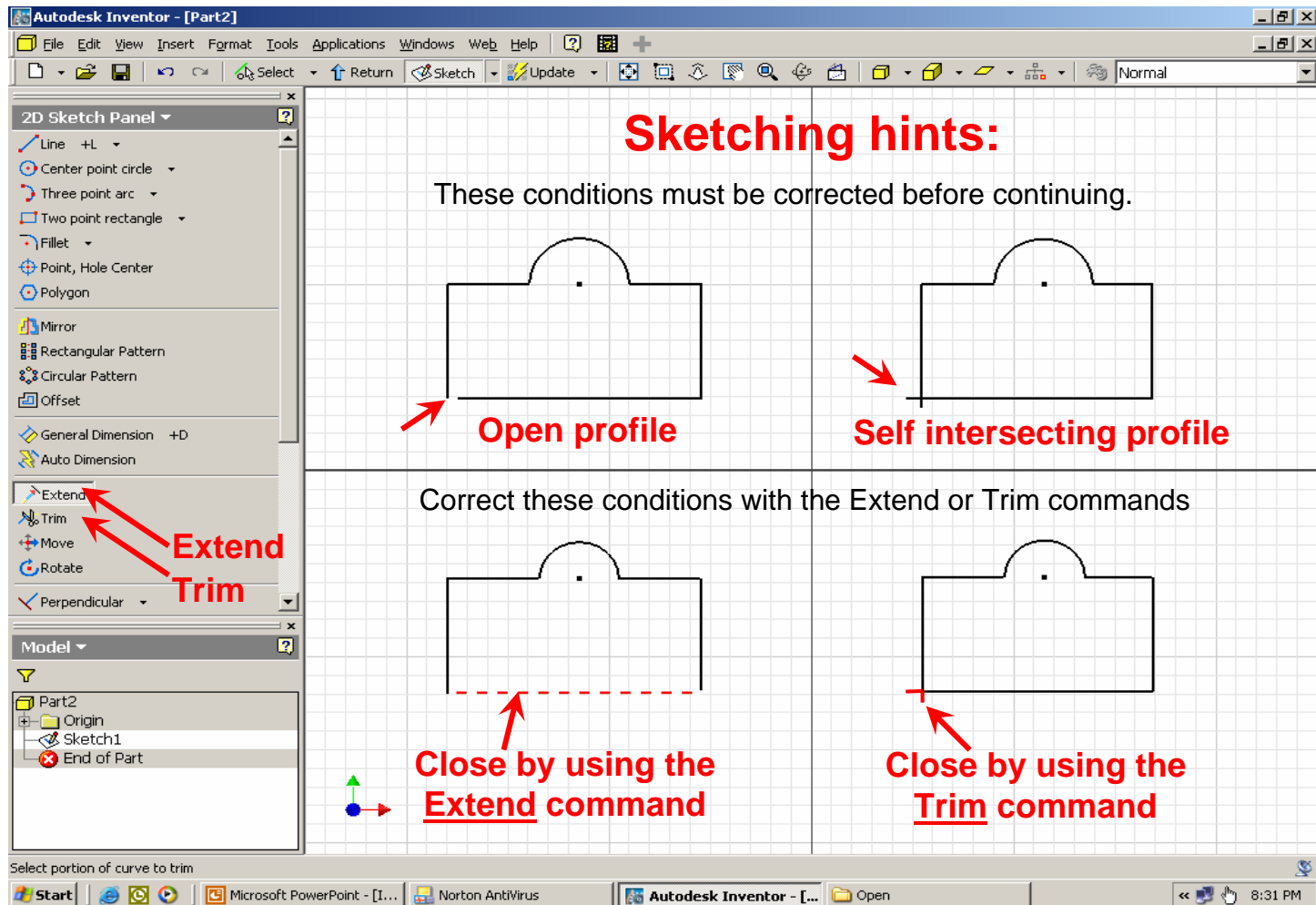
**NOTE:** – When sketching with the **Line** command, a (L) click will start a line (point 1 below); move the mouse and (L) click again to locate the end point of a line (point 2 below). To draw arcs while in the **Line** command, (L) click and hold the mouse button down (point 2 below) and move the mouse to trace an arc. When you have traced the desired arc, release the left mouse button (point 3 below). When finished with the line command, (R) click and select **Done** from the pop-up menu.



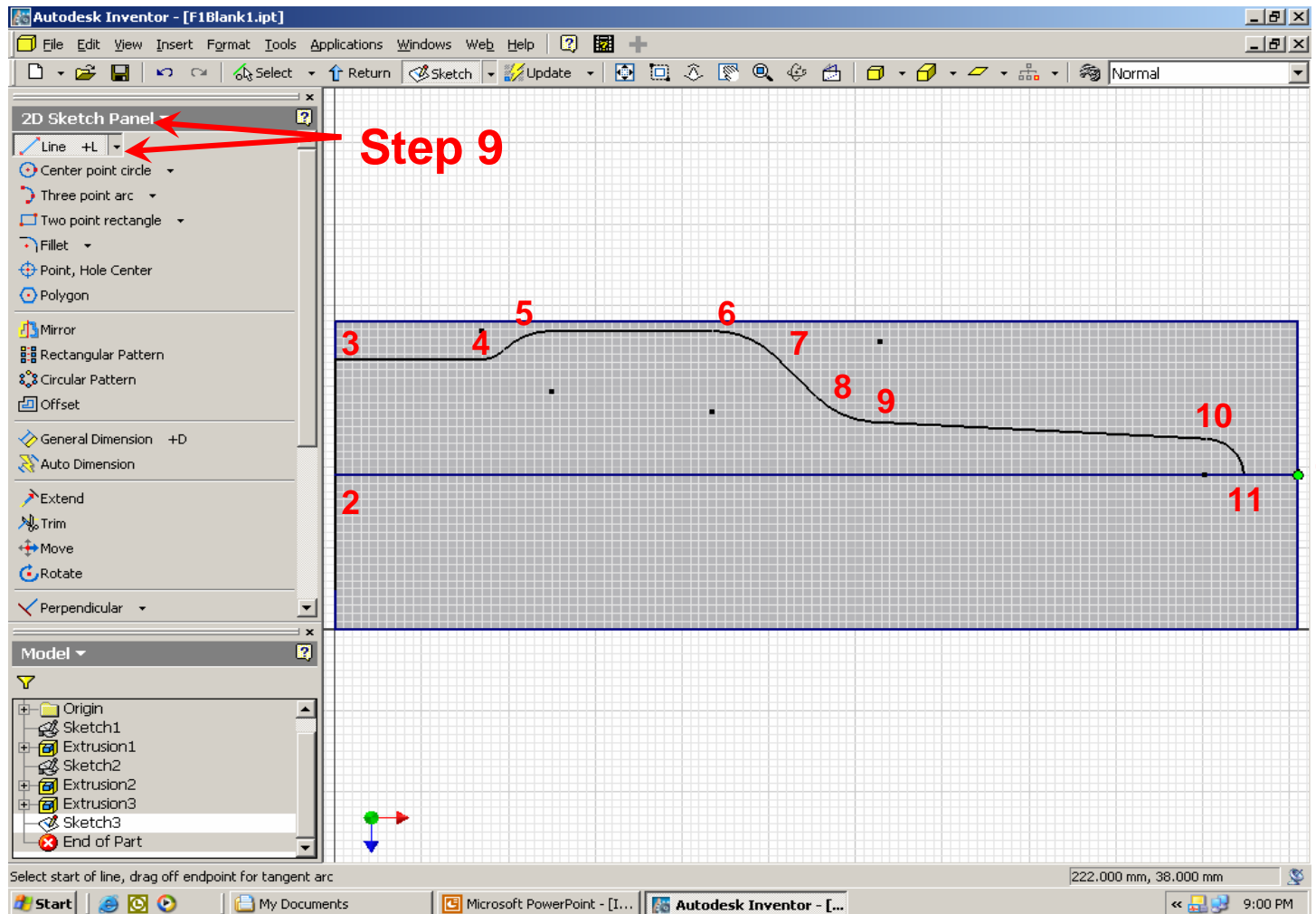
**Design Alternative:** – You can also sketch with splines. Splines are free-formed curves. To use the **Spline** command, (L) click on the down arrow beside the **Line** command then (L) click on **Spline**. When you have placed the last point of the spline, (R) click and select **Continue** from the pop-up menu with a (L) click. Practice with the **Line** and **Spline** commands before going to the next step.



**Sketching hint:** When sketching, be sure that the shape you are sketching forms a **closed profile**. If there are **open profiles** or **self intersecting profiles** in your sketch, **they will not extrude!** These conditions can be corrected by using the **Extend** command and/or the **Trim** command.

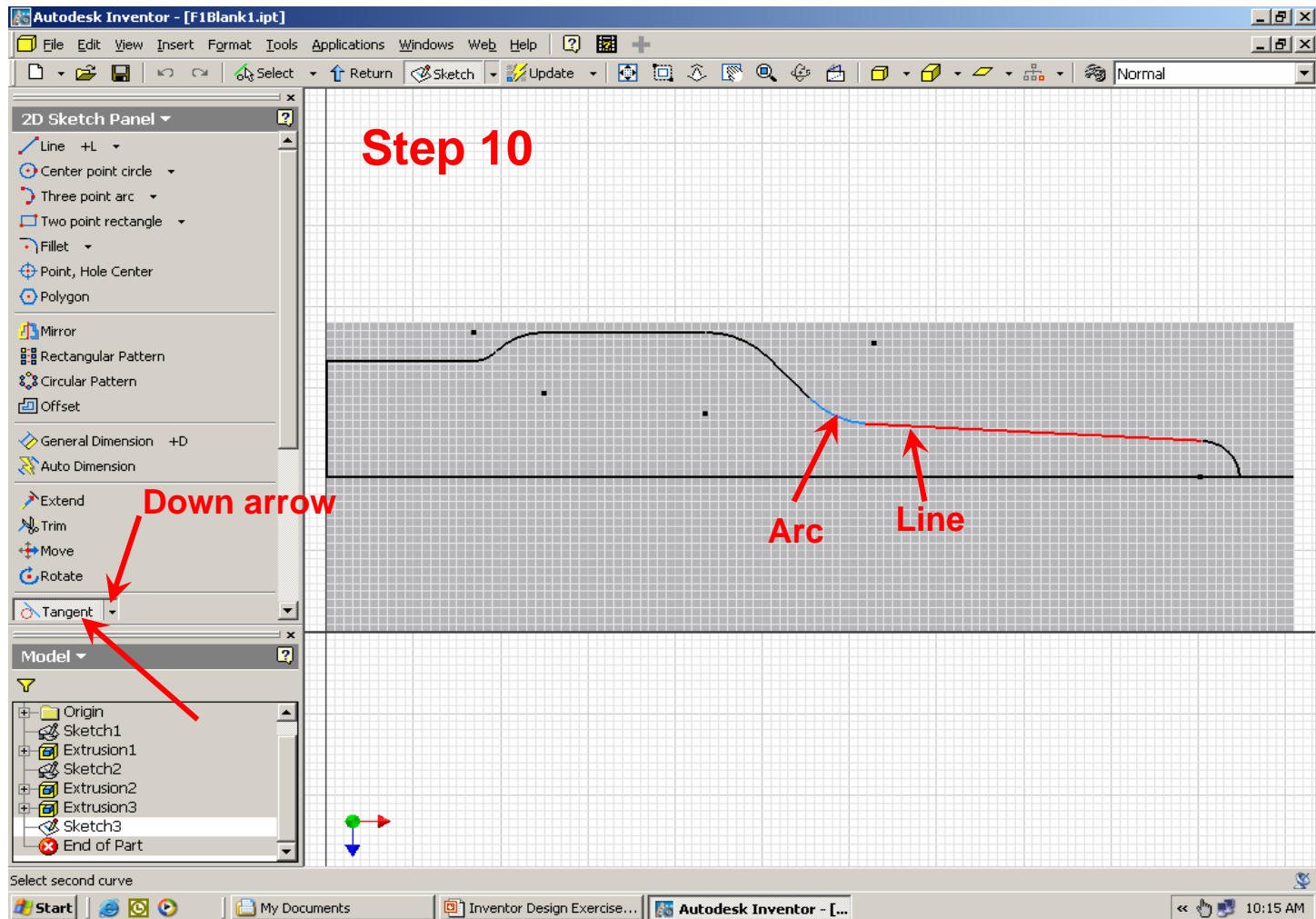


**Step 9.** - From the **2D Sketch Panel**, select the **Line** command with a single (L) click. Draw the rough sketch shown below. Notice that the cursor turns to a yellow dot, and as you move the dot along the outline of the sketch, it will turn green at the end points and mid-points of the lines. Once a mid-point or end point has been found, construction lines will project from the yellow dot as you move the cursor horizontally or vertically. Don't worry about exact dimensions. When placing your points, follow the order below and be sure that your sketch forms a closed profile. **REMEMBER - If the profile is not closed it will not extrude.**

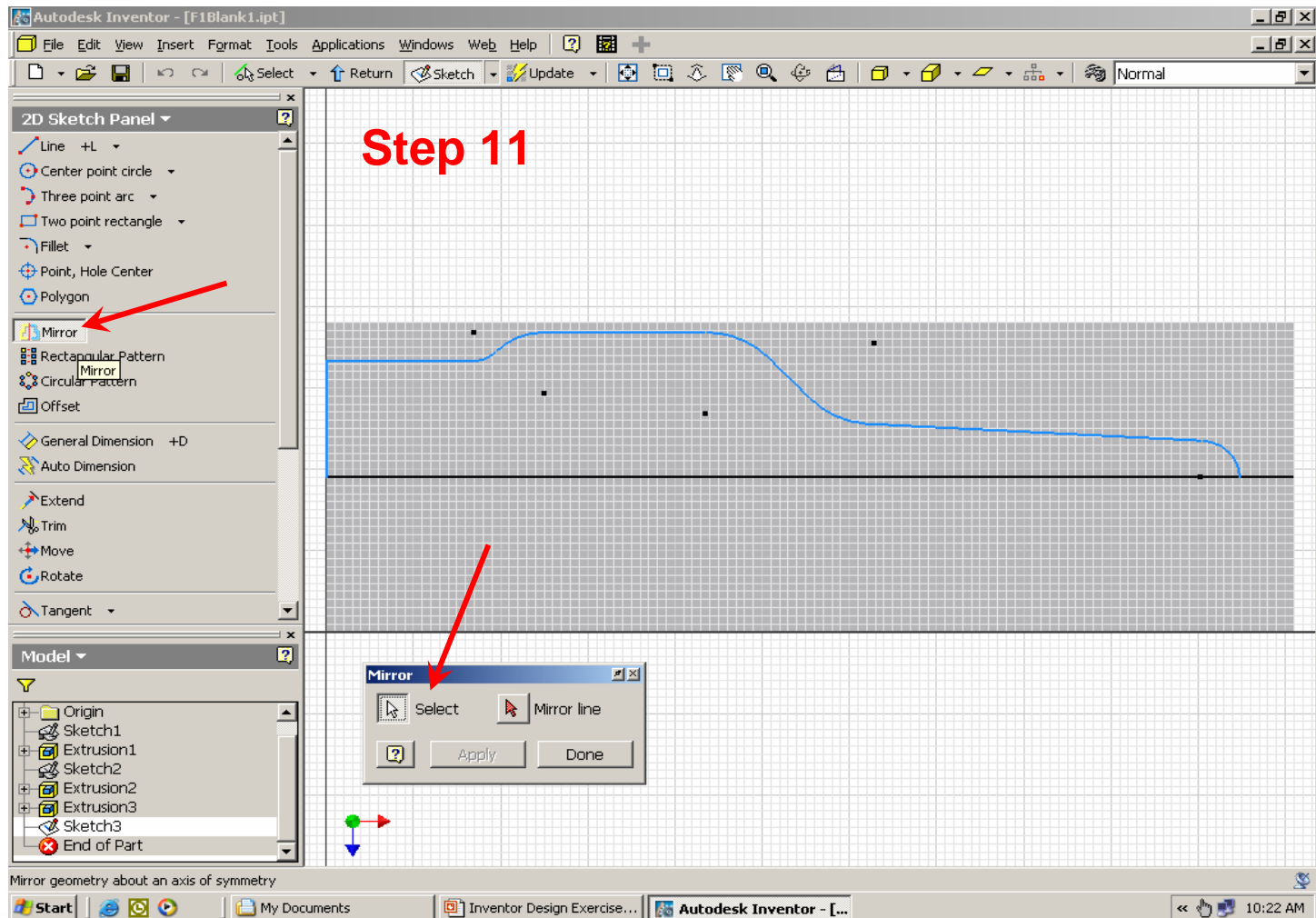




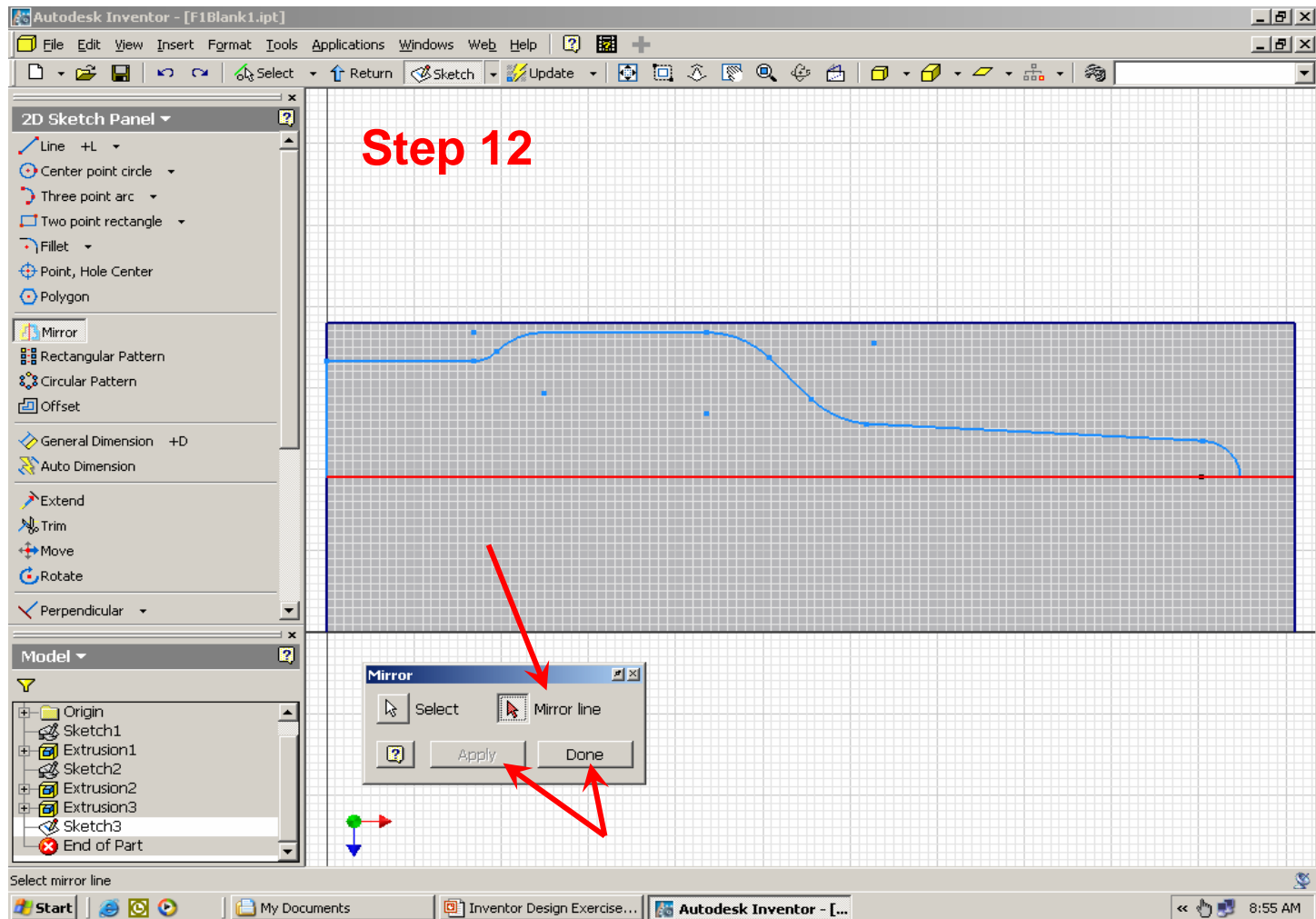
**Step 10.** – When your sketch is finished, (L) click on the black down arrow beside the Constraint command and (L) click on **Tangent**. Apply tangent constraints between the lines and arcs of your sketch by first (L) clicking on a line, then on the adjacent arc. This will insure that you have a smooth transition from the lines to the arcs.



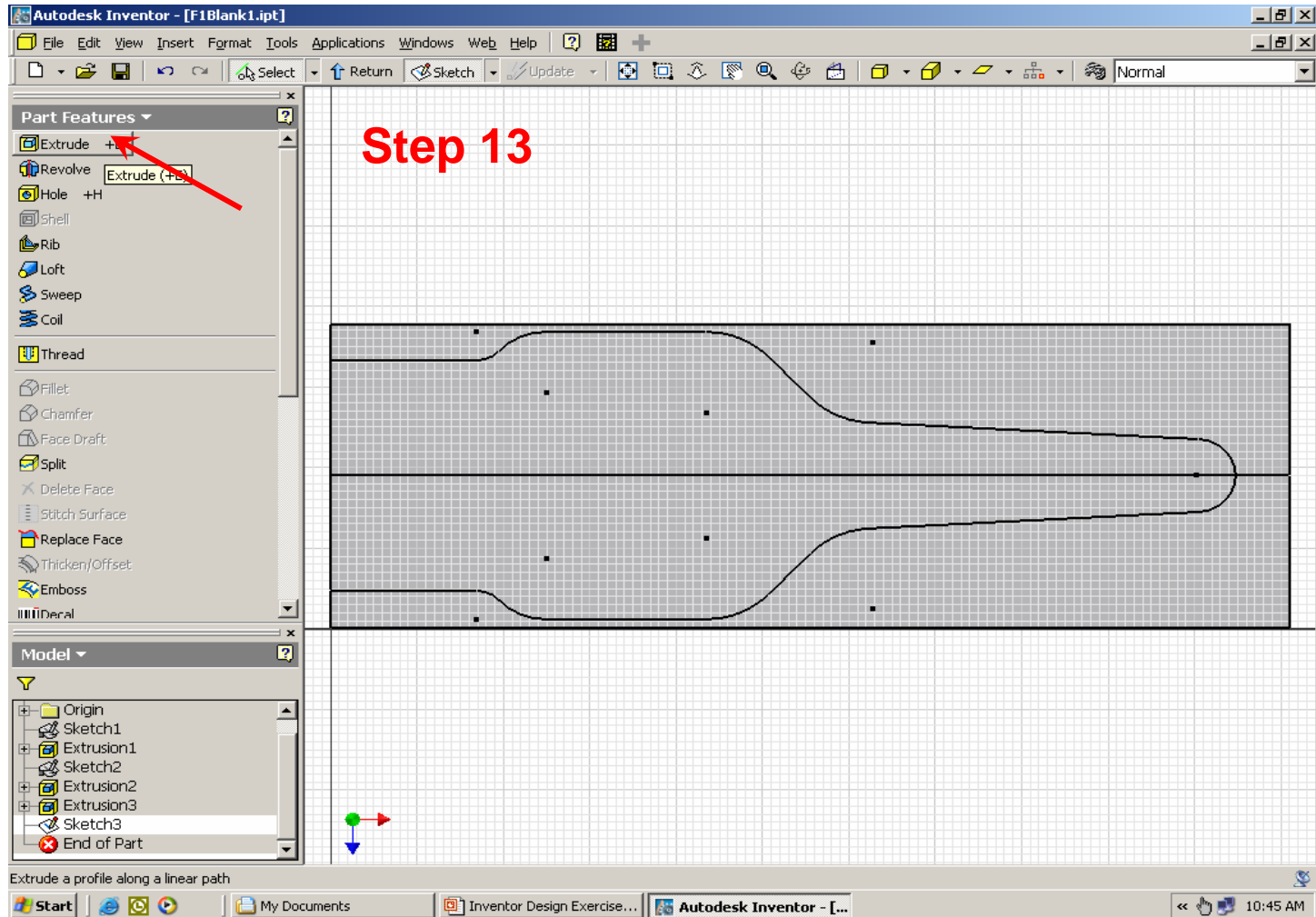
**Step 11.** – (L) click on the **Mirror** command. Select each of the lines in your sketch by (L) clicking on them individually (holding the Shift key down will allow you to select multiple lines) or by windowing the sketch. Note that as you select the lines they highlight. **DO NOT select the horizontal center line.** Be careful to select ALL of the lines in your sketch (in blue below) so that, when they are mirrored, they form a **CLOSED PROFILE**.



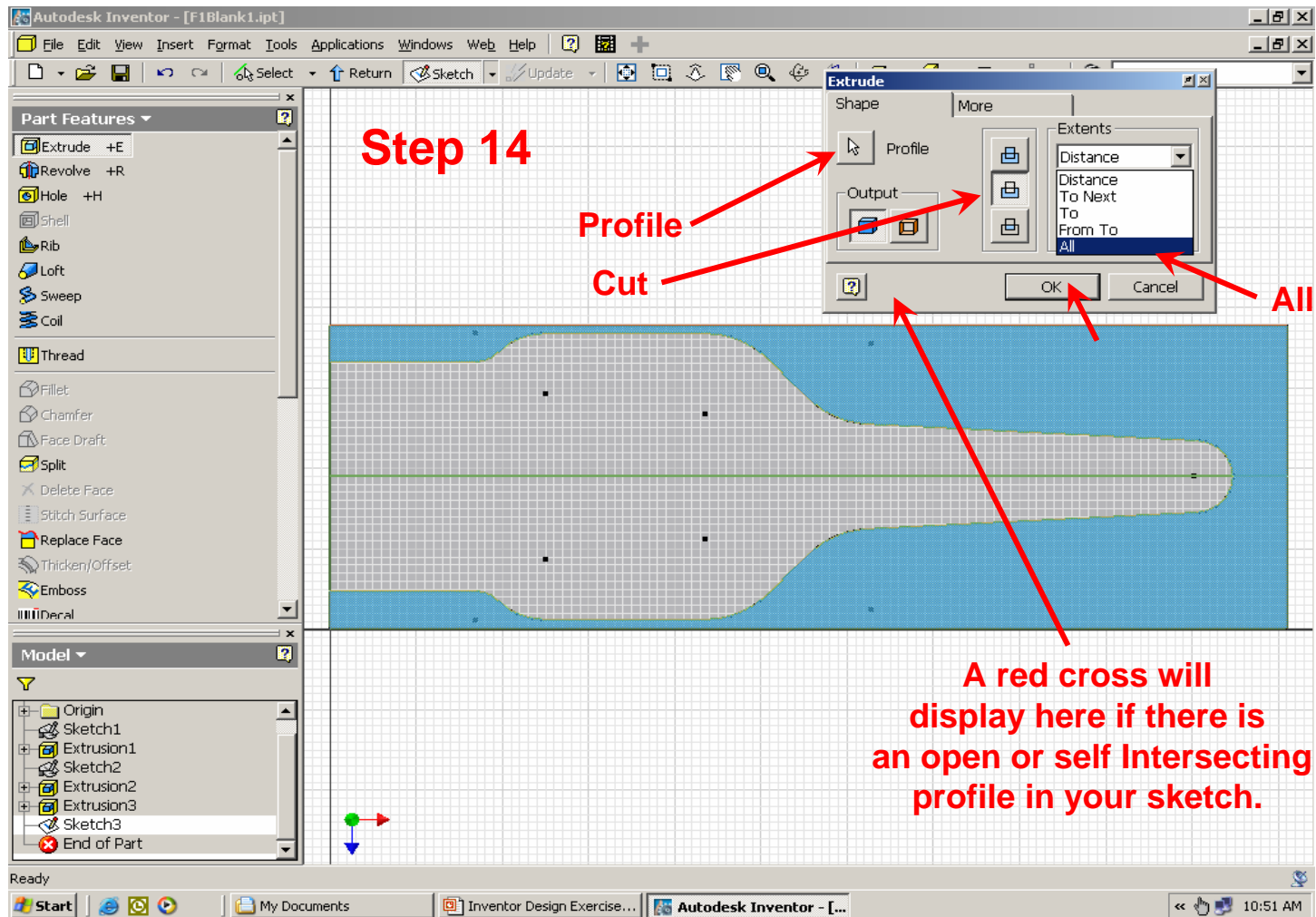
**Step 12.** – After all of the sketch lines have been selected (highlighted), (L) click on the **Mirror line** button and select the horizontal line (highlighted in red below). (L) click on **Apply** to complete the mirror image, then click on **Done** to end the **Mirror** command.



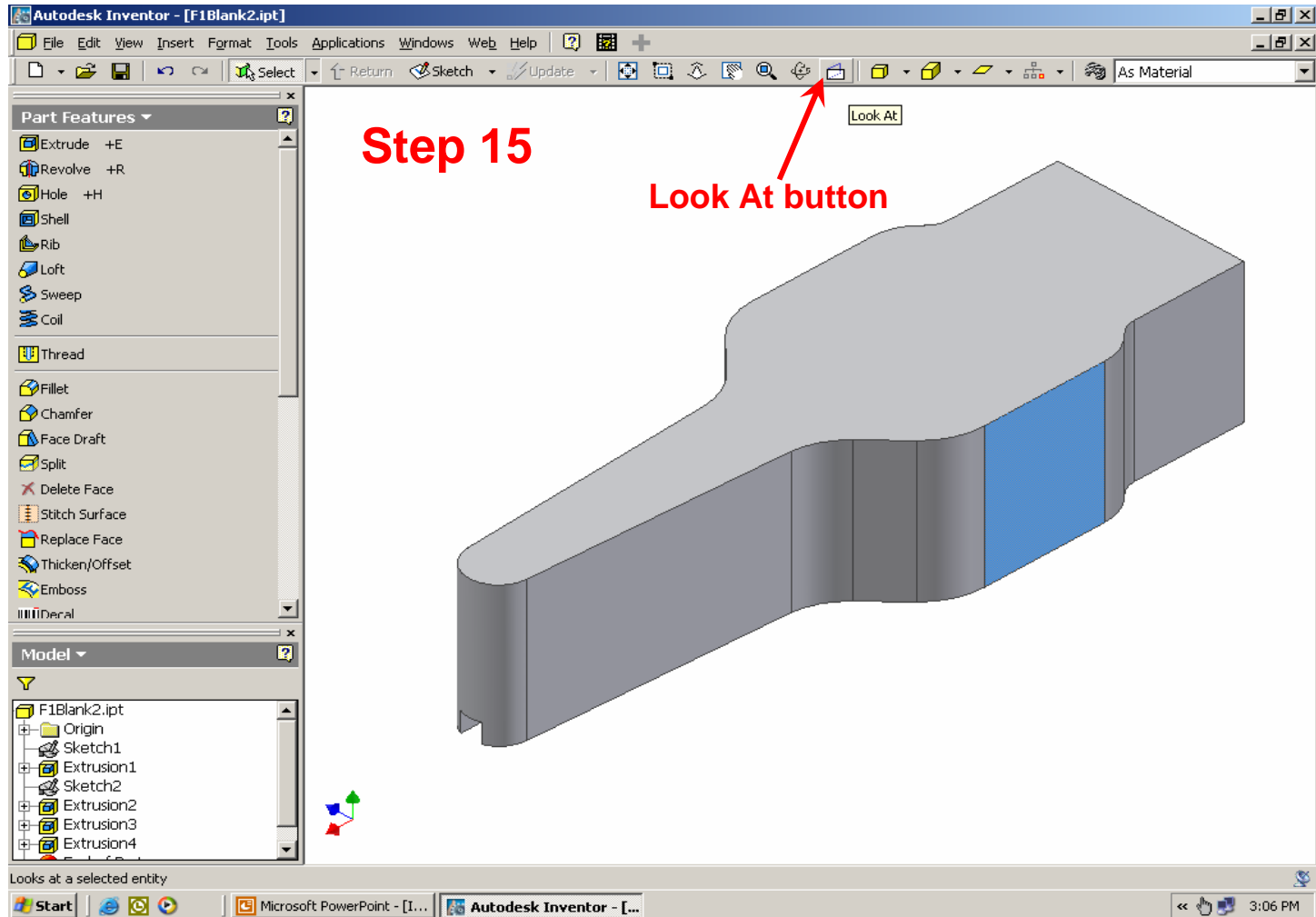
**Step 13.** – (L) click on the **2D Sketch Panel** and switch to the **Part Features** panel. Then (L) click on **Extrude** to open the Extrude dialog box.



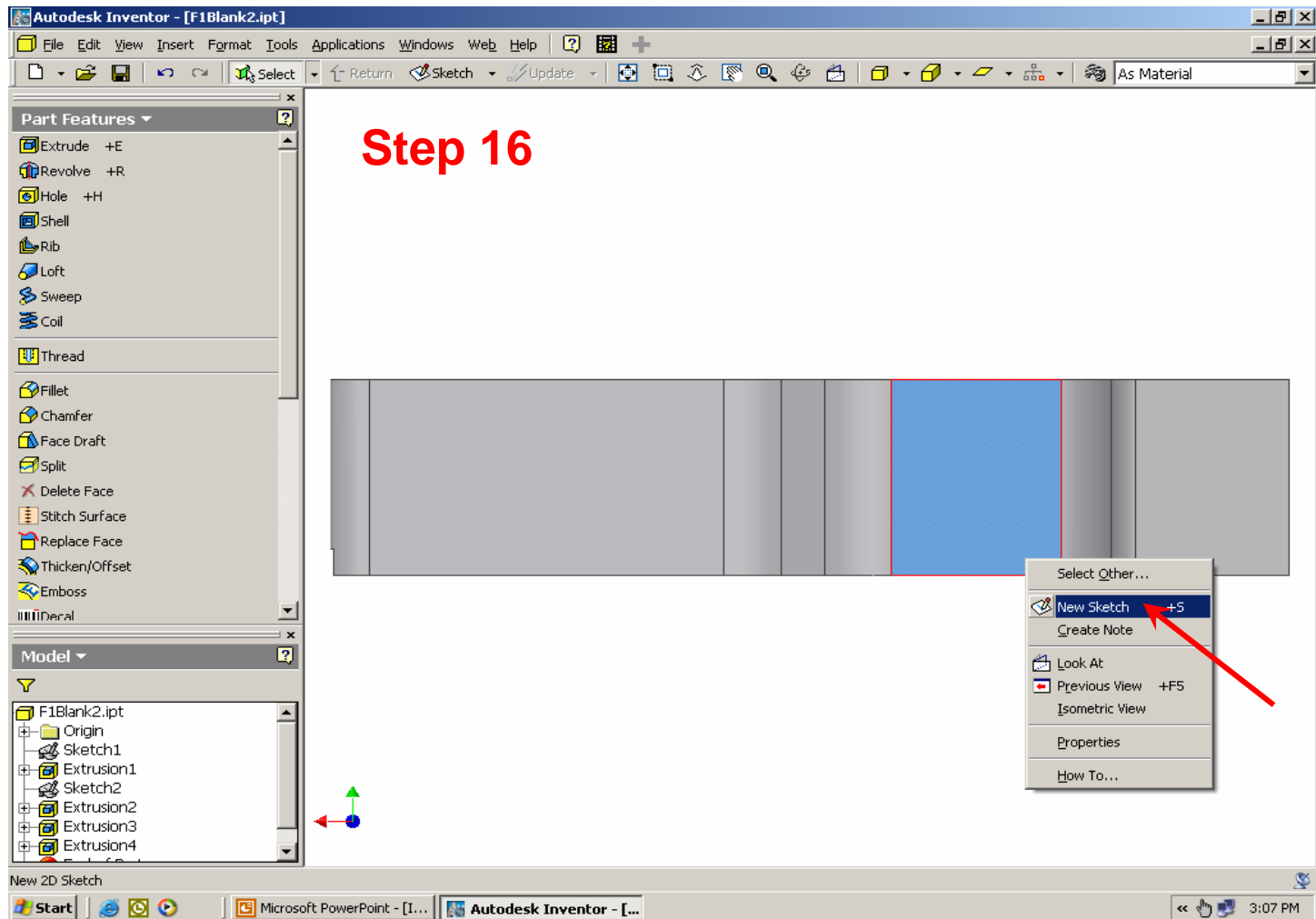
**Step 14.** – First select the **Profile** by moving your cursor over the sketch until the top half of the profile highlights and (L) clicking. Hold the **Shift key**, move the cursor over the bottom half of the sketch and (L) click when the bottom half of the profile highlights. **NOTE:** - if you are unable to select both profiles (highlighted in blue below), and a red cross is displayed in the **Extrude** dialog box, this indicates that there is an open profile in your sketch – return to the sketch mode and correct the problem. Next (L) click on the **Cut** button and set the **Extents** Distance to **All**. To finish making the cut extrusion, (L) click on **OK**.



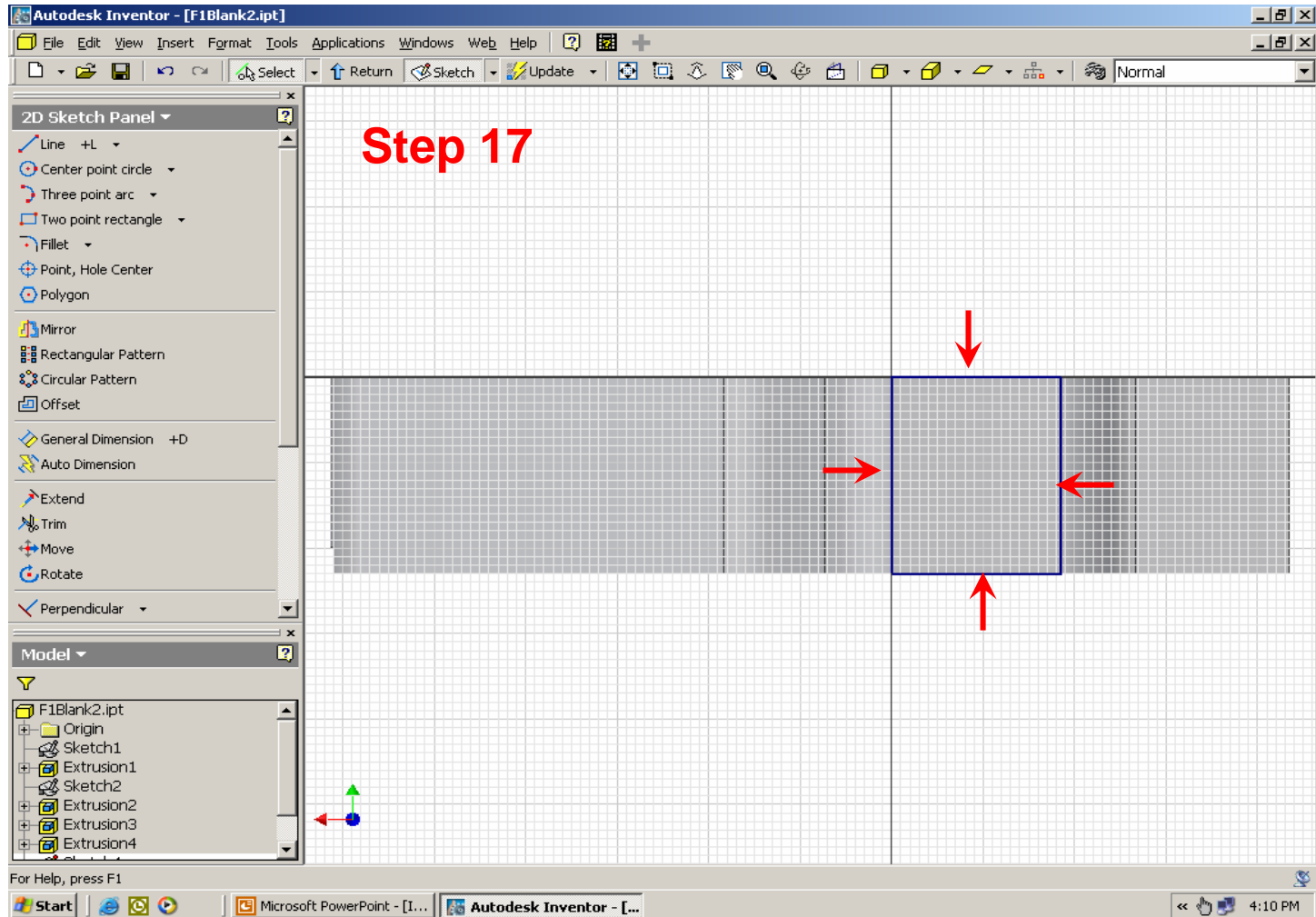
**Step 15.** – Use the **Rotate** commands to examine your model. (L) click on the face highlighted in blue below then (L) click on the **Look At** button on the Standard Toolbar.



**Step 16.** – You are now ready to draw the side profile of your F1 car. (R) click on the highlighted surface (below) and (L) click on **New Sketch**.

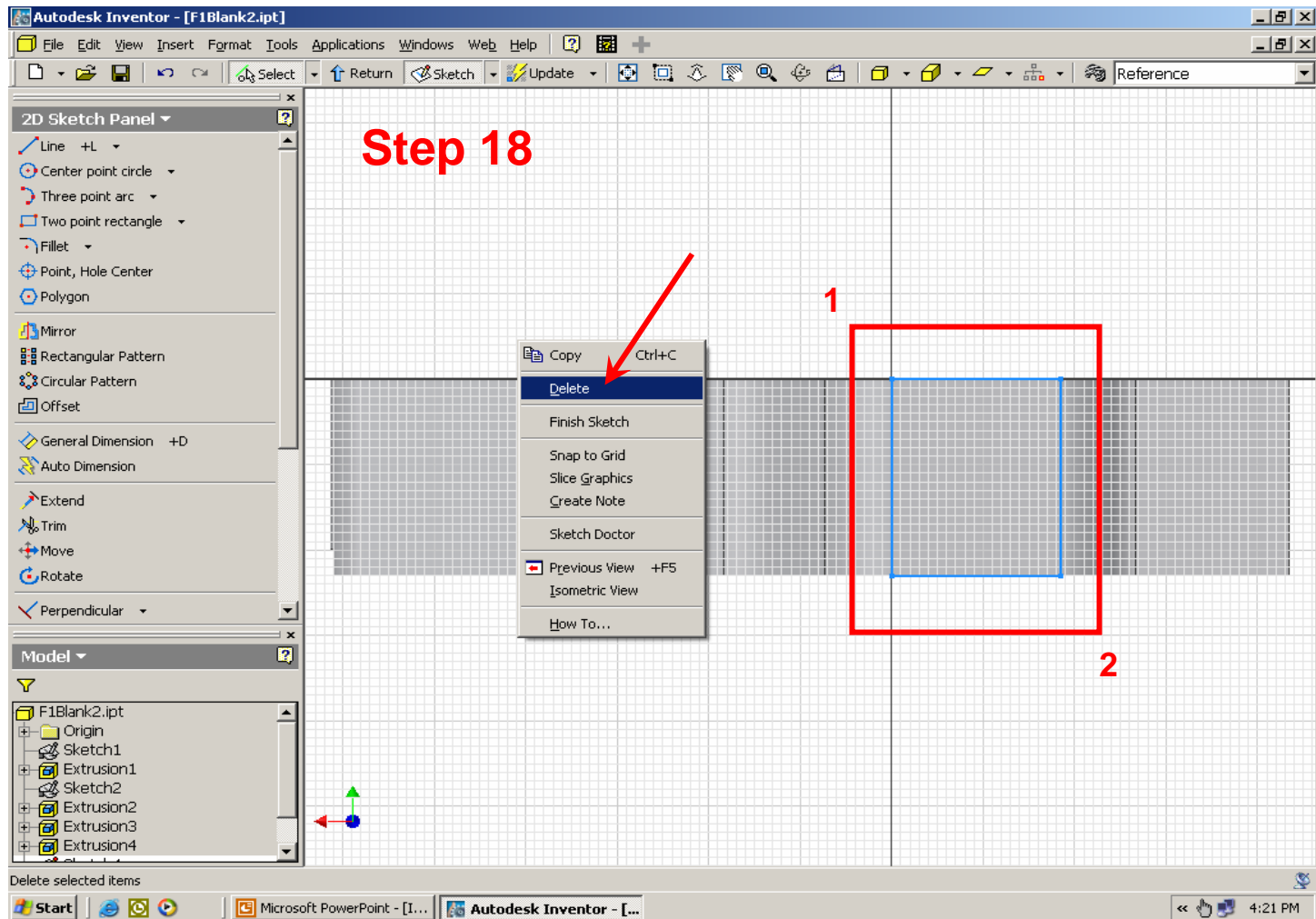


**Step 17.** – Notice that a new sketch has been opened and 4 lines from our model have been automatically projected onto our sketch plane. For this step, these 4 lines must be deleted.

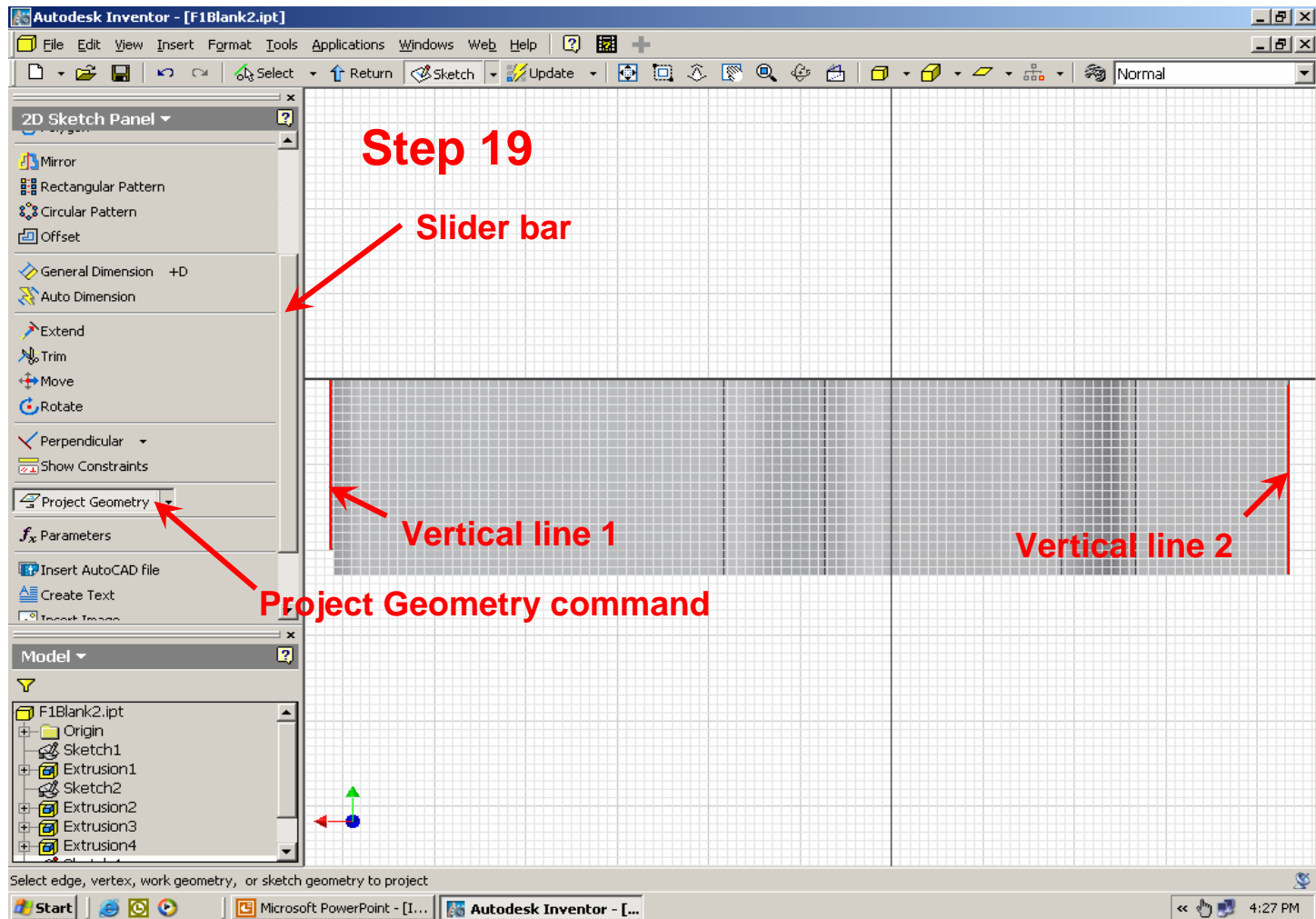




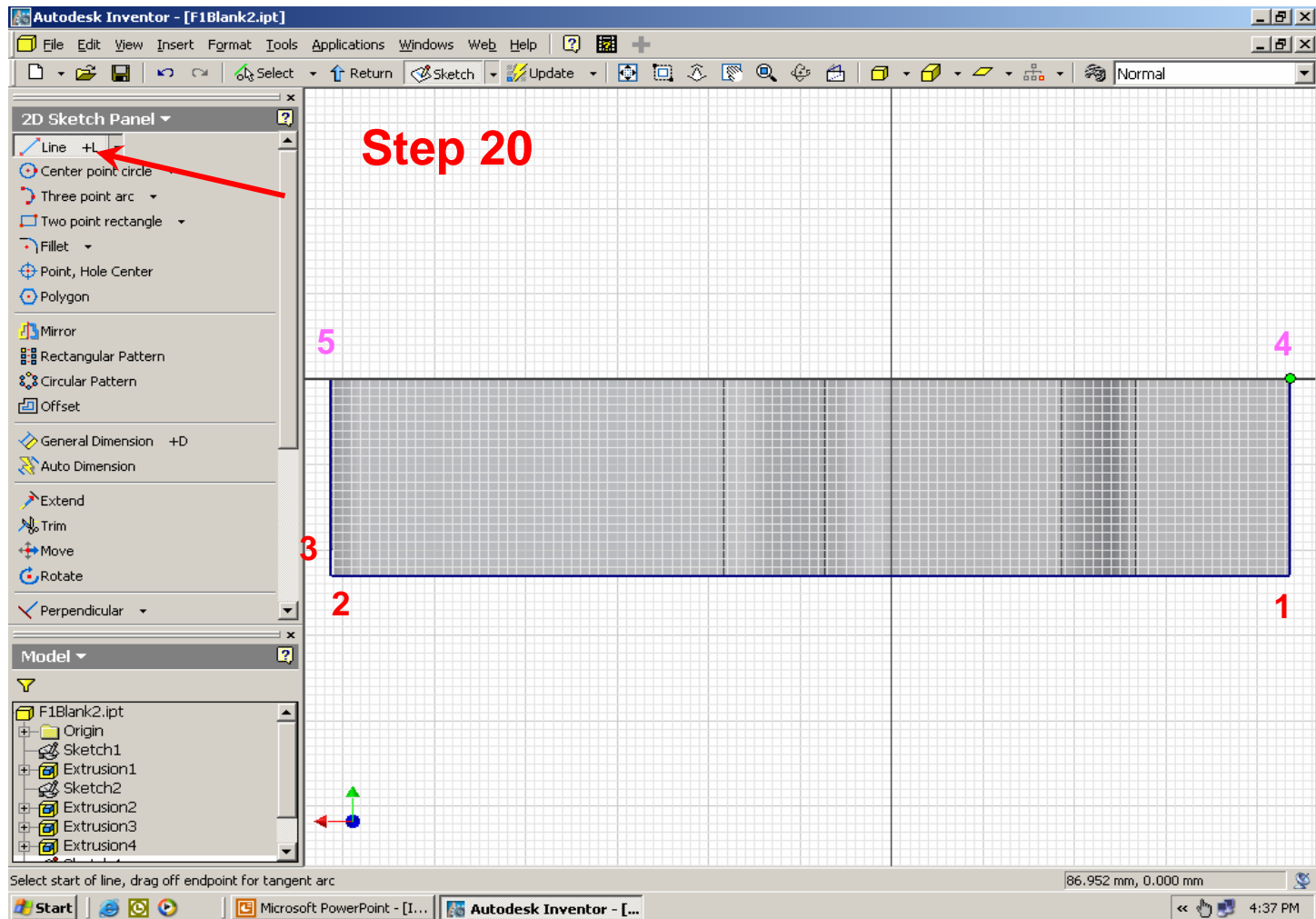
**Step 18.** – Window the 4 lines by (L) clicking at 1 (below) and, while holding the (L) mouse button, drag the mouse to 2 and release the button. NOTICE that all 4 lines will highlight. (R) click and select **Delete** from the pop-up window.



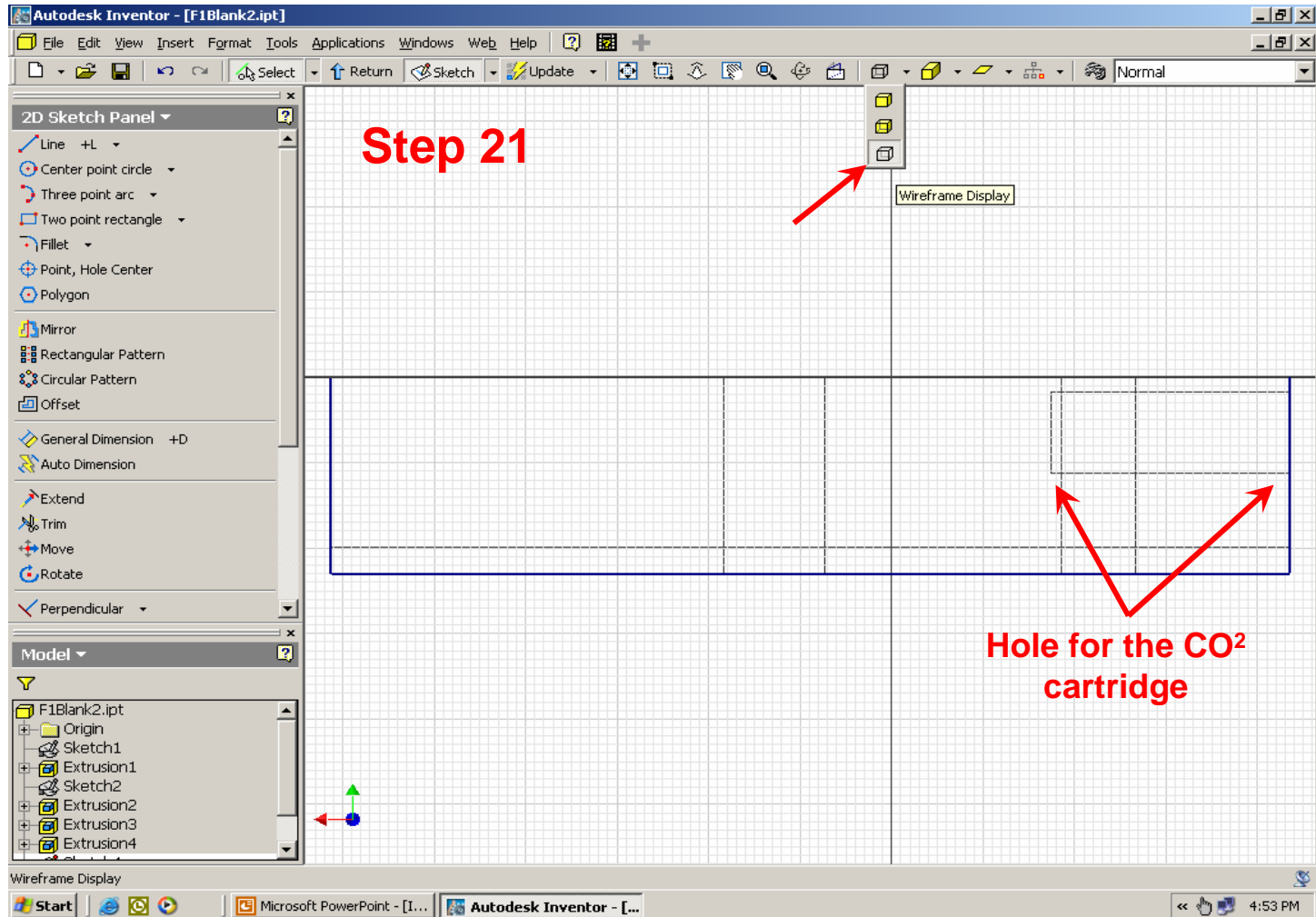
**Step 19.** – Use the slider bar to scroll down to the **Project Geometry** command and (L) click. Select the two vertical lines at either end of the car body (below). Notice that after the lines have been selected they will darken indicating that they have been projected onto the sketch plane.



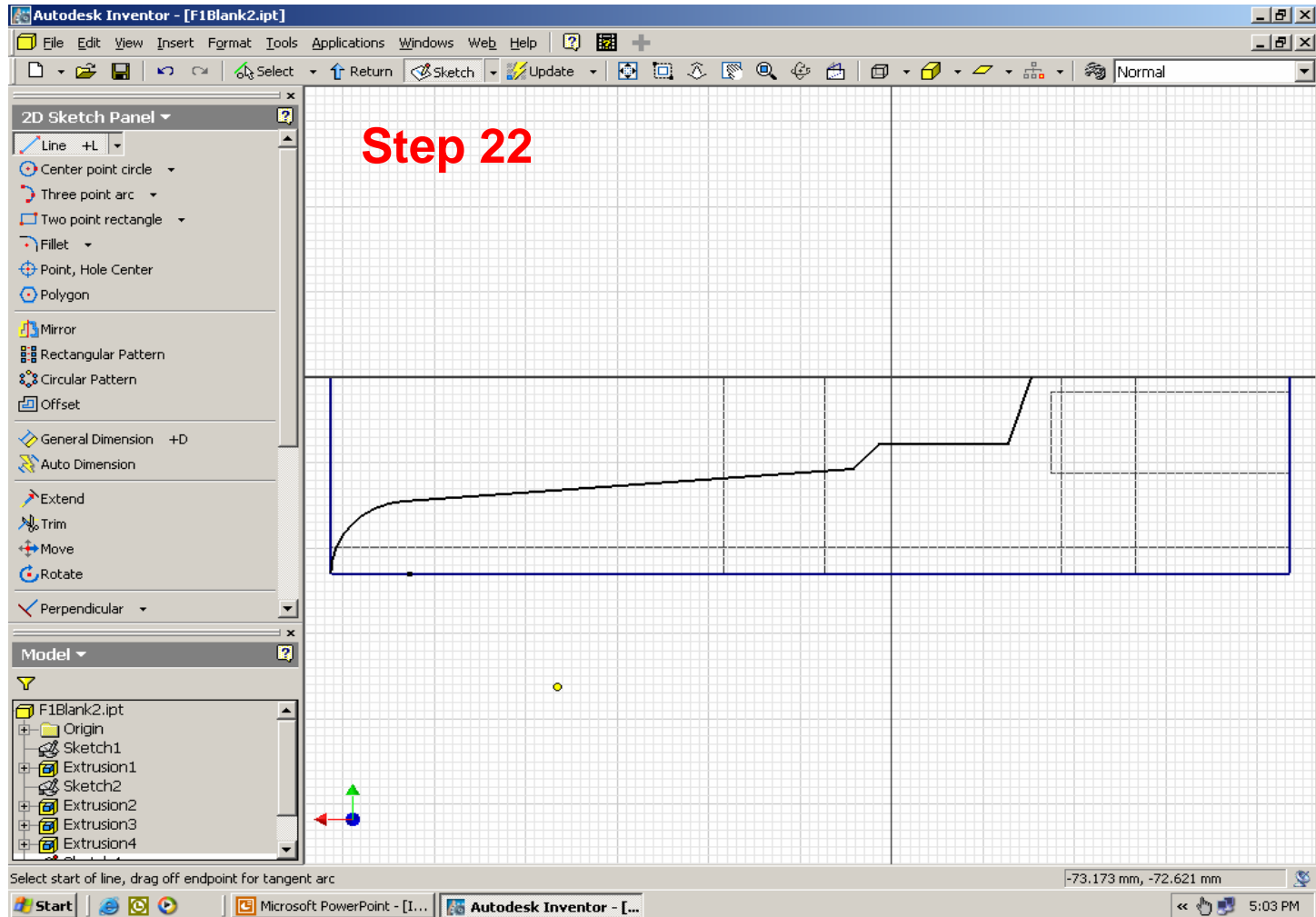
**Step 20.** – Select the **Line** command and draw line 1-2-3 and then line 4-5. Be careful to lock on to the ends of the vertical lines that were projected in the last step so that they form a **CLOSED PROFILE**. The cursor will turn from a yellow dot to a green dot when you have locked on to the end points of the lines.



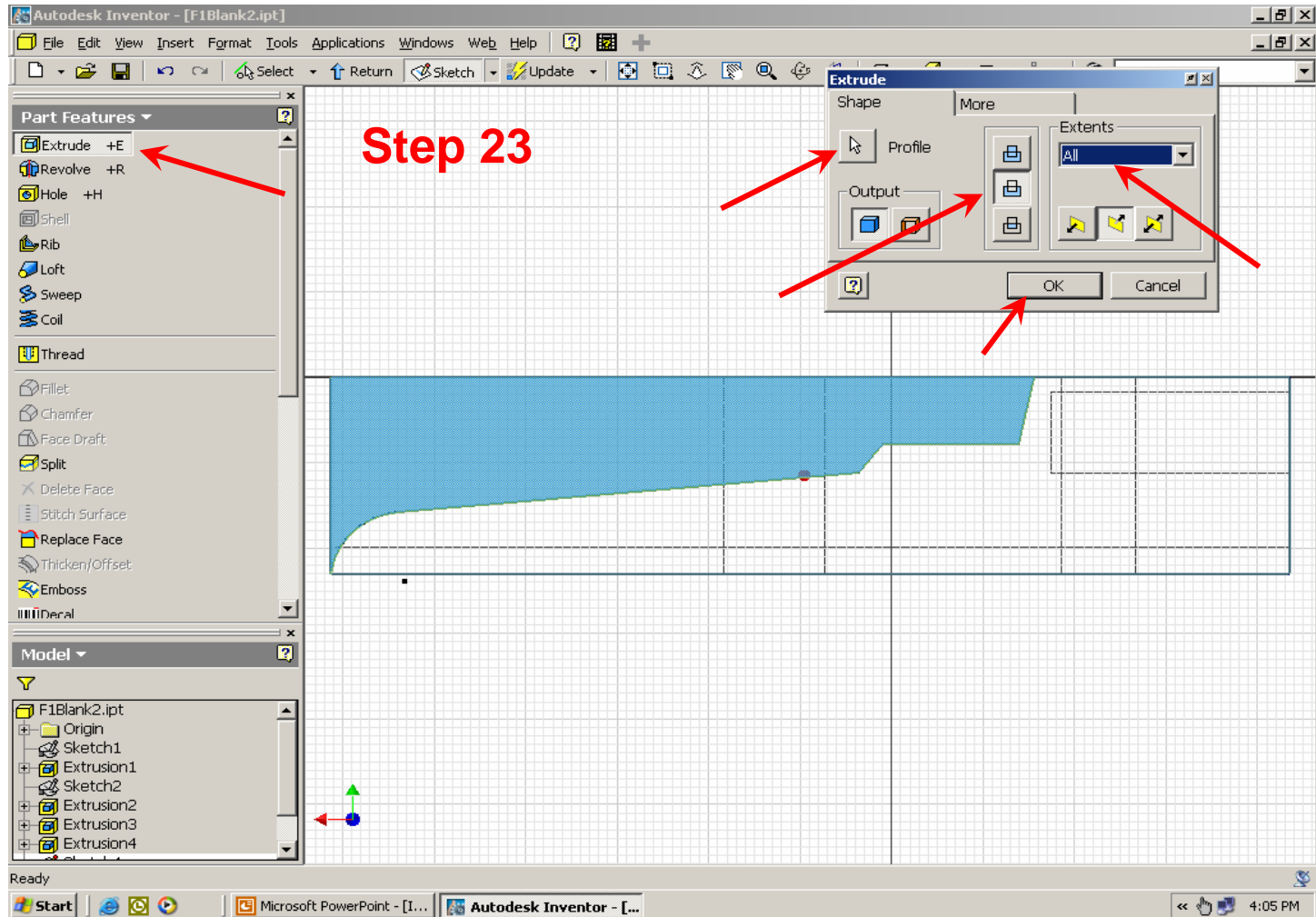
**Step 21.** – Switch to the **Wireframe Display** mode from the Standard Toolbar as shown below. This will make it easier to sketch the side profile. **Be careful not to sketch over top of the hole for the CO<sup>2</sup> cartridge.**



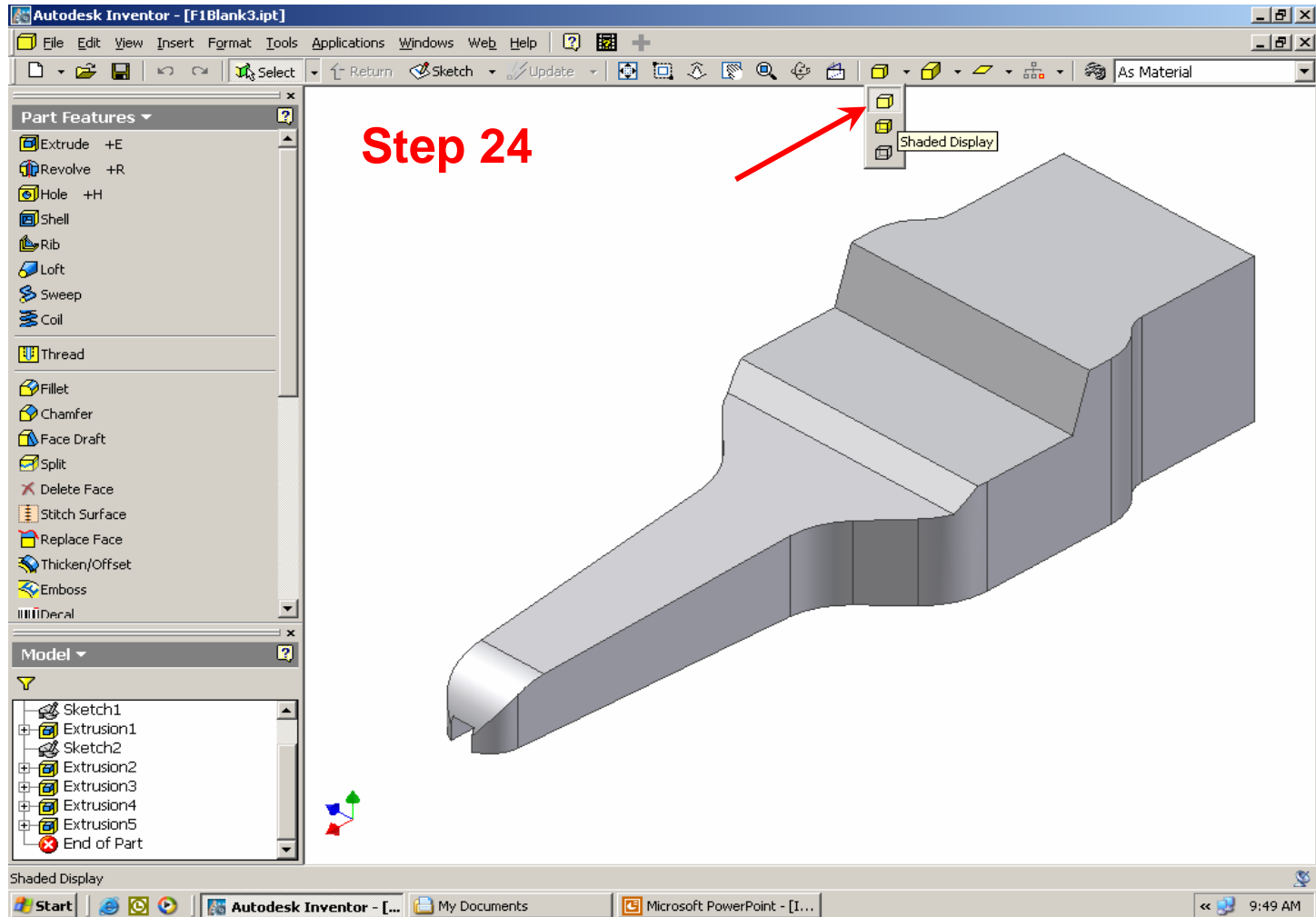
**Step 22.** – Using the **Line** command, sketch a profile similar to the one below. Be sure that your sketch forms a closed profile with no self intersecting lines.



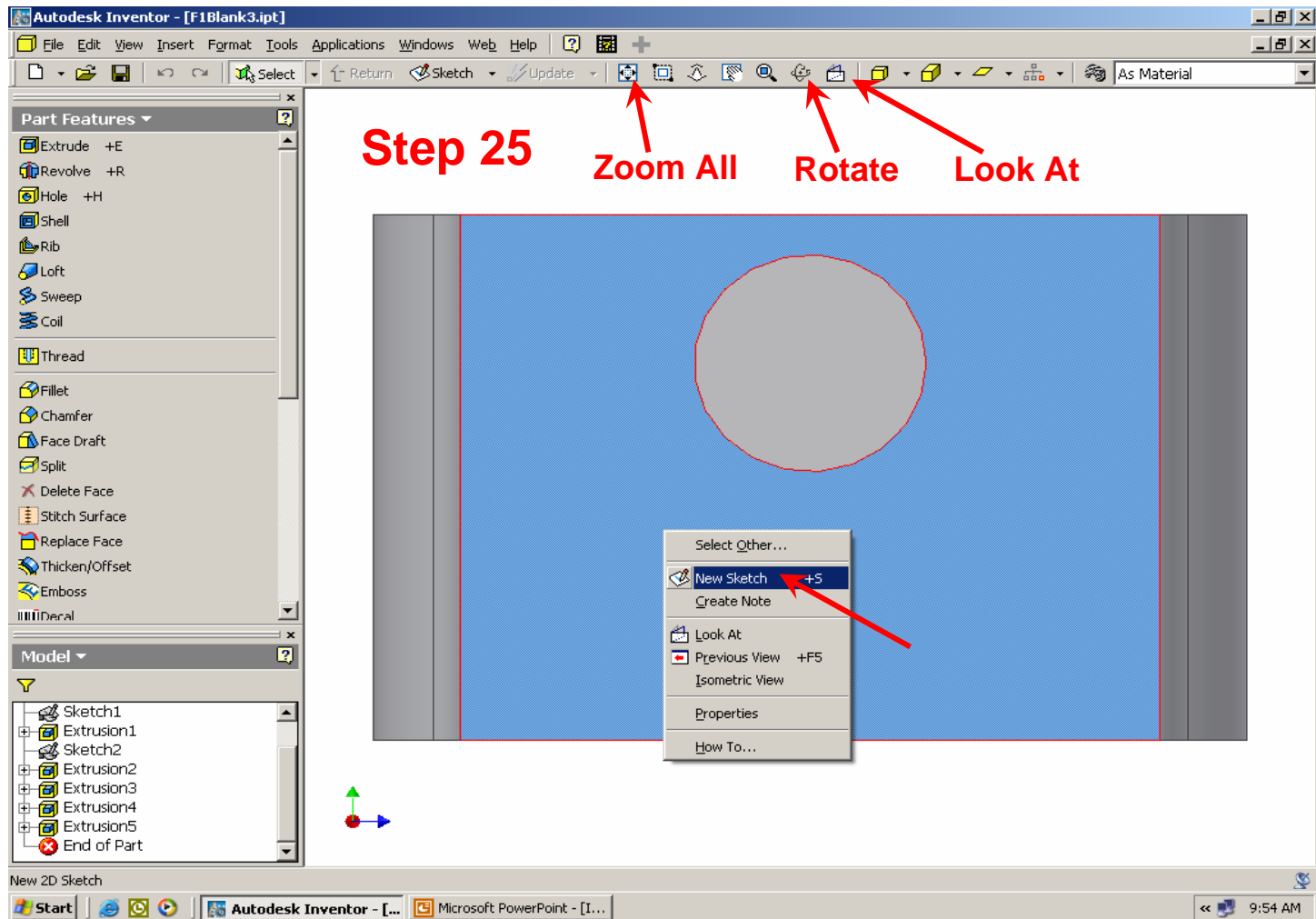
**Step 23.** – Switch to the **Part Features** command panel. (L) click on **Extrude** and (L) click again on the profile shown in blue below. (L) click on the **Cut** button and set the distance under **Extents** to **All**. (L) click on **OK** to complete the extrusion. NOTE: if a red cross is displayed in the Extrude dialog box, this indicates that there is an open profile in your sketch – return to the sketch mode and correct the problem.



**Step 24.** – From the Standard Toolbar, switch the display back to **Shaded Display**. Use the **Rotate** commands to examine your model.

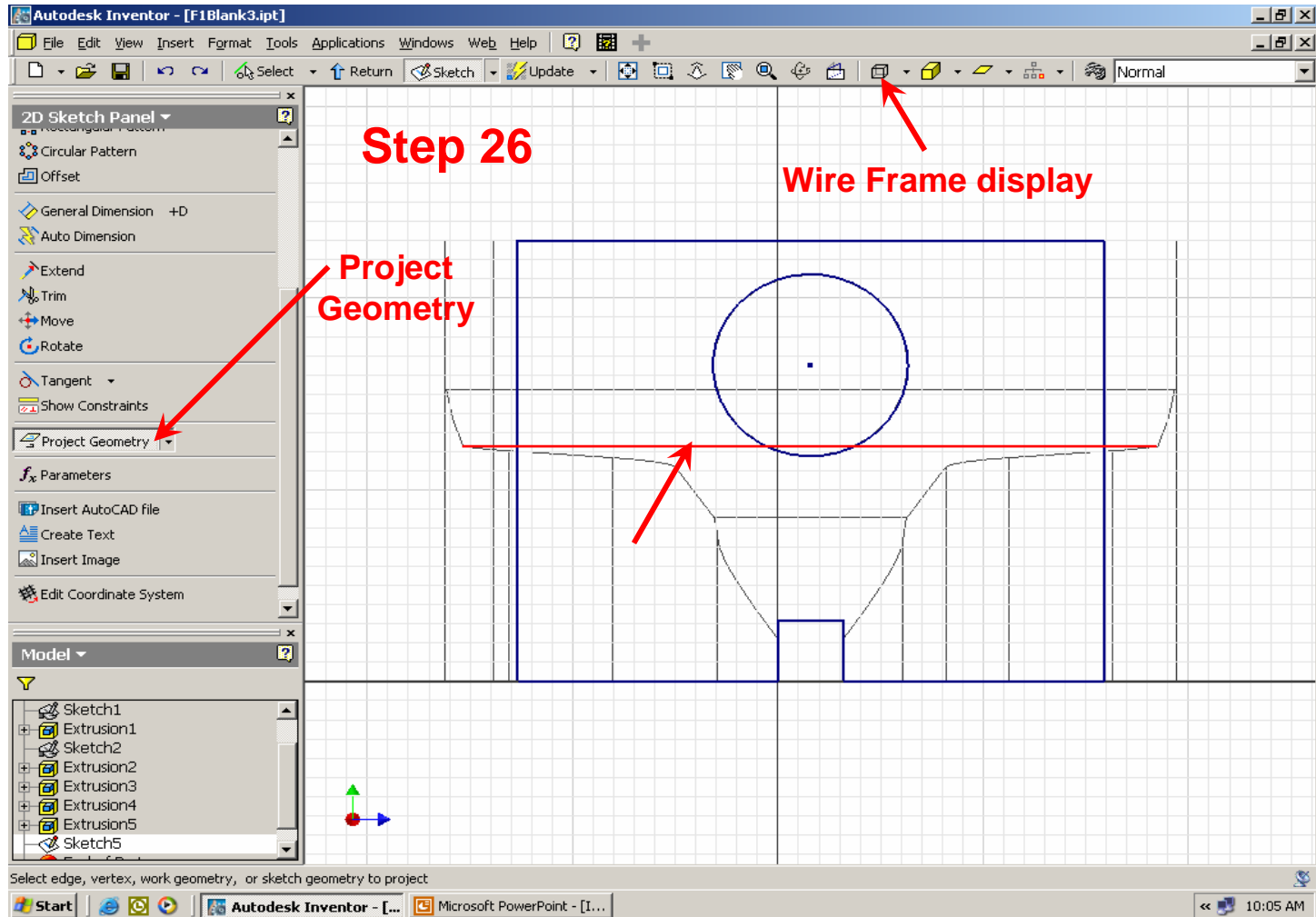


**Step 25.** – Use the **Rotate**, **Look At** and **Zoom All** commands in the Standard Toolbar to rotate your model so you are looking directly into the back end of the car. Move the cursor over the model, (R) click to select the surface highlighted below and select **New Sketch**.

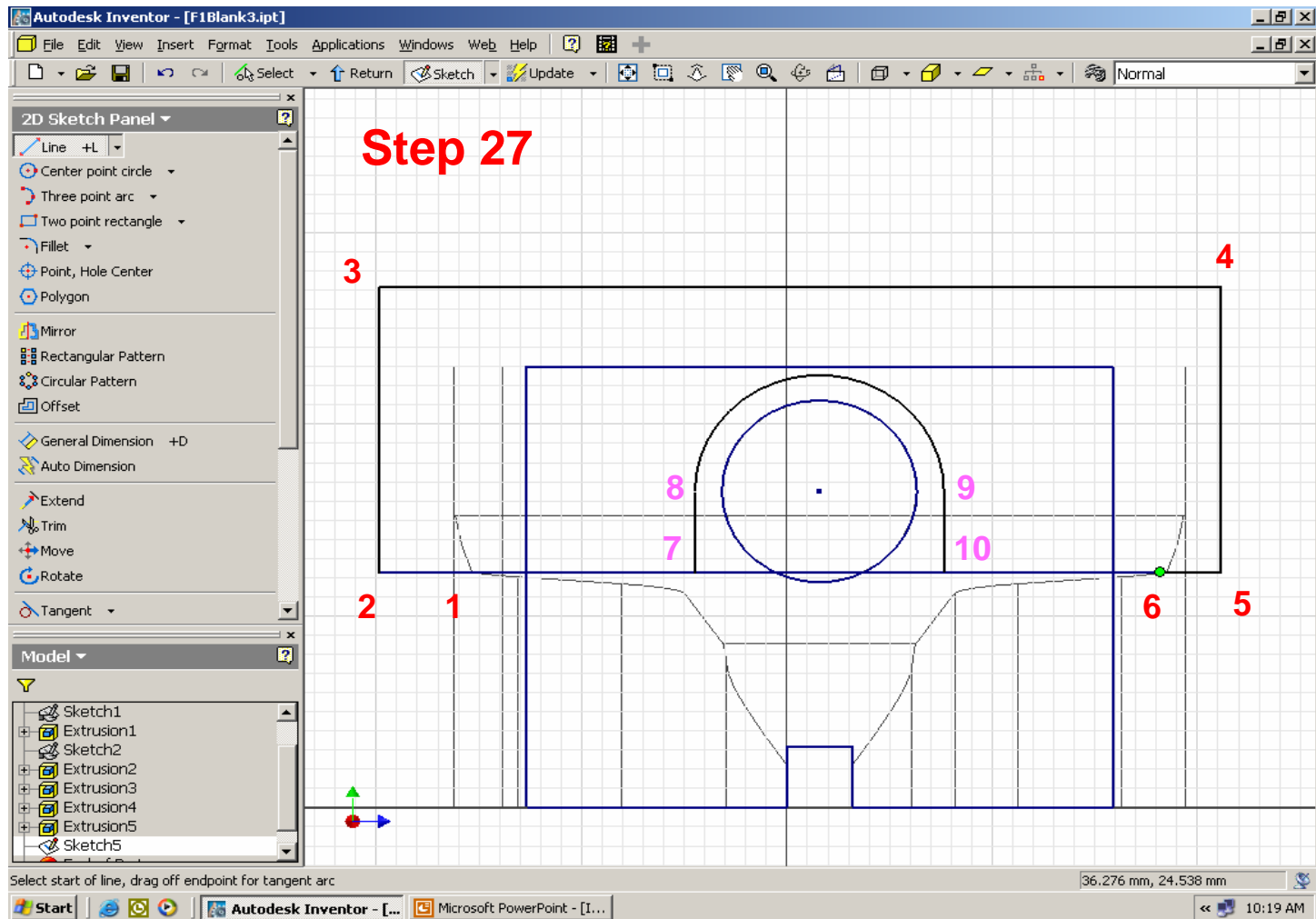




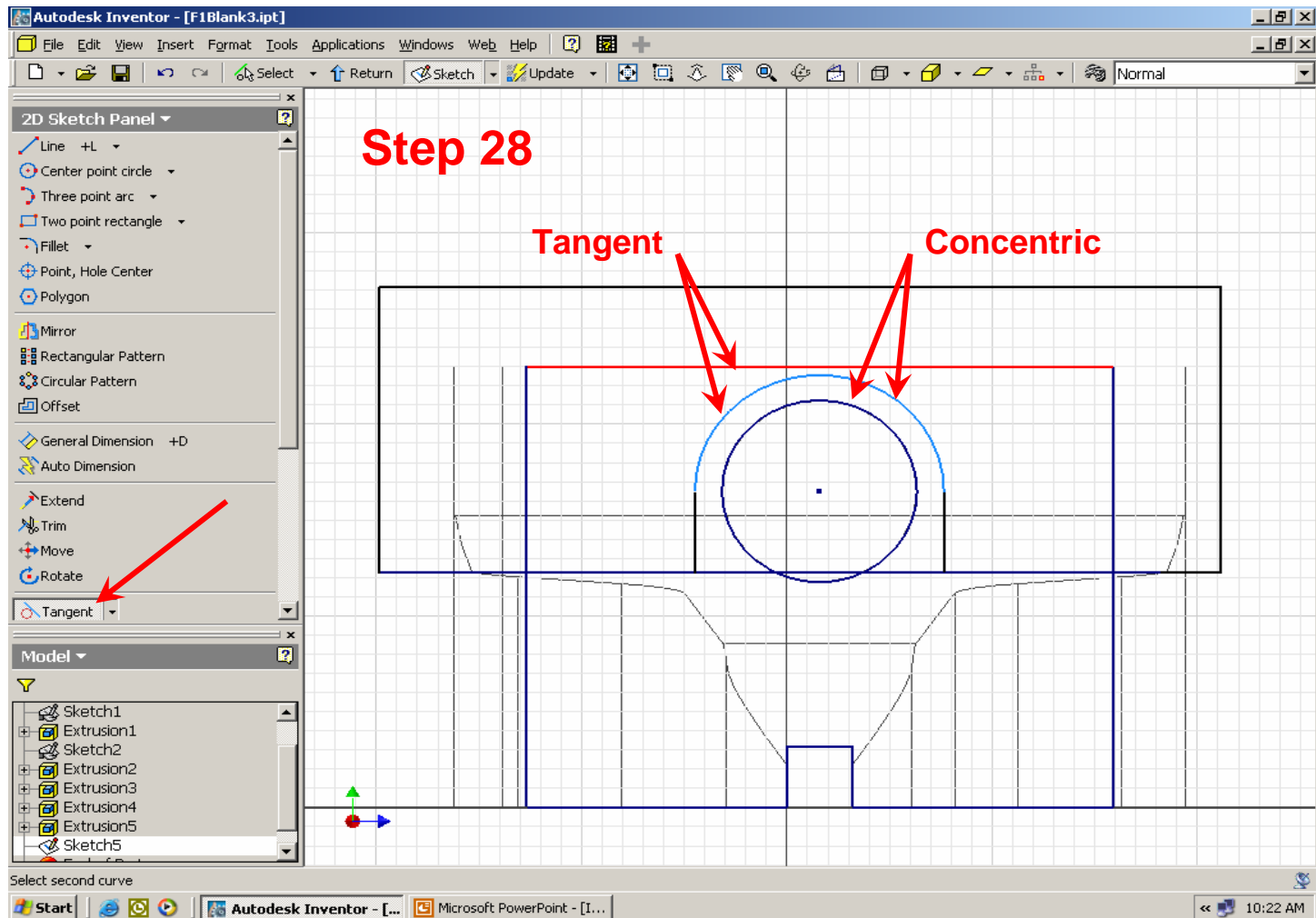
**Step 26.** – Change the display to **Wireframe**. Scroll the slider bar down and select the **Project Geometry** command. (L) click on the line highlighted in red below.



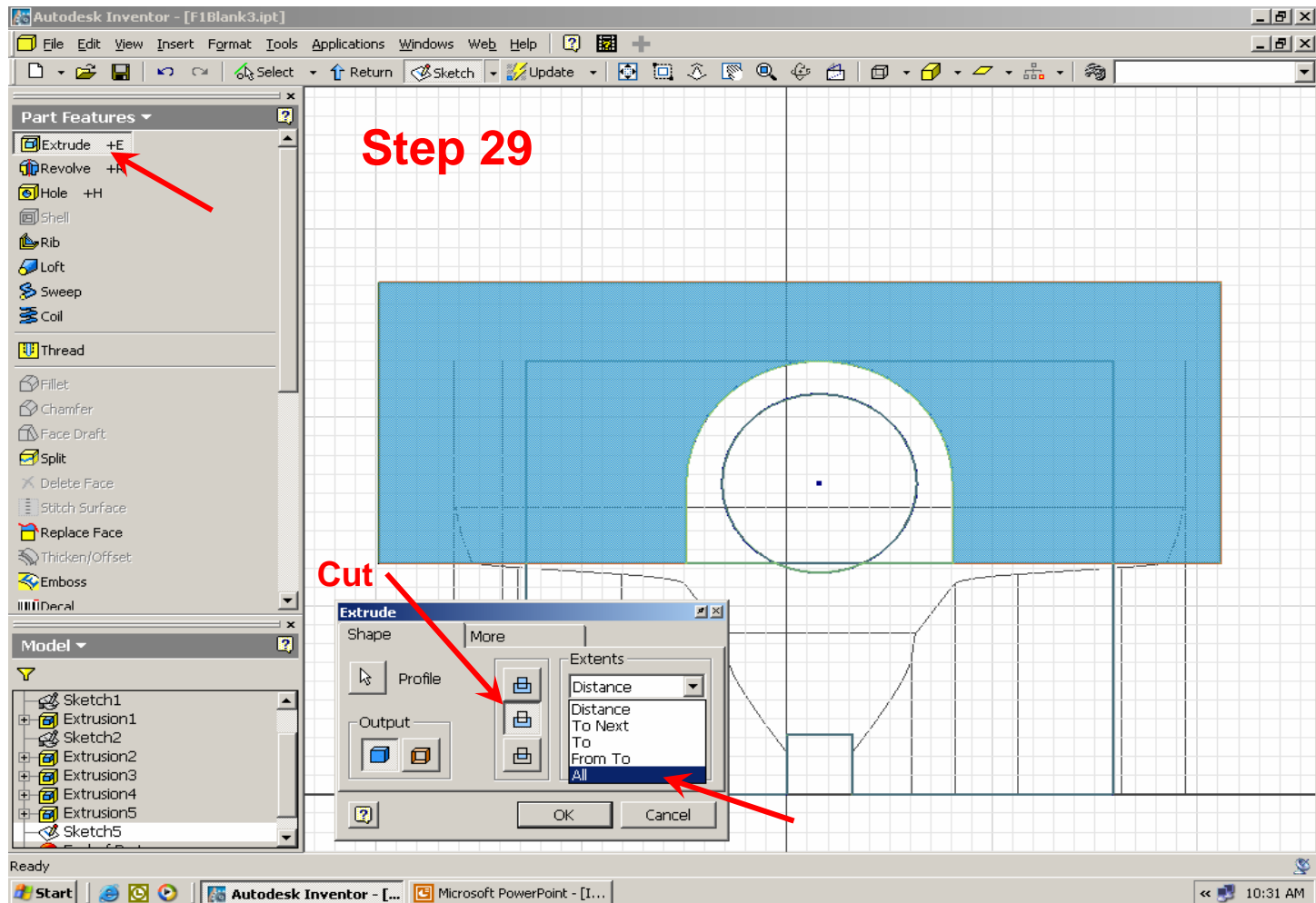
**Step 27.** – Use the **Line** command to sketch the lines 1-2-3-4-5-6 and lines 7-8-9-10.



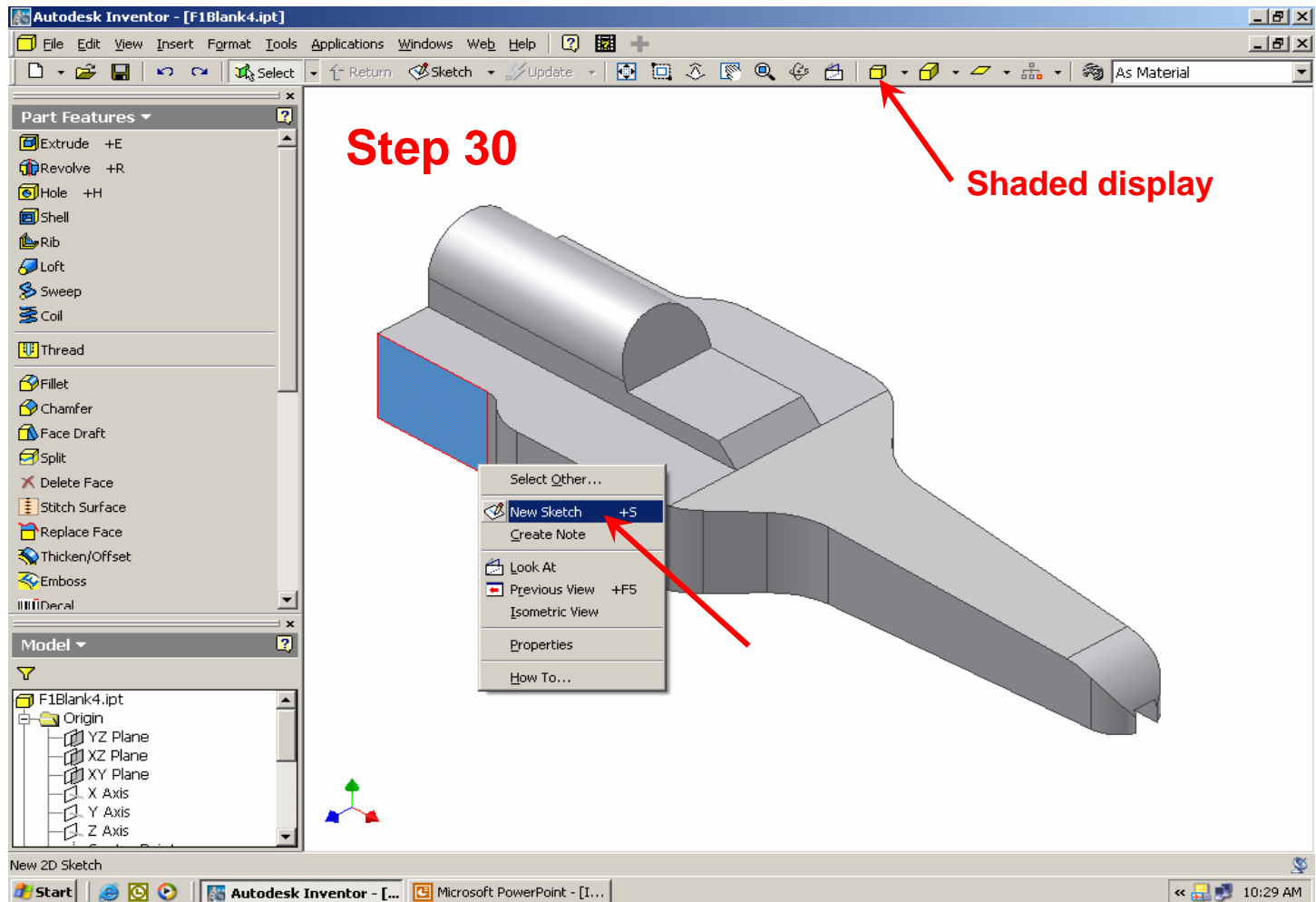
**Step 28.** – From the 2D Sketch Panel, scroll down with the slider bar to the Constraint command and select **Tangent**. (L) click on the arc and the line indicated below. **NOTE:** be sure that the circle and the arc are concentric (have the same center point). If not, also apply a **Concentric** constraint to the circle and arc.



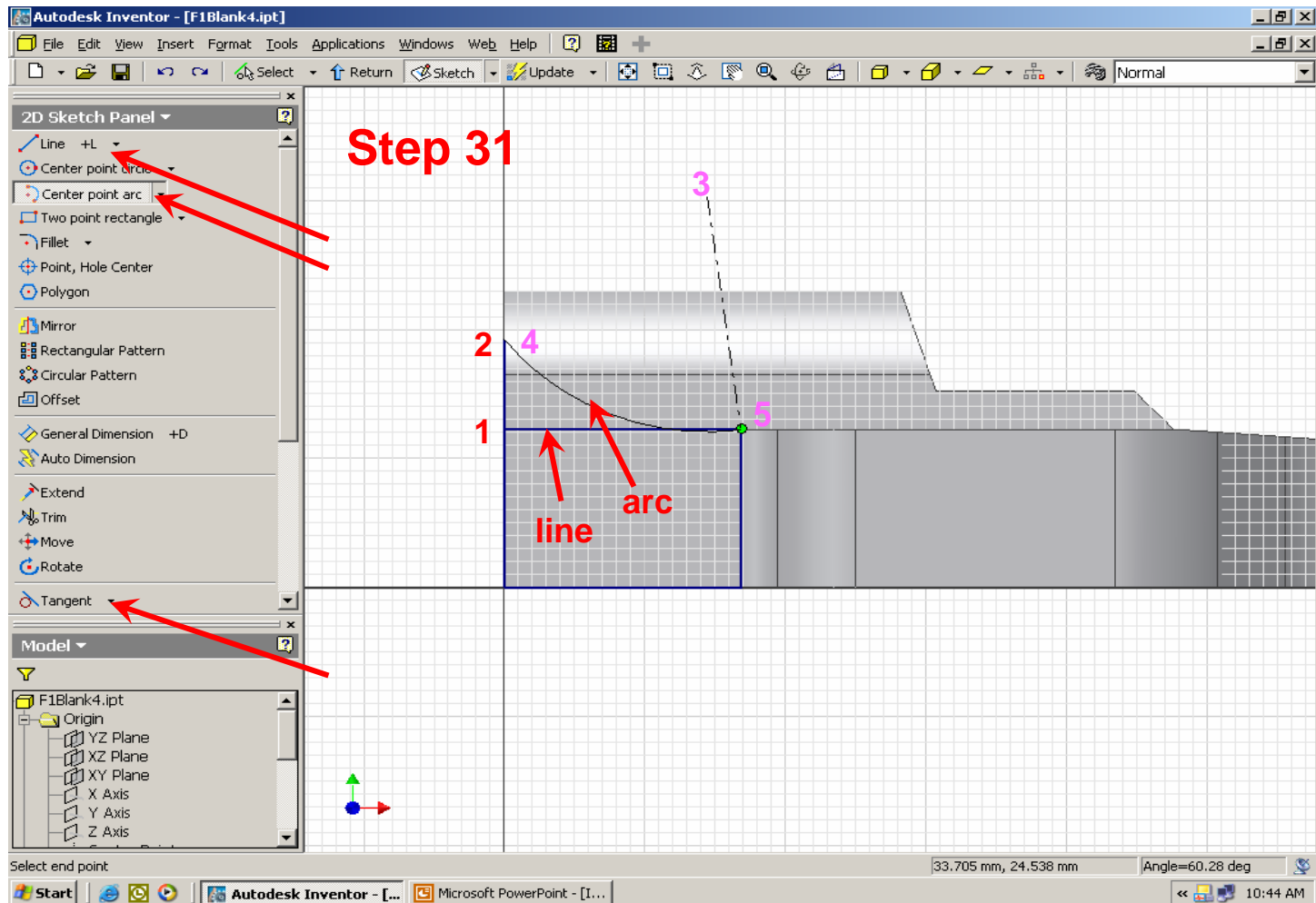
**Step 29.** – Switch to the **Part Features** command panel, and (L) click on **Extrude**. (L) click on the **Profile** highlighted below, select the **Cut** option and set the distance to **All**. (L) click on **OK** to complete the cut extrusion.



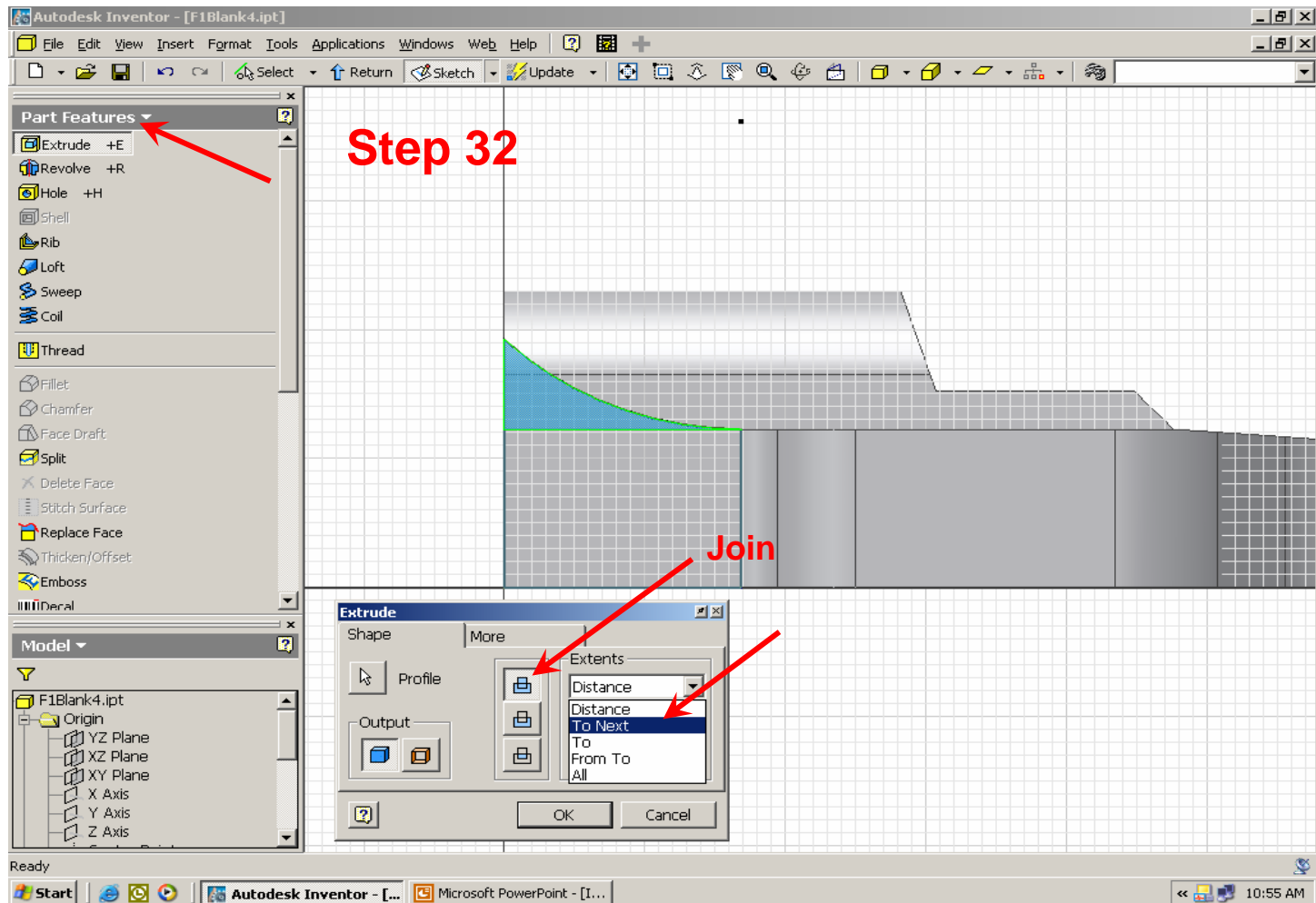
**Step 30.** – Switch back to the **Shaded Display** and examine your design. (R) click on the surface highlighted below and select **New Sketch**.



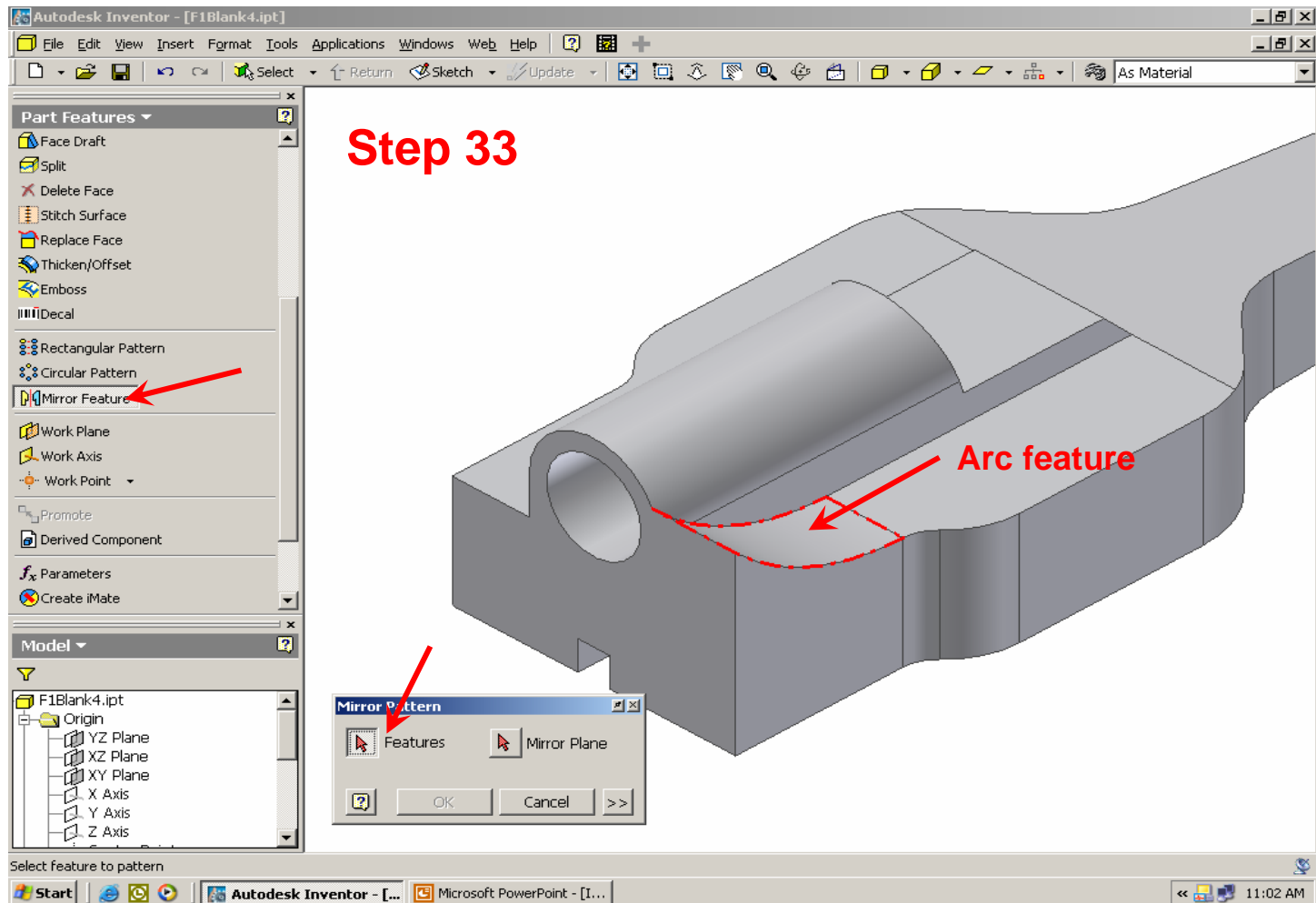
**STEP 31.** – Rotate the view so you are looking directly into the side of the car (see below). Use the **Line** command to draw the line 1-2. Next, select the **Center point arc** command and draw arc 3-4-5; then apply a **Tangent** constraint between the line and arc indicated below. **NOTE:** you may need to adjust the end point of the arc to be sure that it is locked onto the end point of the line (at point 5) so that it forms a **closed profile**.



**Step 32.** – Switch from the **2D Sketch Panel** to the **Part Features** panel and select **Extrude**. Select the **Profile** highlighted below with a (L) click, be sure the **Join** button is depressed and set the Distance to **To Next**. Click on **OK** to complete the join extrusion.

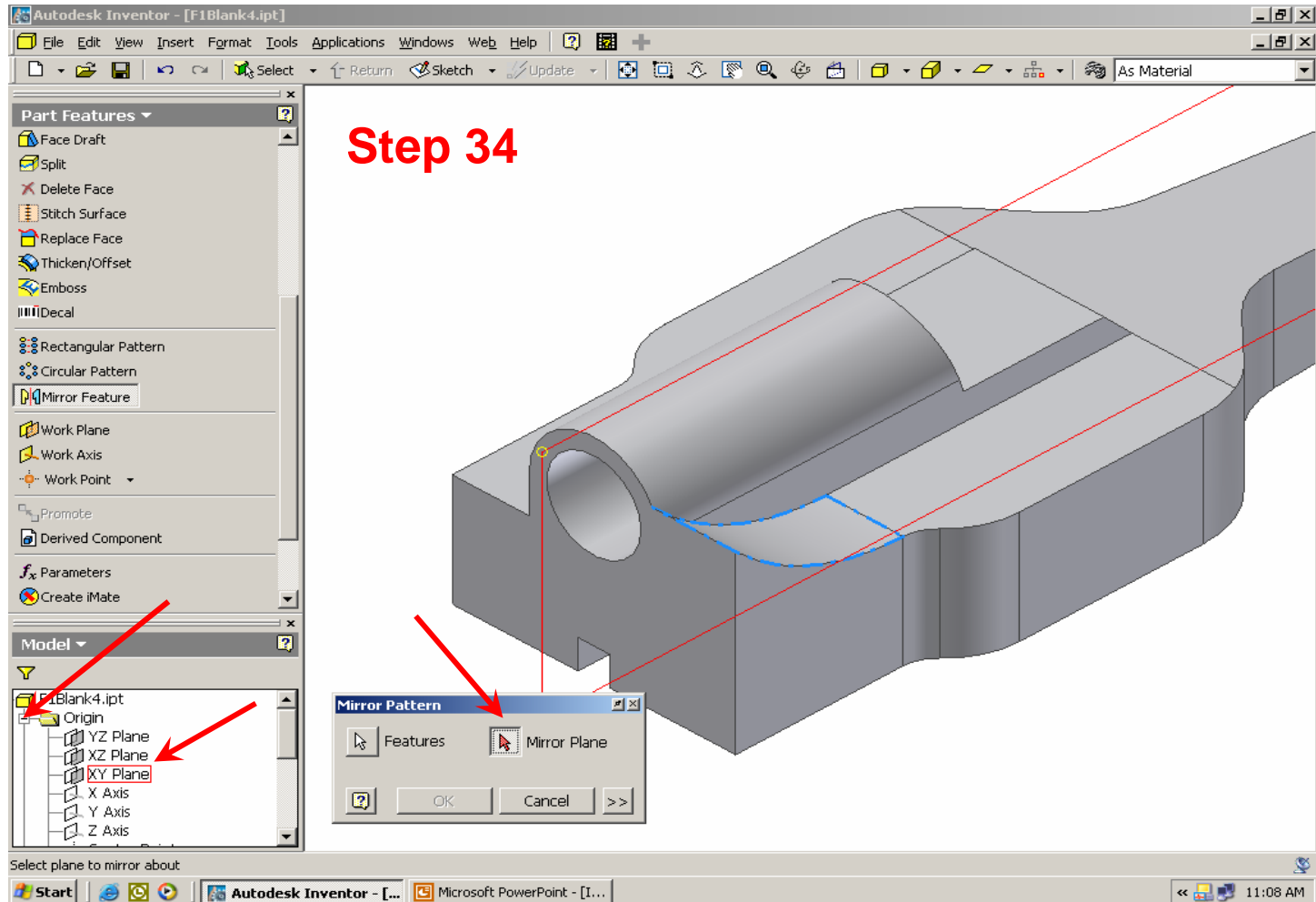


**Step 33.** – Rotate your model to the isometric view shown below. Use the slider bar to scroll down the Part Features panel to the **Mirror Feature** command. (L) click on the arc feature that was extruded in the previous step.

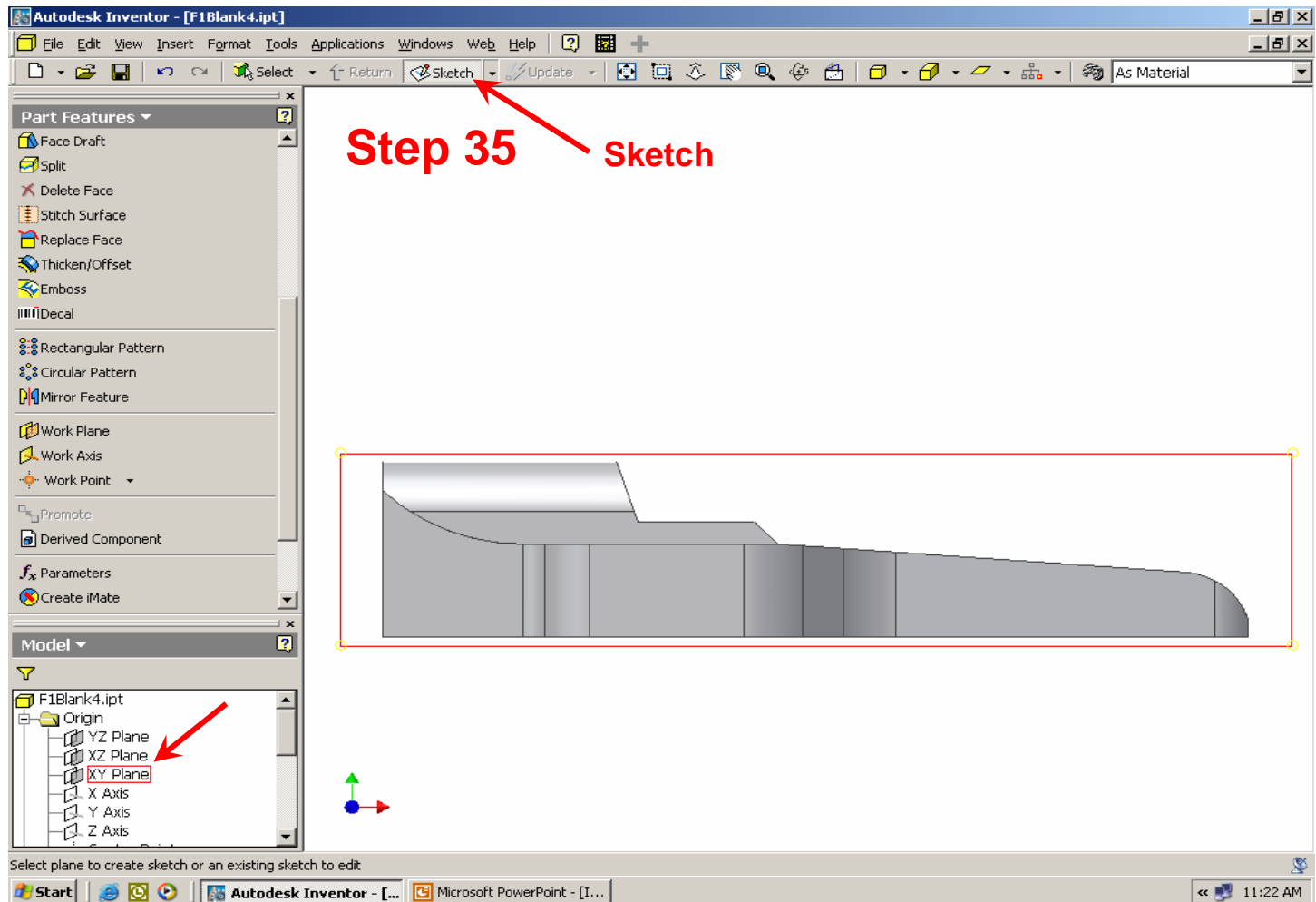




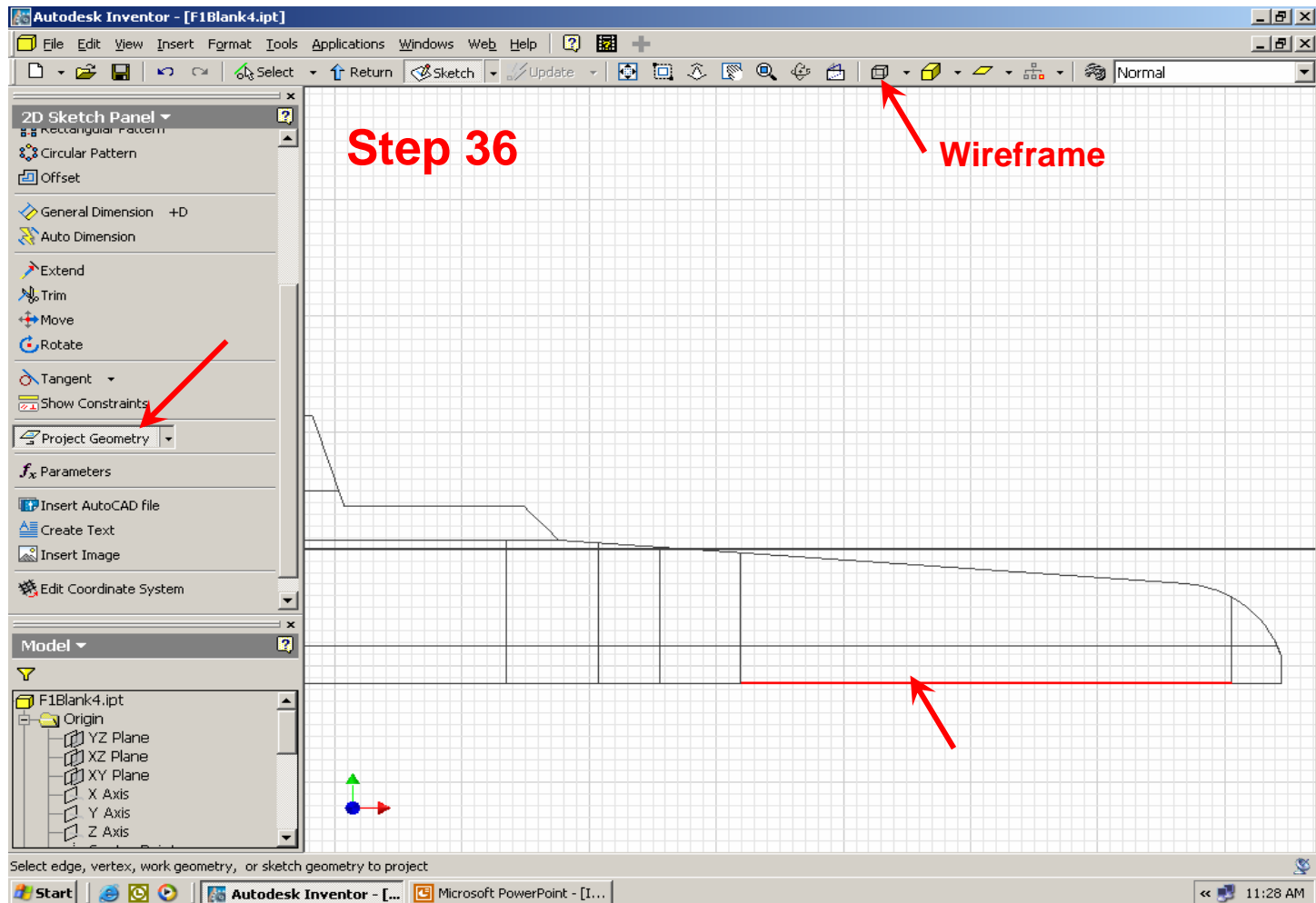
**Step 34.** – (L) click on the **Mirror Plane** button and, from the **Model Browser**, (L) click on the “+” sign to the left of **Origin**. Then, (L) click on **XY Plane**. **NOTE:** that the different planes will highlight in the graphics window as you move the cursor over them. Finish the **Mirror Feature** by (L) clicking on **OK**.



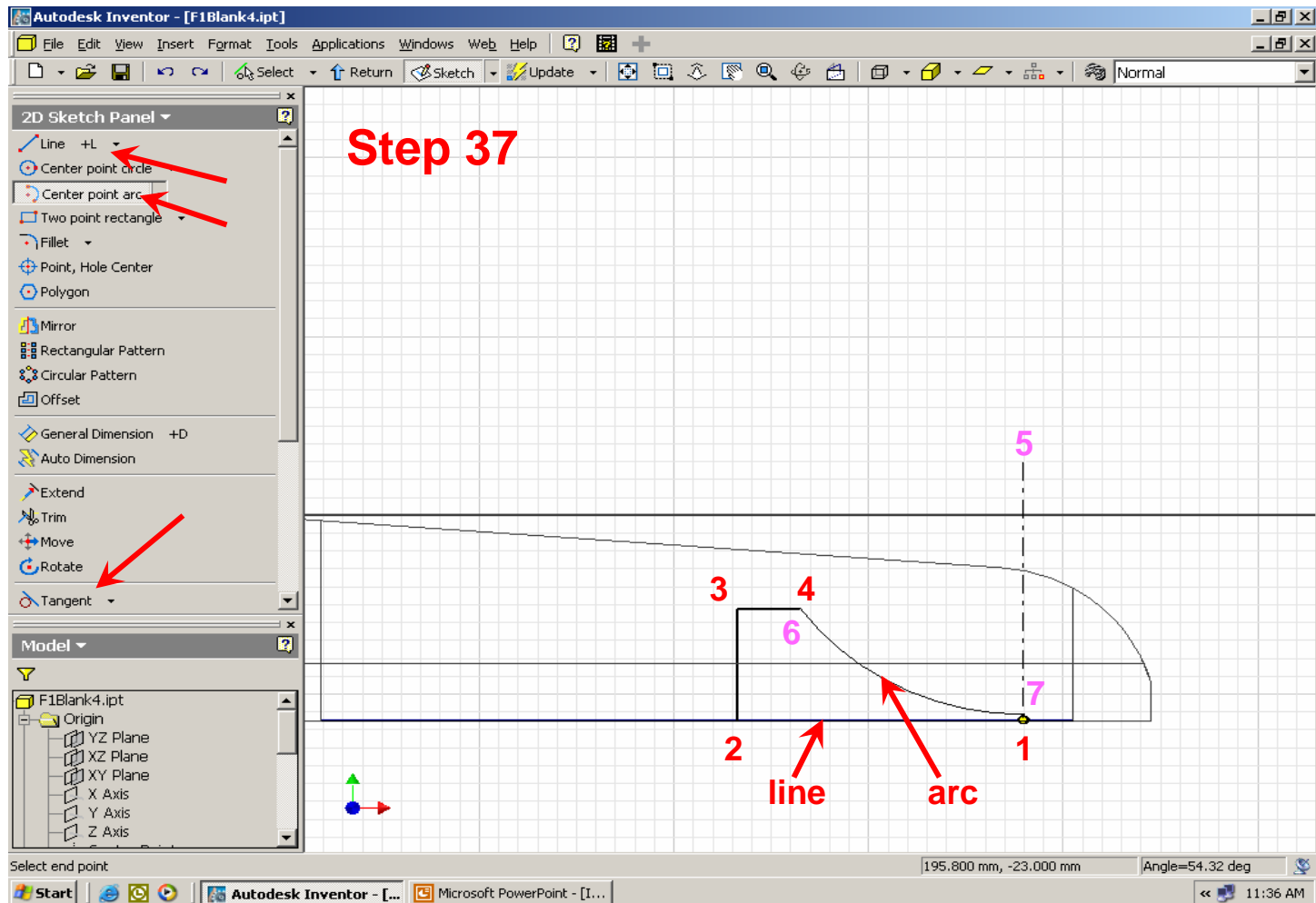
**Step 35.** – Rotate back to the side view of the car and select the **Sketch** command from the standard toolbar. In the **Model** browser, select the **XY Plane** to open a new sketch



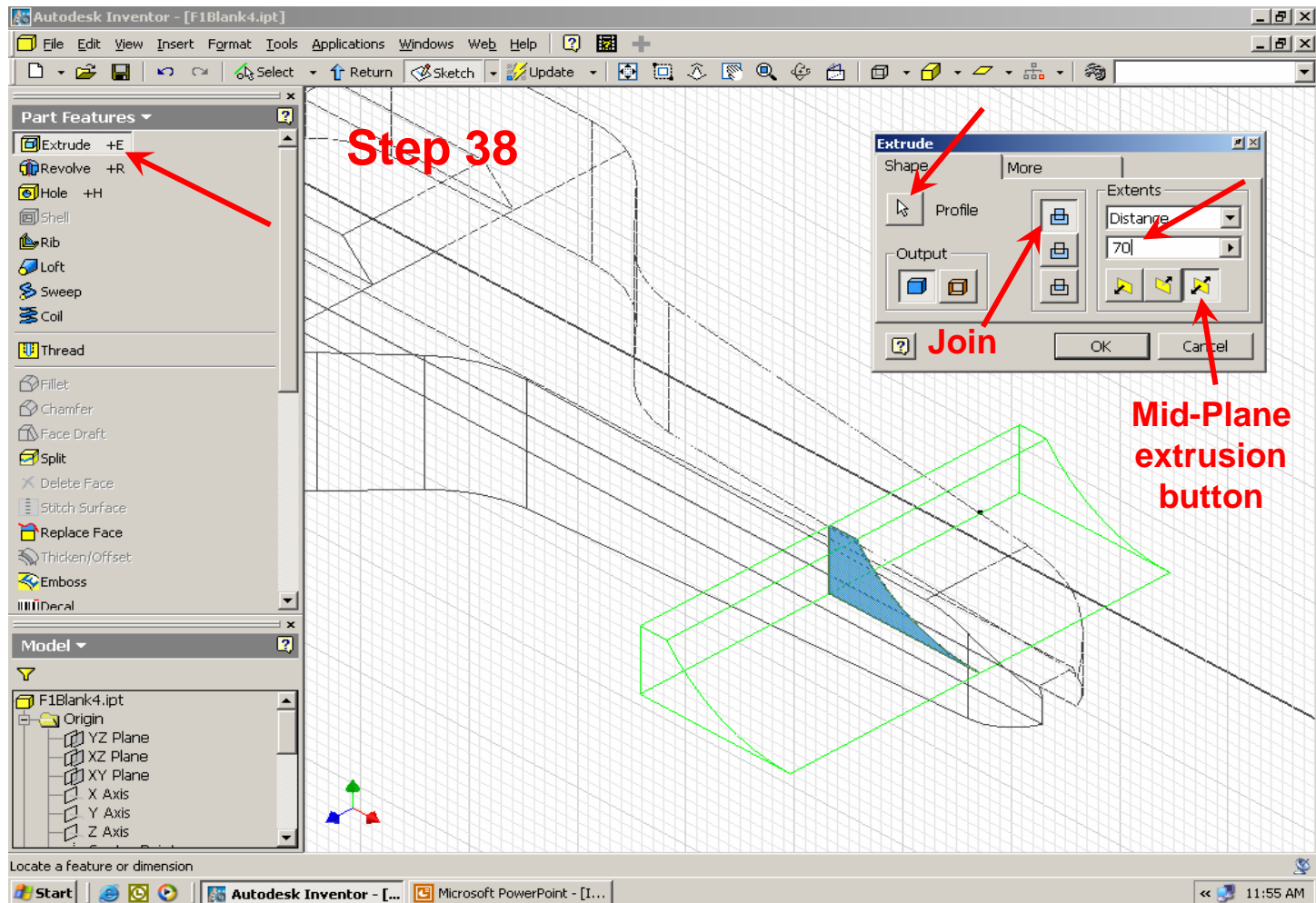
**Step 36.** – Pan and Zoom in on the nose of the car and switch display mode to **Wireframe**. From the **2D Sketch Panel**, select **Project Geometry** and (L) click on the line highlighted in red below.



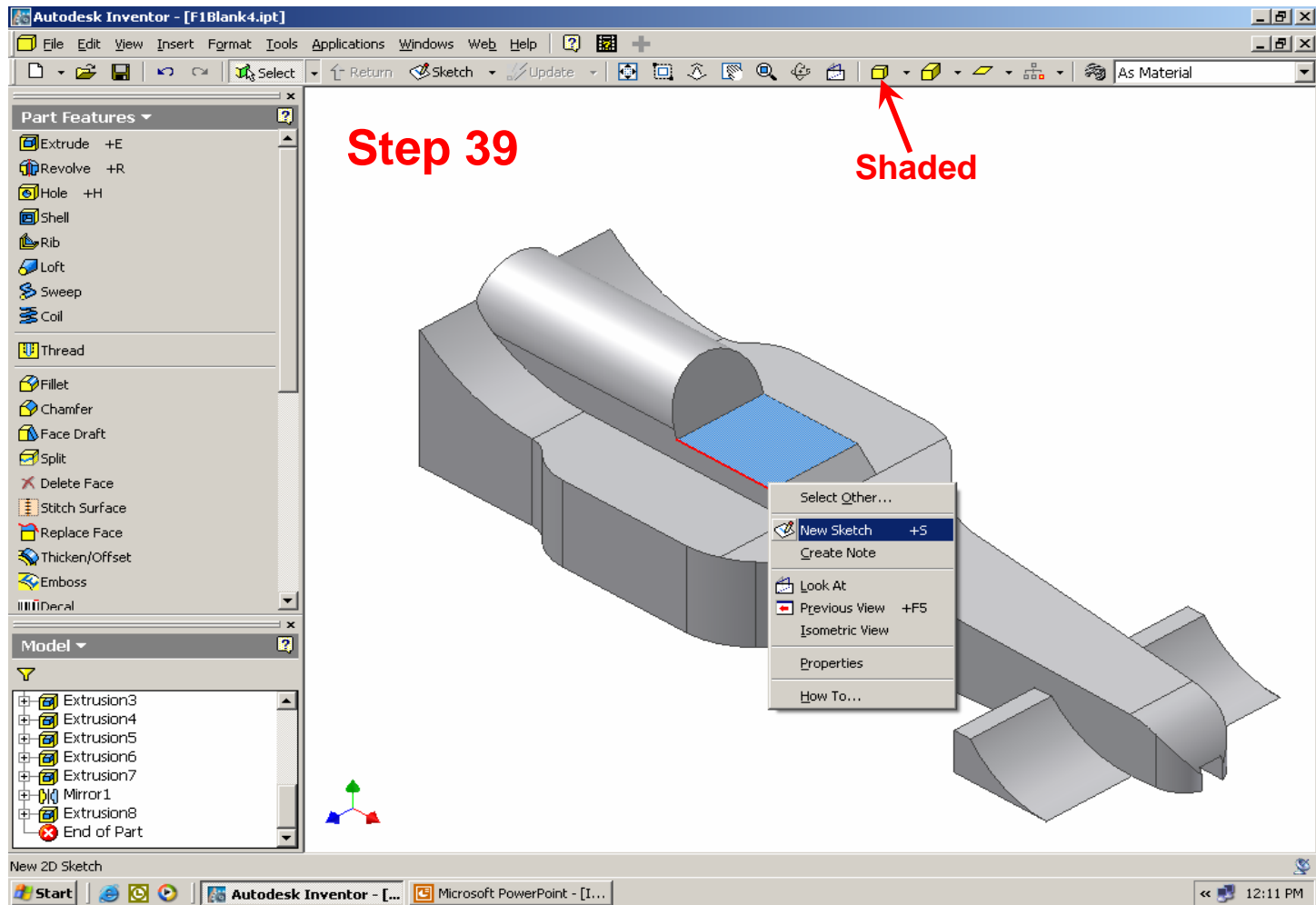
**Step 37.** – Use the **Line** command to draw lines 1-2-3-4. Use the **Center point arc** command to draw arc 5-6-7. Apply a **Tangent** constraint between the line and arc shown below. **NOTE:** be sure that the end points of the arc are locked onto the endpoints of the lines so that they form a **closed profile**.



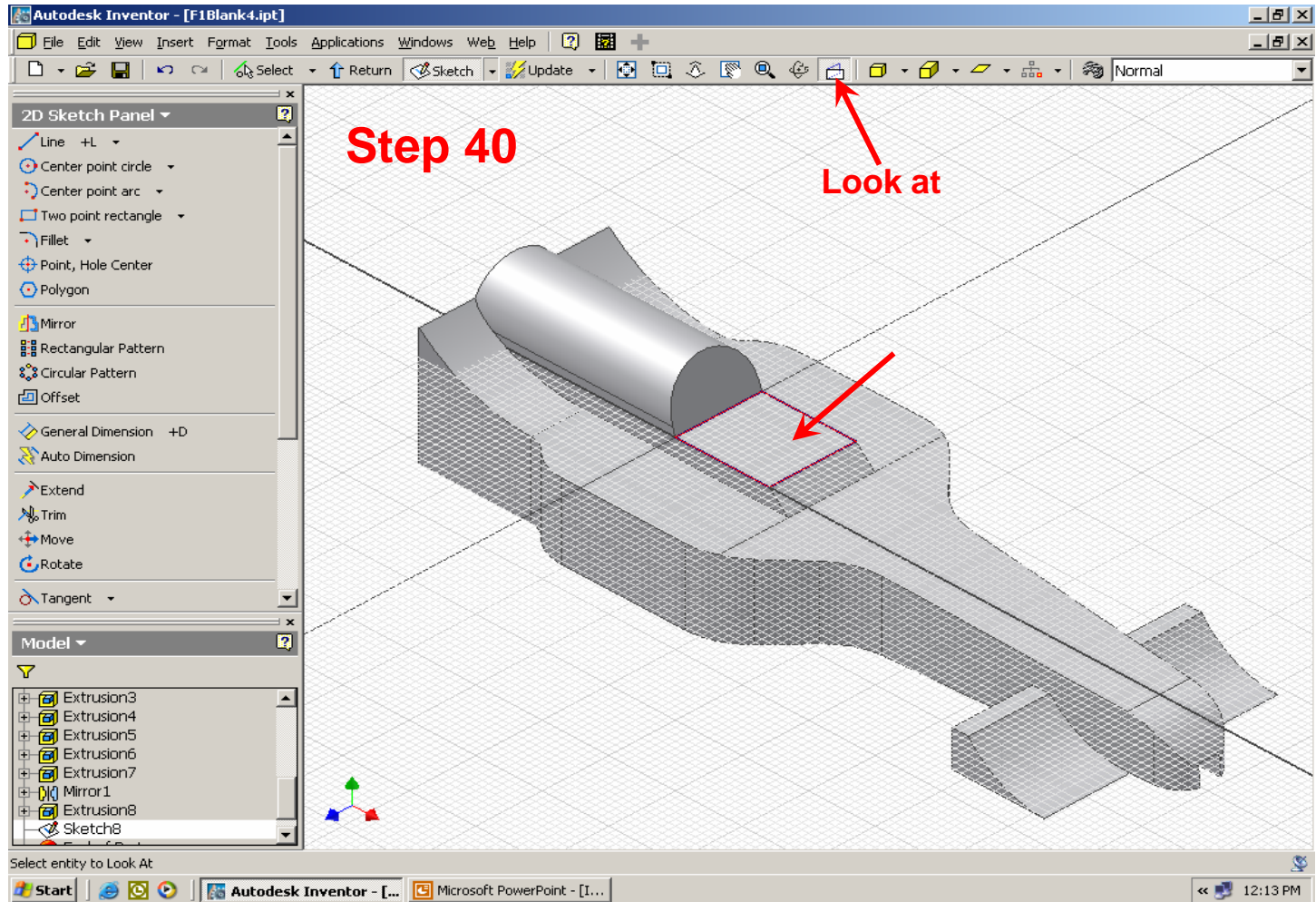
**Step 38.** – Rotate the display to an isometric view. Switch from the **2D Sketch Panel** to the **Part Features** panel and select **Extrude**. (L) click on the **Profile** highlighted in blue below. Be sure the **Join** button is depressed, set the **Distance** to **70** and (L) click on the **Mid-Plane** extrusion button. Check to be sure that the extrusion preview (highlighted in green) looks similar to what is shown below. If so, (L) click on **OK**.



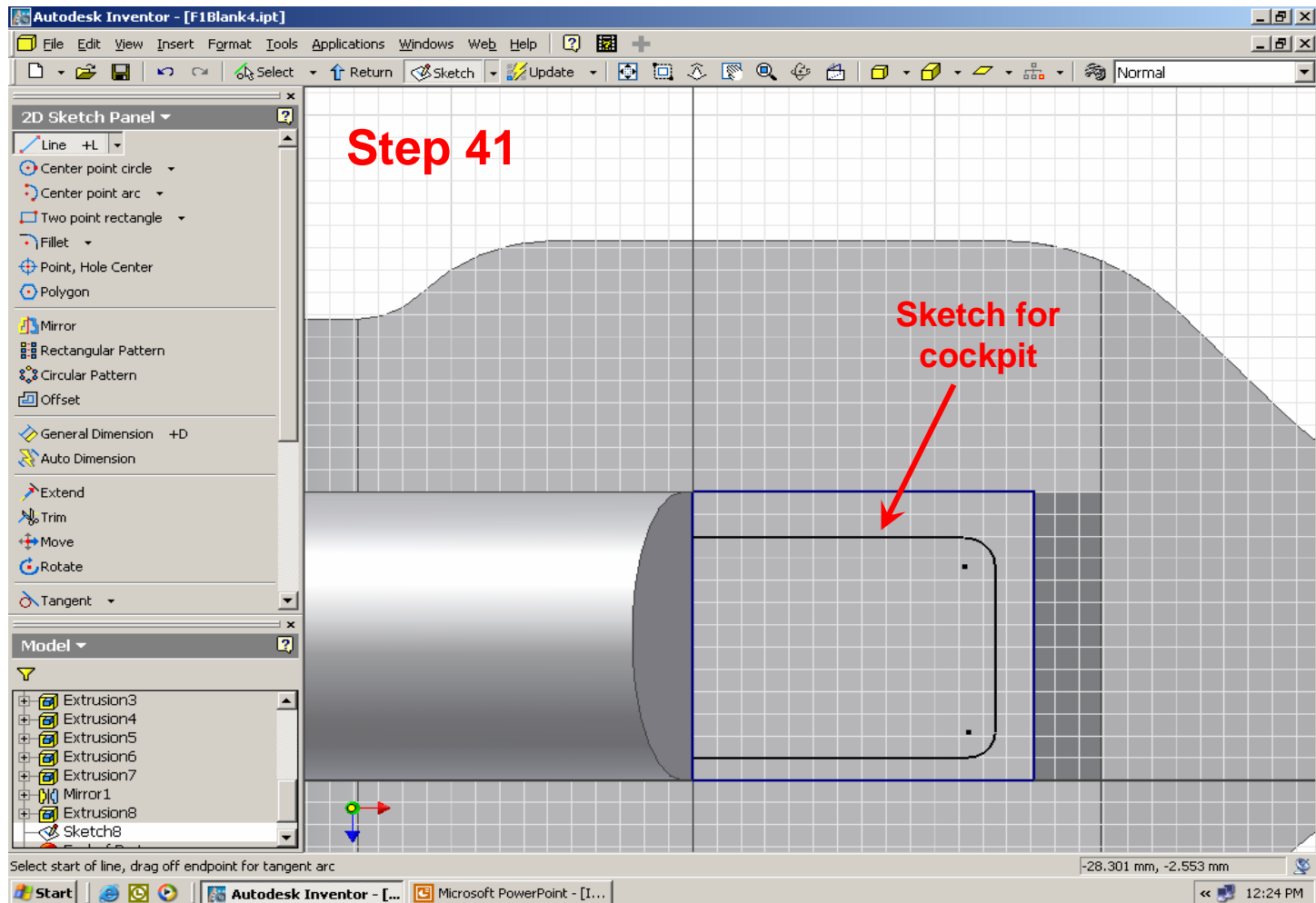
**Step 39.** – Switch the display back to **Shaded**. (R) click on the surface highlighted in blue, below, and (L) click on **New Sketch**.



**Step 40.** – (L) click on the **Look At** button on the **Standard Toolbar** and (L) click on the surface highlighted below.

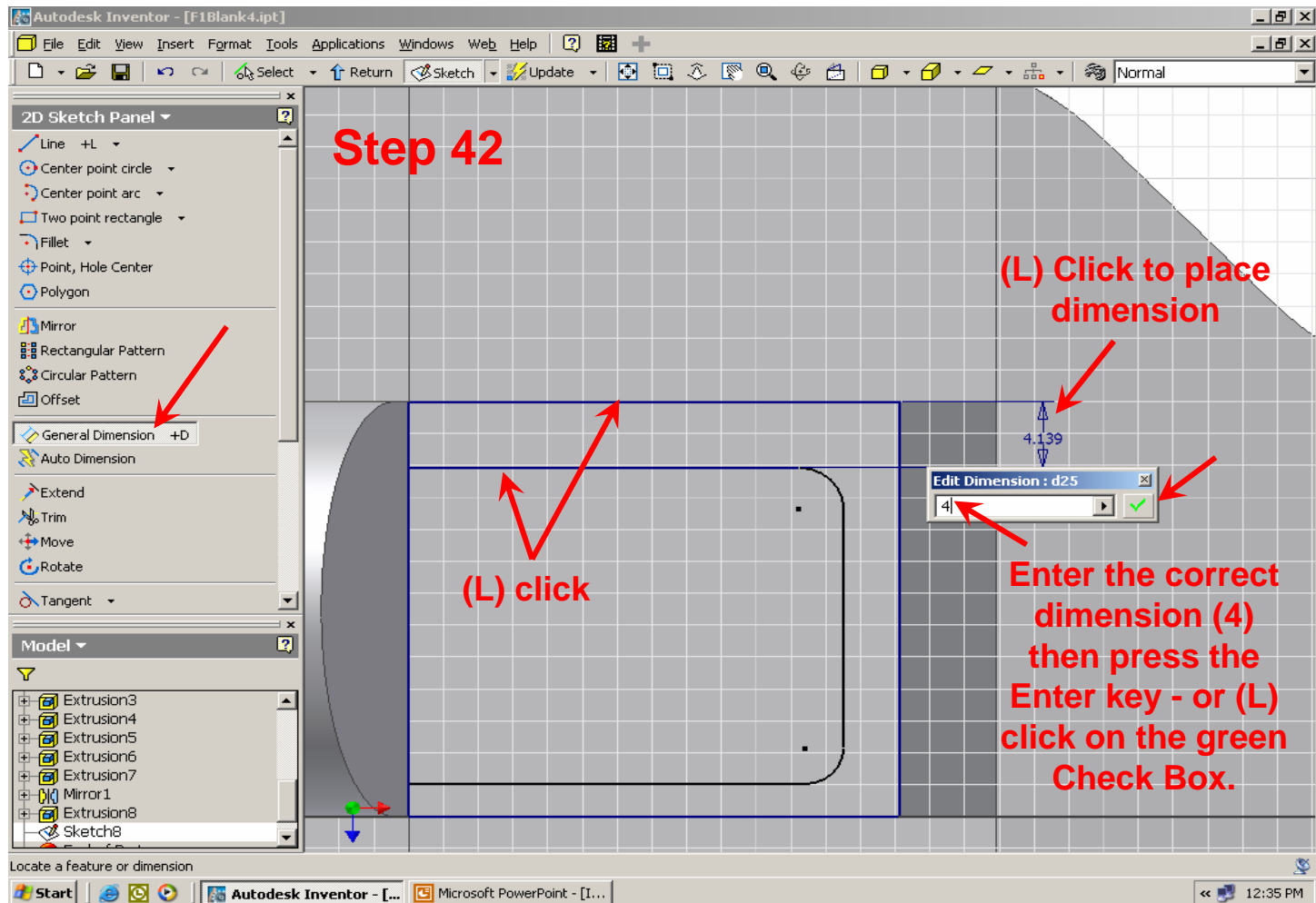


**Step 41.** – **Zoom** in on the cockpit area of the car and use the **Line** command to draw the sketch shown below.

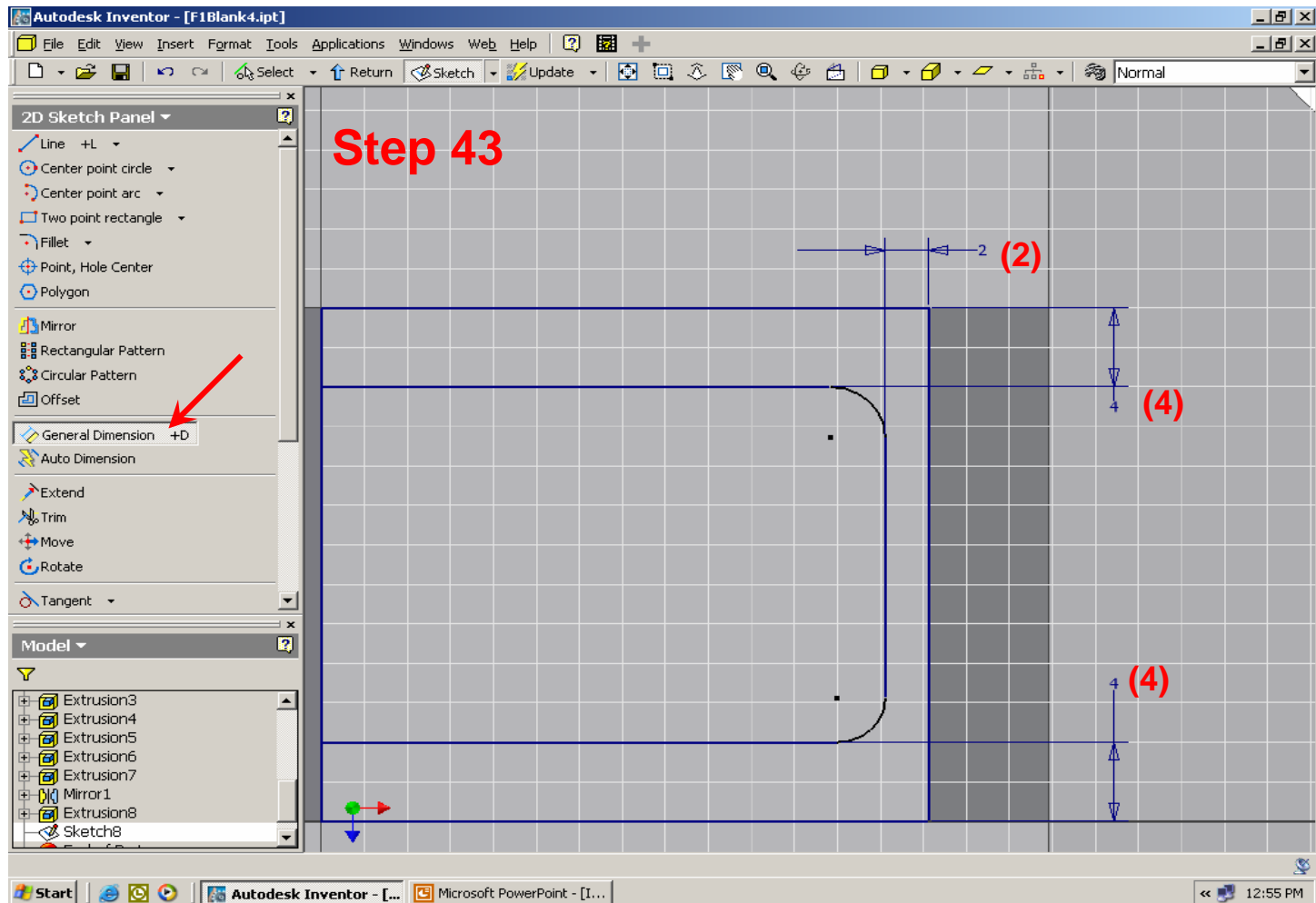




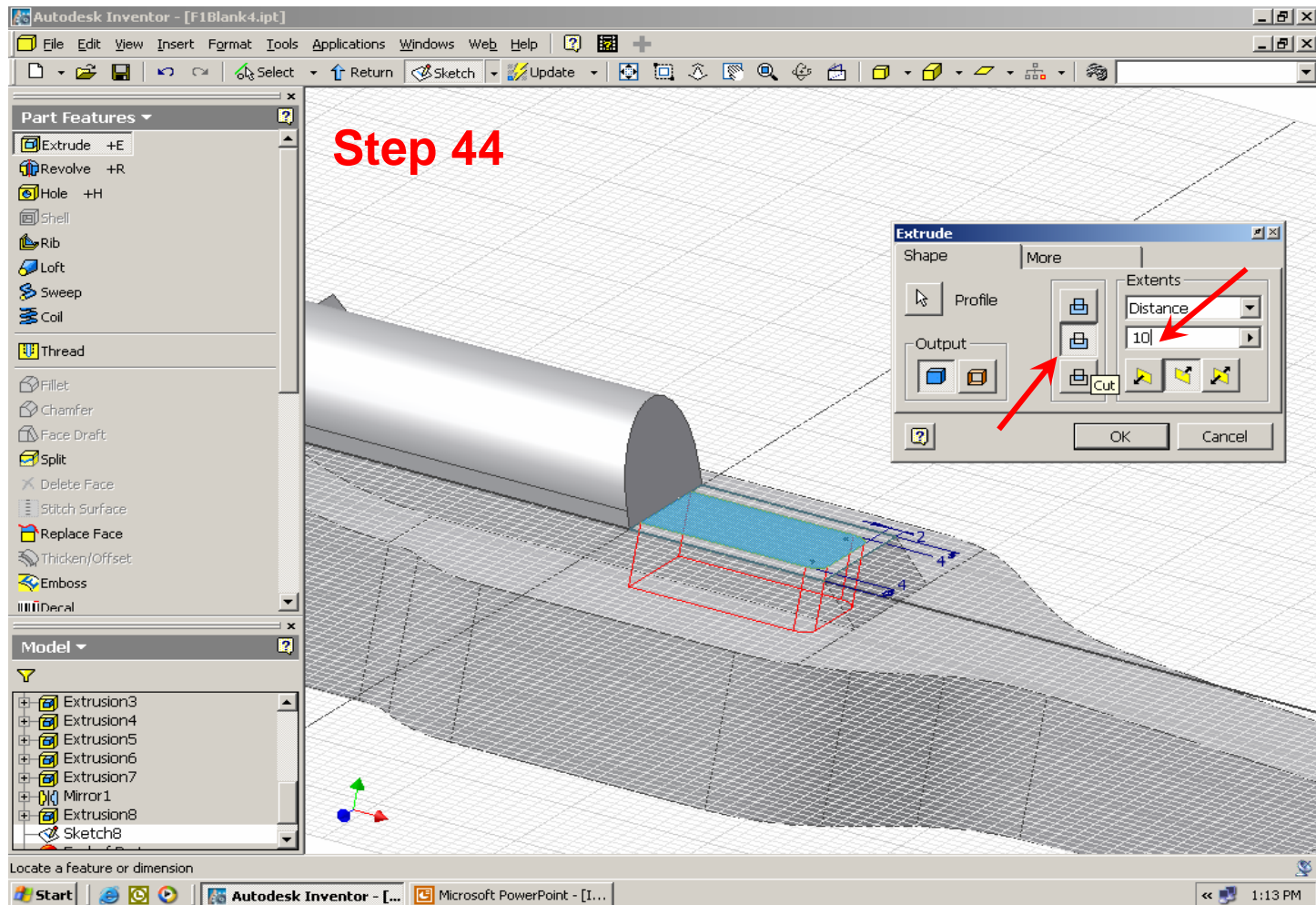
**Step 42.** – On the **2D Sketch Panel**, (L) click on the **General Dimension** command. (L) click on the 2 lines you want to dimension then move your cursor to where you want to place the dimension and (L) click again. This will open the **Edit Dimension** dialog box and display the actual dimension. From the keyboard, type in the correct dimension (**4**) and press the **Enter** key or (L) click on the green check mark in the **Edit Dimension** dialog box. Notice how the lines move according to the dimension you entered. This is called *parametric dimensioning*.



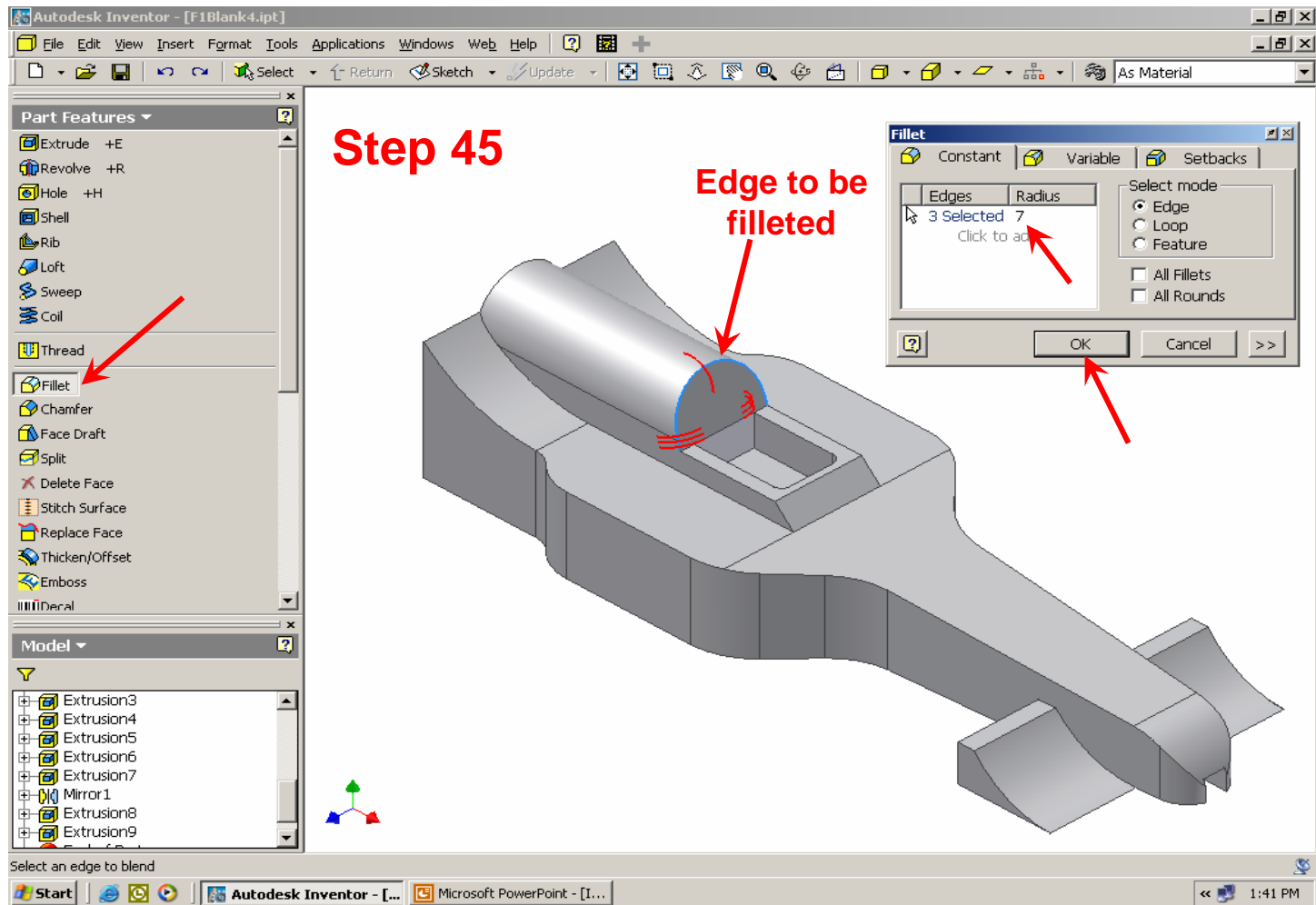
**Step 43.** – Continue using the **General Dimension** command to add the dimensions shown below. **NOTE:** after dimensions have been placed on a sketch, they can be easily changed at any time by double (L) clicking on the dimension you want to change. This will re-open the **General Dimension** dialog box. Type the new dimension and press the **Enter** key or (L) click on the green check in the **Edit Dimension** dialog box. Your sketch will automatically re-size to the new dimension. Try changing dimensions before going on to the next step.



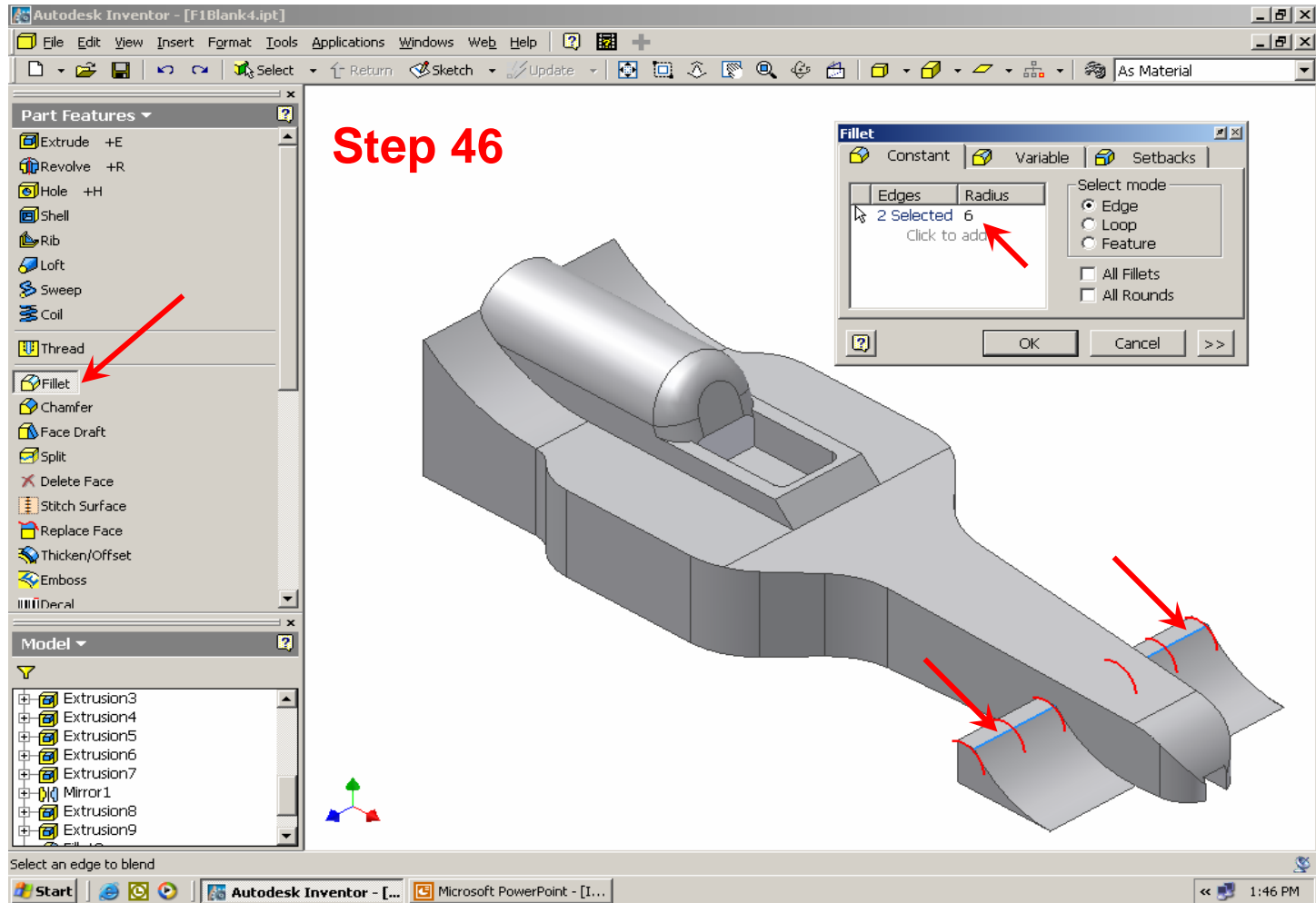
**Step 44.** – Rotate the view similar to that shown below. Switch from the **2D Sketch Panel** to the **Part Features** panel. (L) click on **Extrude**. (L) click on the Profile highlighted in blue, (L) click on the **Cut** button and set the **Distance** to **10**. Complete the Cut extrusion for the cockpit by (L) clicking on the **OK** button.



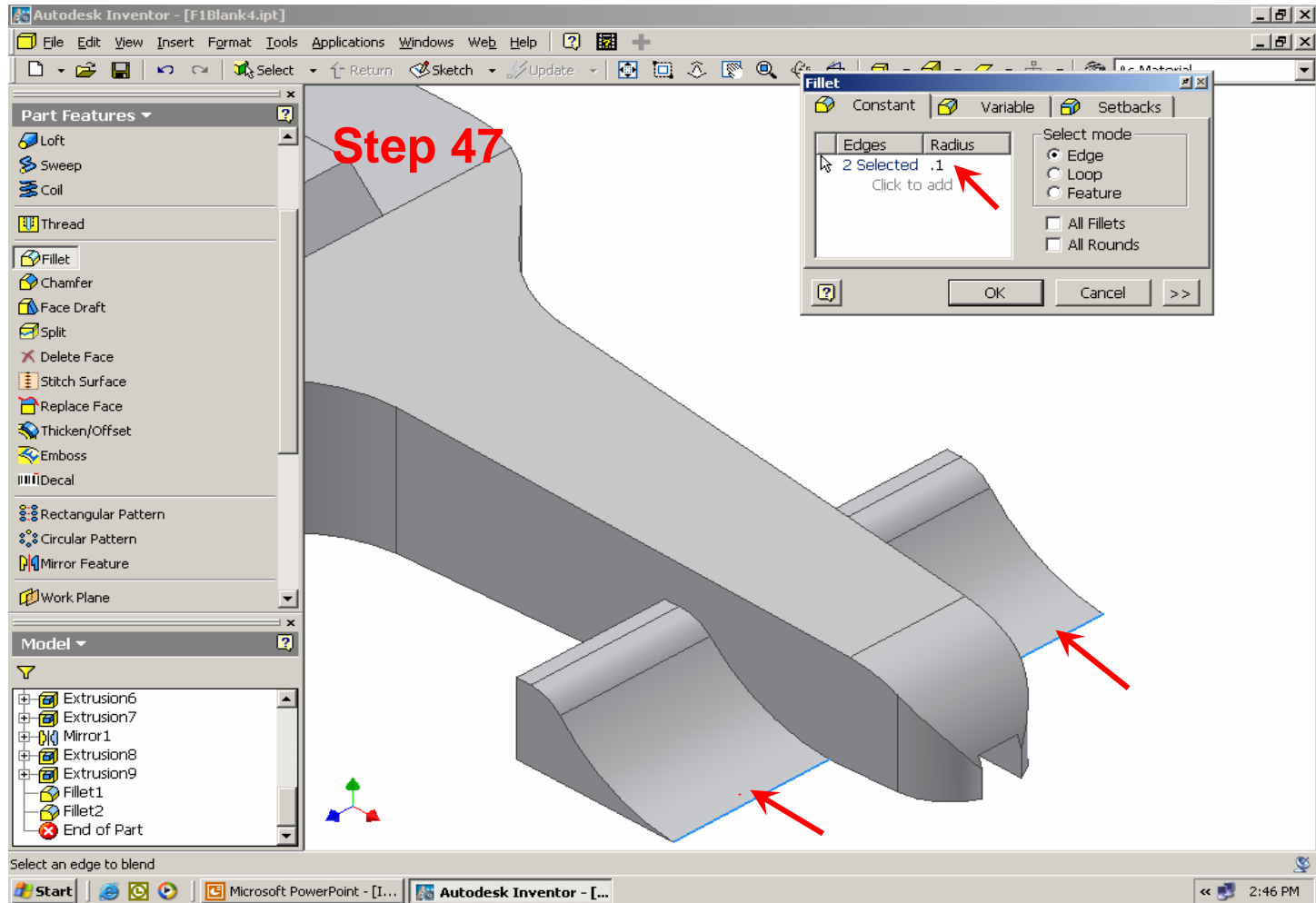
**Step 45.** – This completes the basic design of your F1 car. To finish the design process, you will need to smooth or round the sharp edges with the **Fillet** command. On the **Part Features** panel, (L) click on **Fillet**. The **Fillet** dialog box will open. Under **Radius**, type in the number **7**. (L) click on the edge shown below (the red arcs give a preview of what the fillet will look like) and (L) click on the **OK** button. **NOTE:** that multiple edges that are to have the same radius fillets can all be selected at the same time by (L) clicking on them.



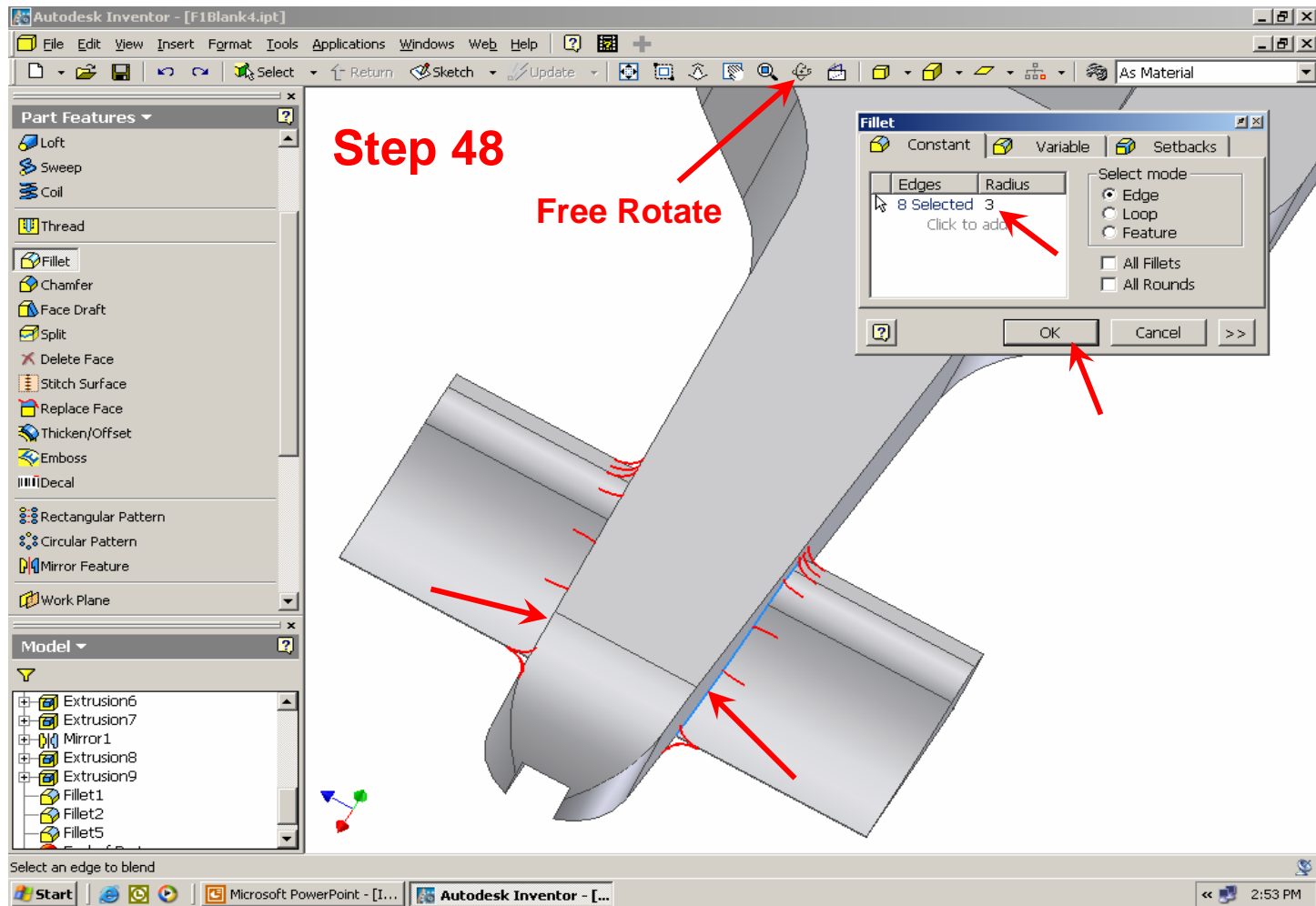
**Step 46.** – (L) click on the **Fillet** command again, set the fillet radius to **6** and select the 2 edges shown below. (L) click on **OK** to complete the fillets.



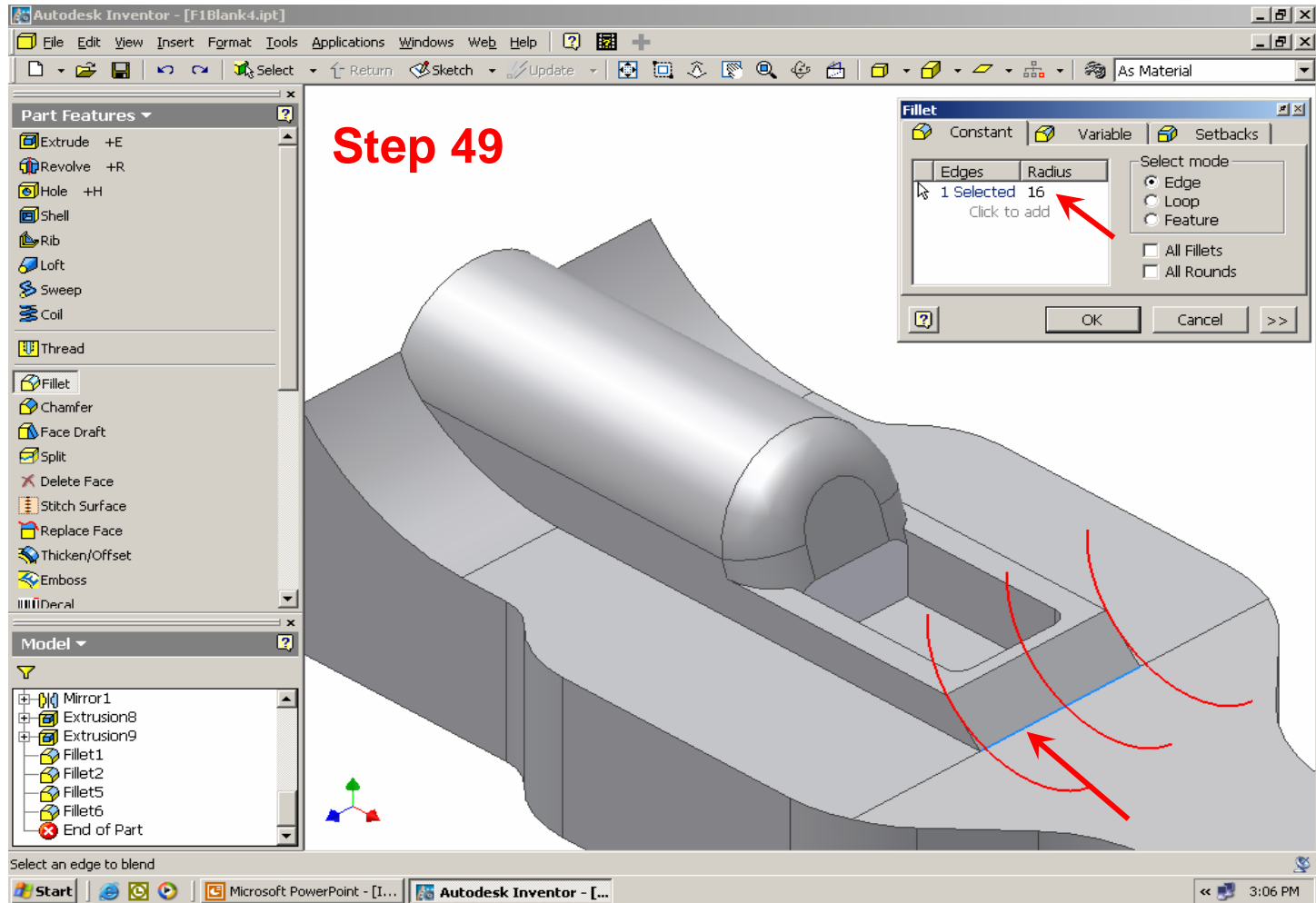
**Step 47.** – Select the **Fillet** command again, set the radius to **.1** and select the 2 edges highlighted in blue below. (L) click on **OK**.



**Step 48.** – Select the **Fillet** command, set the radius to **3** and select the 2 edges shown below. NOTE: you will have to use the **Free Rotate** command, while in the middle of the **Fillet** command, in order to select both edges. Select the first edge, (L) click on the **Free Rotate** command and rotate the view until you can see the second edge, press the **Escape** key to exit the **Free Rotate** command, then select the second edge. (L) click on **OK**.

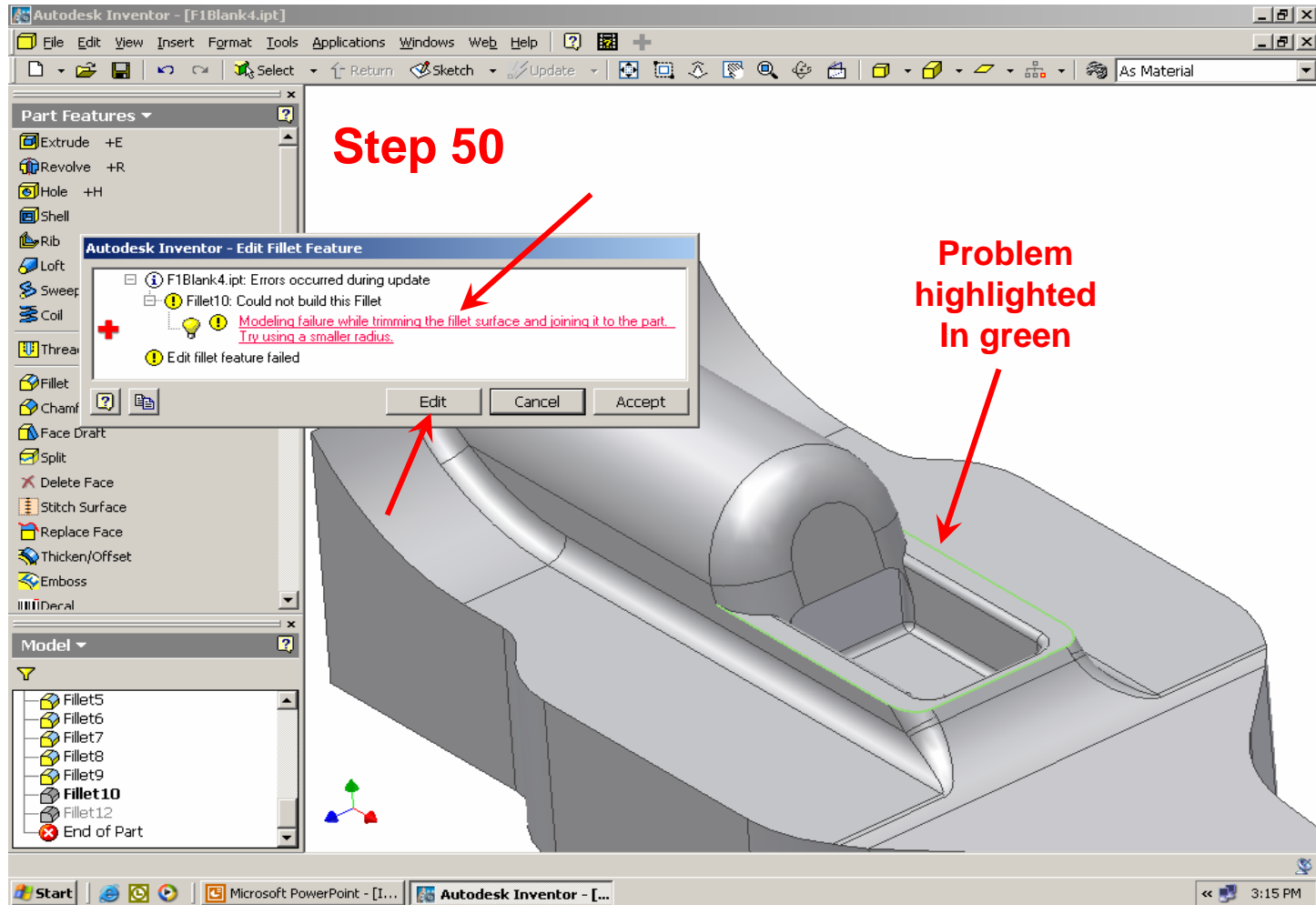


**Step 49.** – Pan and Zoom in on the cockpit area of your car. (L) click on **Fillet**, set the radius to **16** and select the edge highlighted below. (L) click on **OK**.

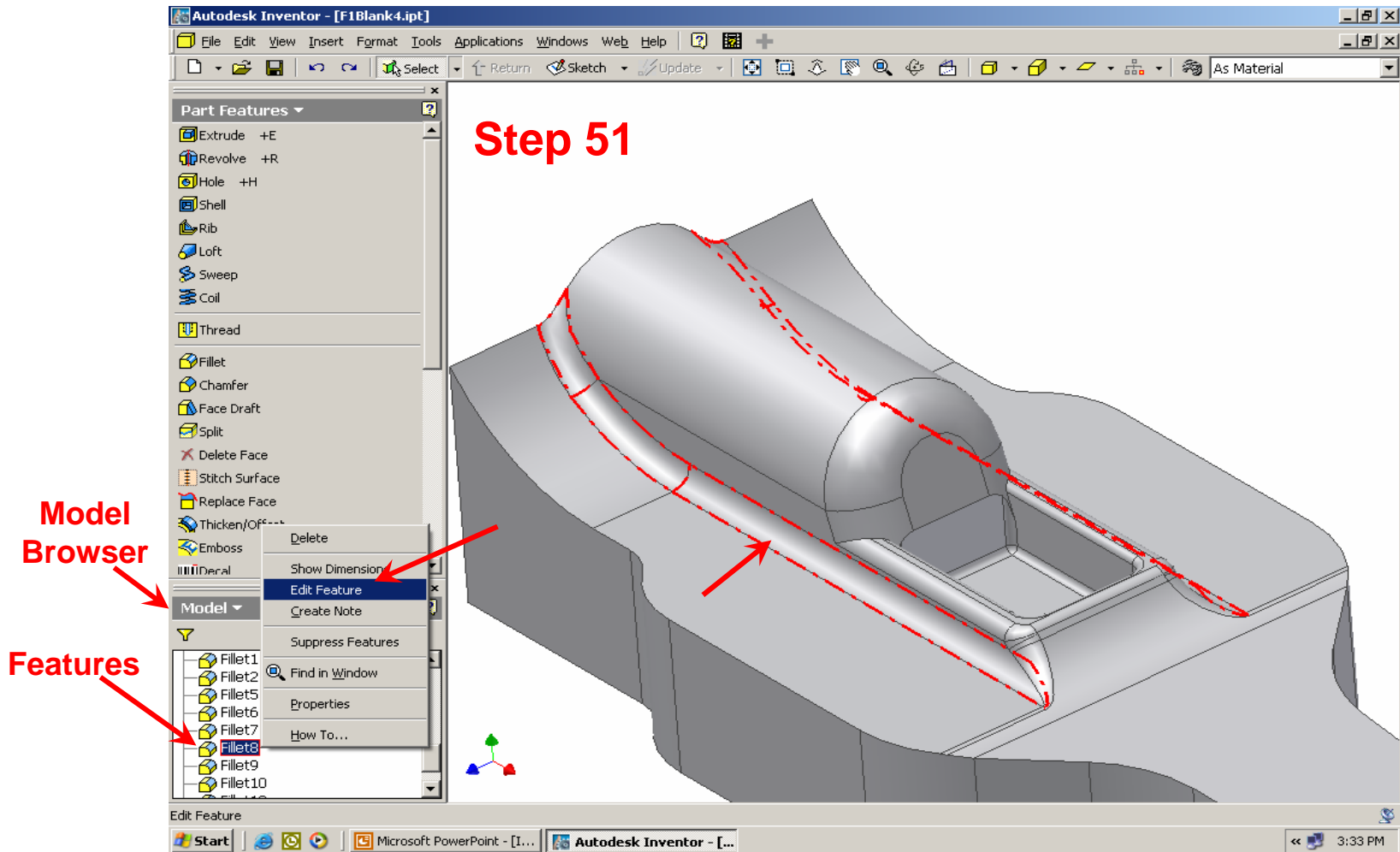




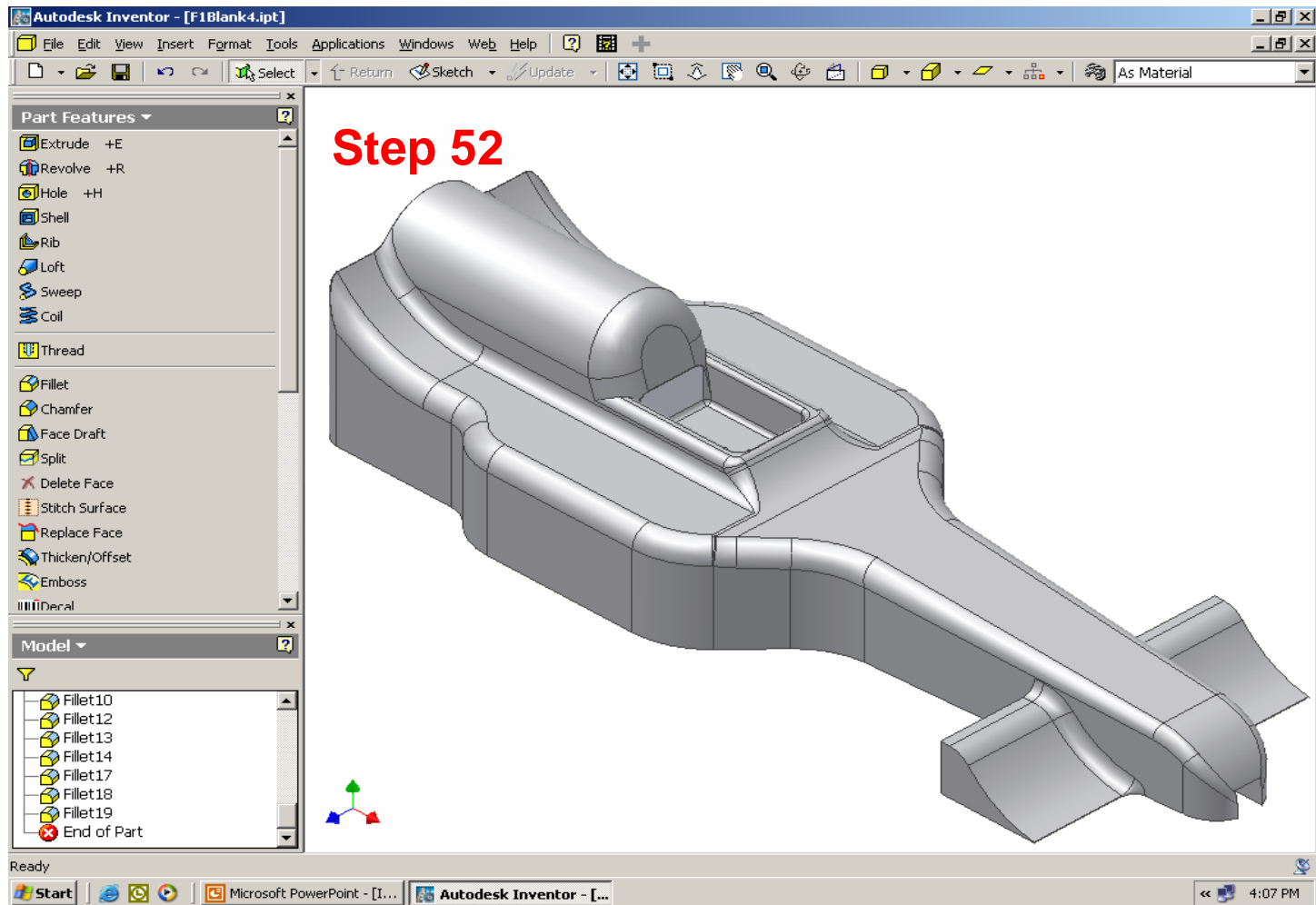
**Step 50.** – Continue filleting the cockpit area. If you select a fillet radius that is too large or can not be successfully cut, the **Edit Fillet Feature** dialog box will open, the problem will highlight in the graphics window, and an explanation of the problem and a suggested solution will be given. In this case, the radius specified was *too large*. (L) click on the **Edit** button to re-open the **Fillet** dialog box and enter a smaller fillet radius.



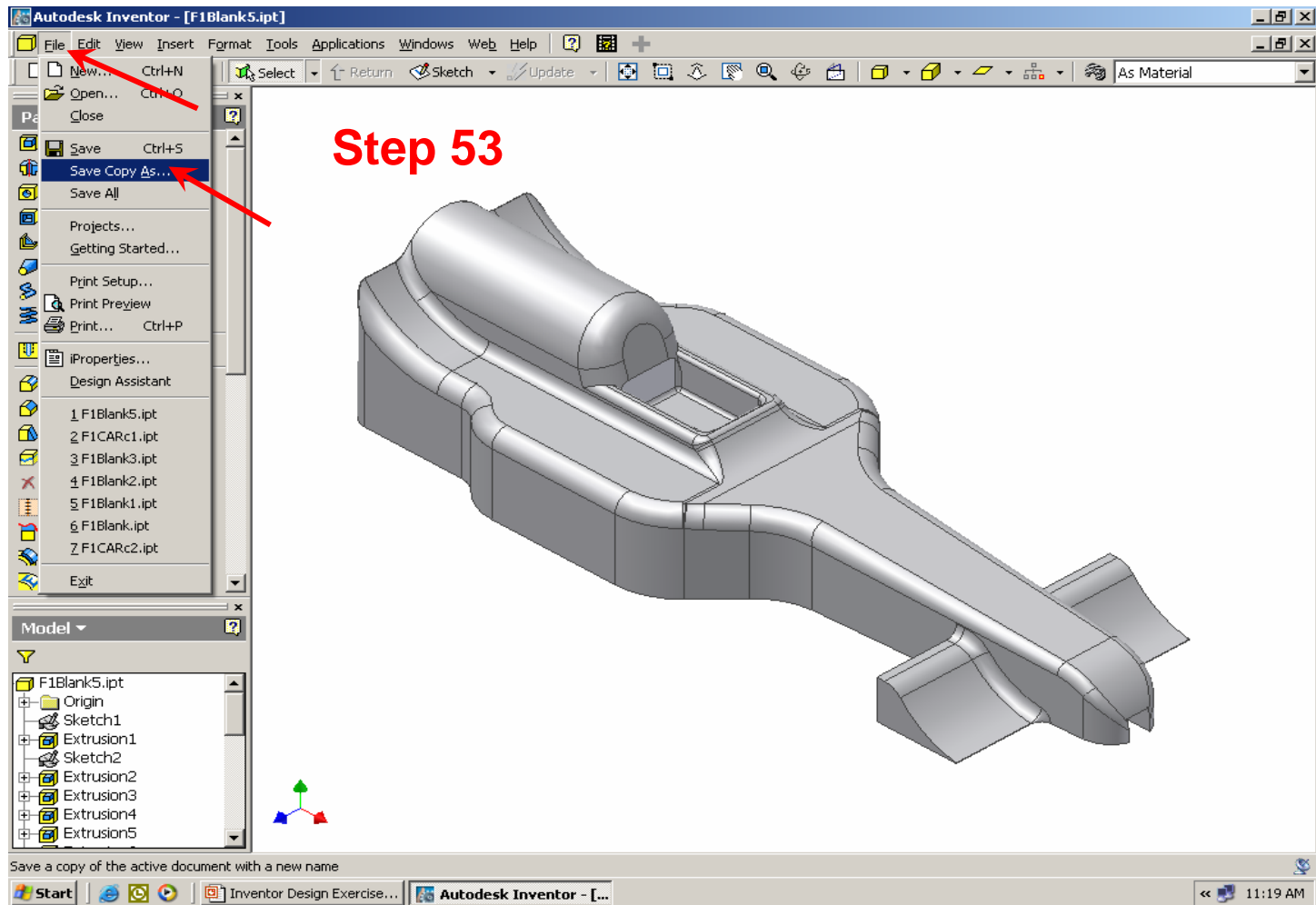
**Step 51.** – **NOTE:** after a feature has been placed, it can be easily edited, at any time, by using the **Model Browser**. Notice that as you move the cursor over each feature in the **Model browser**, that feature will highlight on your model in the graphics window. When the feature you want to edit highlights, (R) click on that feature in the **Model Browser** and then (L) click on **Edit Feature**. The dialog box for that feature will open and the new values can be entered. (L) clicking on the **OK** button will update the model.



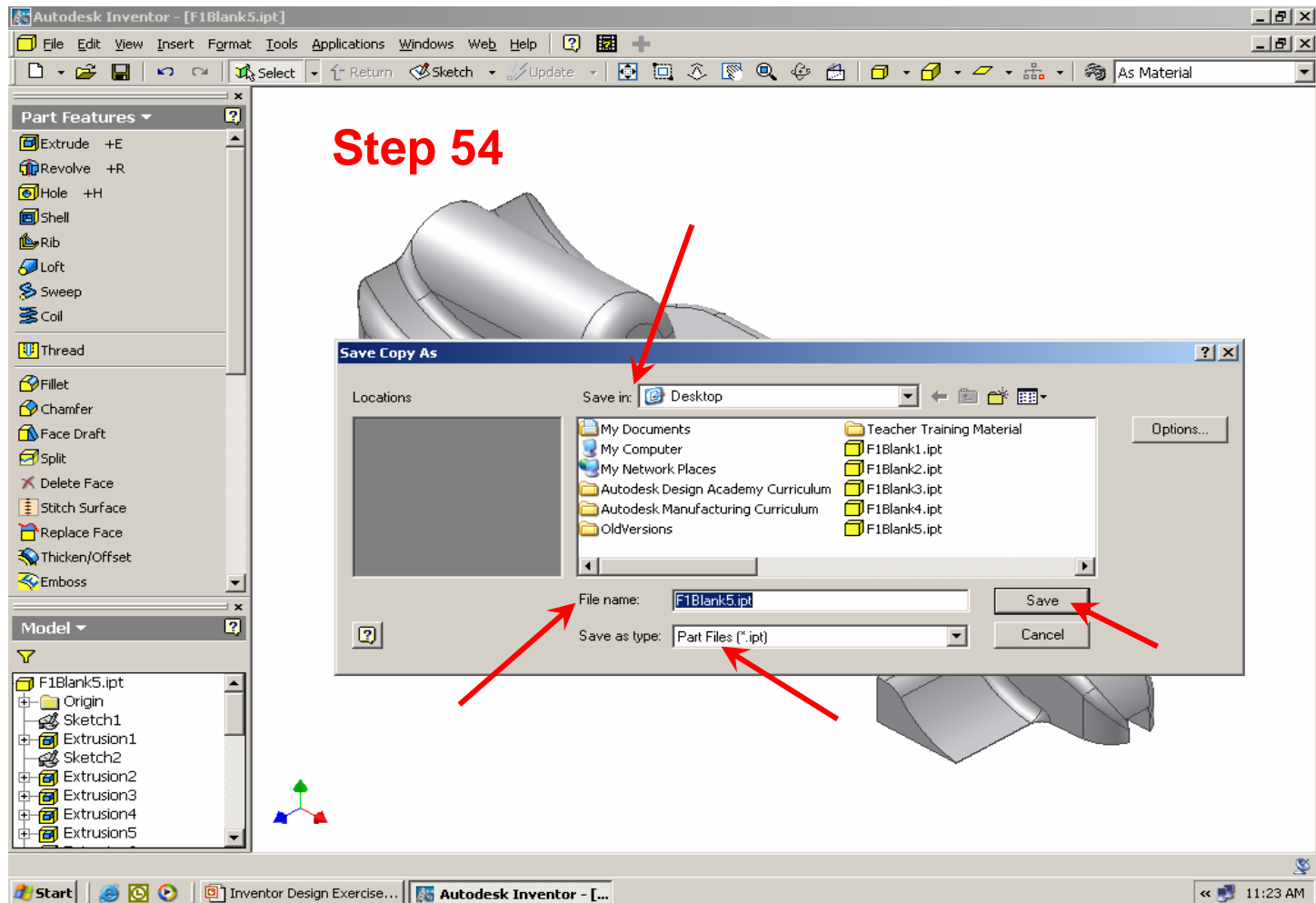
**Step 52.** – Continue using the **Fillet** command until you have a design that you are happy with. Your finished F1 car design should look something like the one below.



**Step 53.** – From the **Standard Toolbar**, (L) click on the **File** command, then (L) click on **Save Copy As**. The **Save Copy As** dialog box will open.



**Step 54.** – In the **Save in:** box, select the folder that you want to save your file in. In the **File name:** box, type in the name you want to use for your F1 model. **NOTE:** that the **Save as type:** will default to **Part Files (\*.ipt)**. (L) click on **Save**.



**Step 55.** – From the **Standard Toolbar (L)** click on the **File** option again, then (L) click on **Save Copy As** to open the **Save Copy As:** dialog box. In the **Save In** box, enter the folder where you want to save your model. In the **File name:** box, type in the same name used in the previous step, then click on the black down arrow to the right of the **Save as type:** box. (L) click on the **STL Files (\*.stl)** option. (L) click on **Save**. The **STL** file format is accepted by all of the popular **CAM** software programs that run **CNC** machine tools. This **.stl** version of your F1 model is what will be used by the **CAM** software to produce your F1 car on a **CNC** mill or router.

