AUTODESK **UNIVERSITY** 2014

Simulation CFD for Newbies

Jim Swain – Synergis Technologies

SM5852-L Have you heard about Simulation CFD software? Do you want to see how easy it is to get started? Would you like to play with a new software system? Come on down to the hands-on lab. Pick which scenario you want to explore, whether it is electronics cooling, airflow in a room, or wind loading on a structure. Choose what type of results you are interested in. Explore and learn.

Learning Objectives

At the end of this class, you will be able to:

- Learn how to bring Inventor software models into Simulation CFD software
- Learn how to set up a simple analysis
- Learn how to run the analysis
- Investigate results and compare different design scenarios

About the Speaker

Jim Swain has over 25 years of CAD experience, including working in the consumer electronics and automotive industries as a design engineer, a test engineer, and a CAD administrator. For the last 17 years he has been a project manager, solutions engineer, and trainer with Synergis Technologies, LLC, an Autodesk Reseller in Pennsylvania. He has also taught college-level design classes and is an Autodesk Inventor Certified Expert, having been in the first group of people to take and pass the Autodesk certification exam at Autodesk University in 2003

jim.swain@synergis.com

This lab is a little different from most that you will see at Autodesk University. In this lab you can pick which of three different scenarios you would like to try. Each looks at different aspects of Simulation CFD.

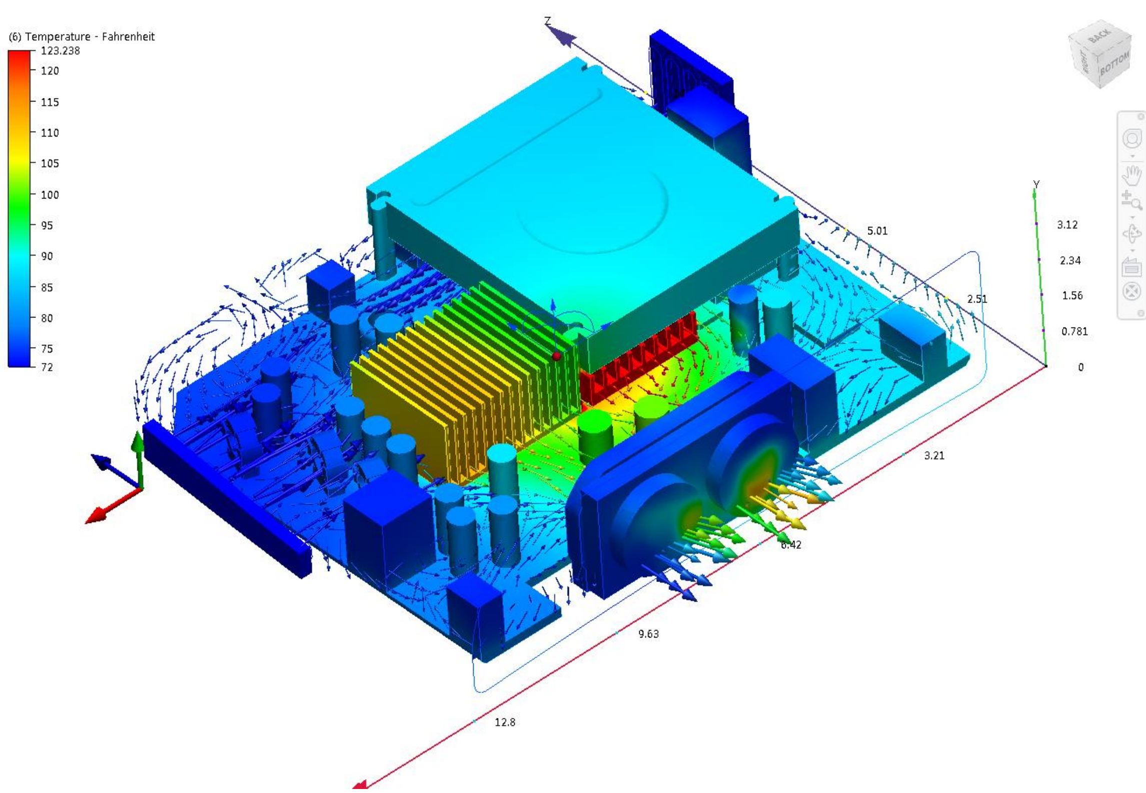
- Scenario 1 – Electronics Cooling
- Scenario 2 – Air Flow in a Room
- Scenario 3 – Wind Loading on a Structure

The scenarios all have a starting Inventor model, a corresponding starting Simulation CFD design study, and a completed run for examining the results.

We will do this lab in the best cooking show tradition:

- You will start the analysis and let it run for a few iterations.
- You will then stop the analysis and open a completed run.
    - This is the part that takes place when the cooking show goes to a commercial break.
- You will then examine the results in completed run file.

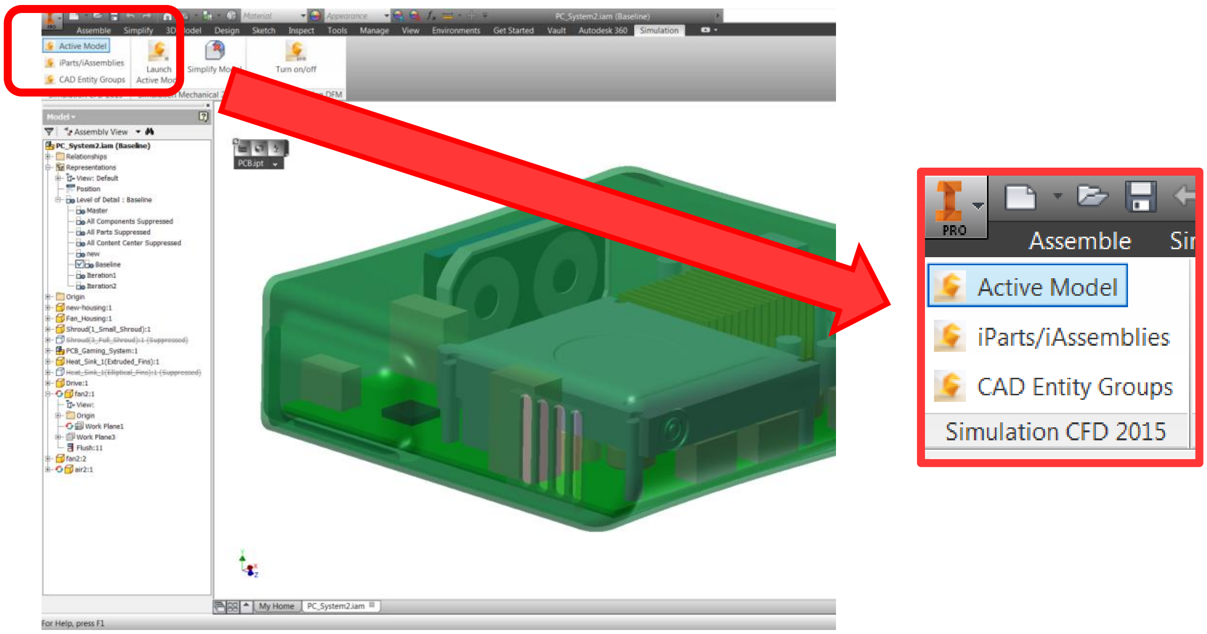
This introduction will outline the basic tools and workflow for Simulation CFD. The instructions for each scenario will then go through the specific picks and clicks.



Basic Workflow for Simulation CFD

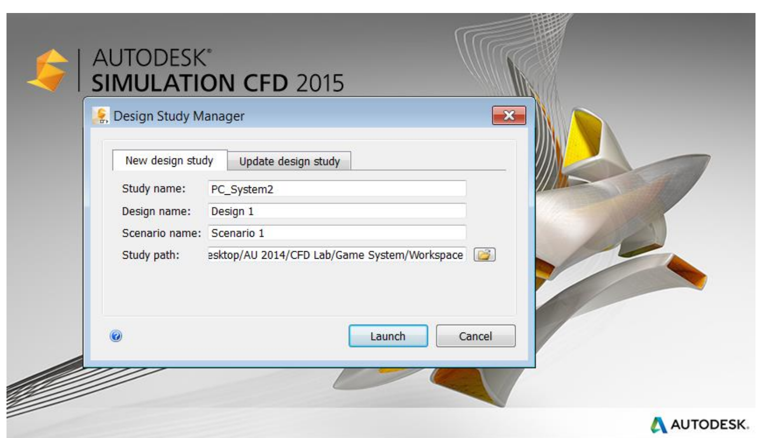
- **Learn how to bring Inventor software models into Simulation CFD software**

The geometry for an analysis can be brought over directly from Inventor. For these lab scenarios we will use the **Active Model** choice.

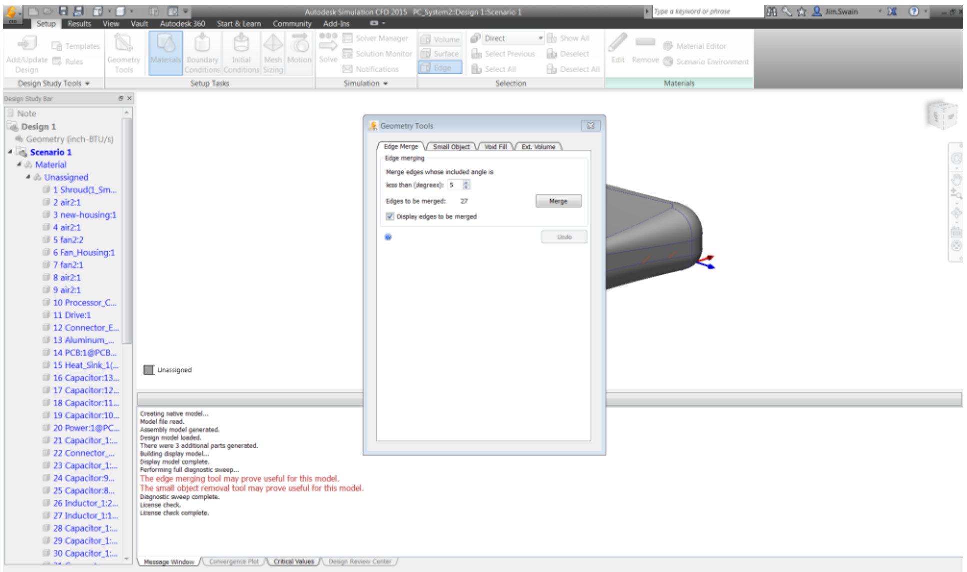


This tool will start the Simulation CFD software and request a name for the simulation's Design and Scenario. The Study name is already the same as the Inventor file that the study was launched from.

The **Launch** button will then begin bringing the geometry into Simulation CFD.



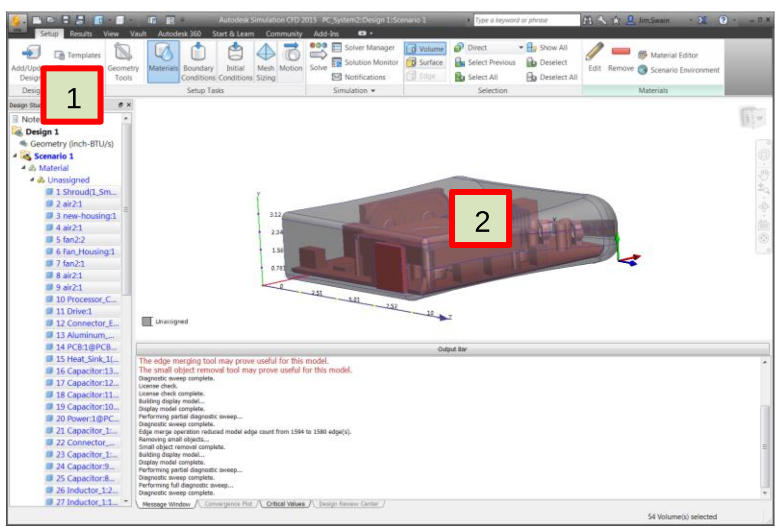
Once the Inventor geometry is brought into Simulation CFD the **Geometry Tools** is automatically started. These tools can also be started again later if this dialog box is closed.



- ## Learn how to set up a simple analysis

Basic Screen Elements – Setup

The Simulation CFD user interface is very similar to other Autodesk software, including Inventor.



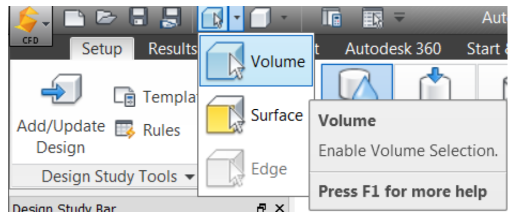
1. Instead of Inventor's Browser we have the Design Study Bar, where the elements of the analysis are listed.
2. There is also the Output Bar. This displays messages from the software, such as the progress of an analysis. Clicking on the Output Bar's title bar will minimize it in the window. Clicking on it again will return it to its previous state.

View Controls

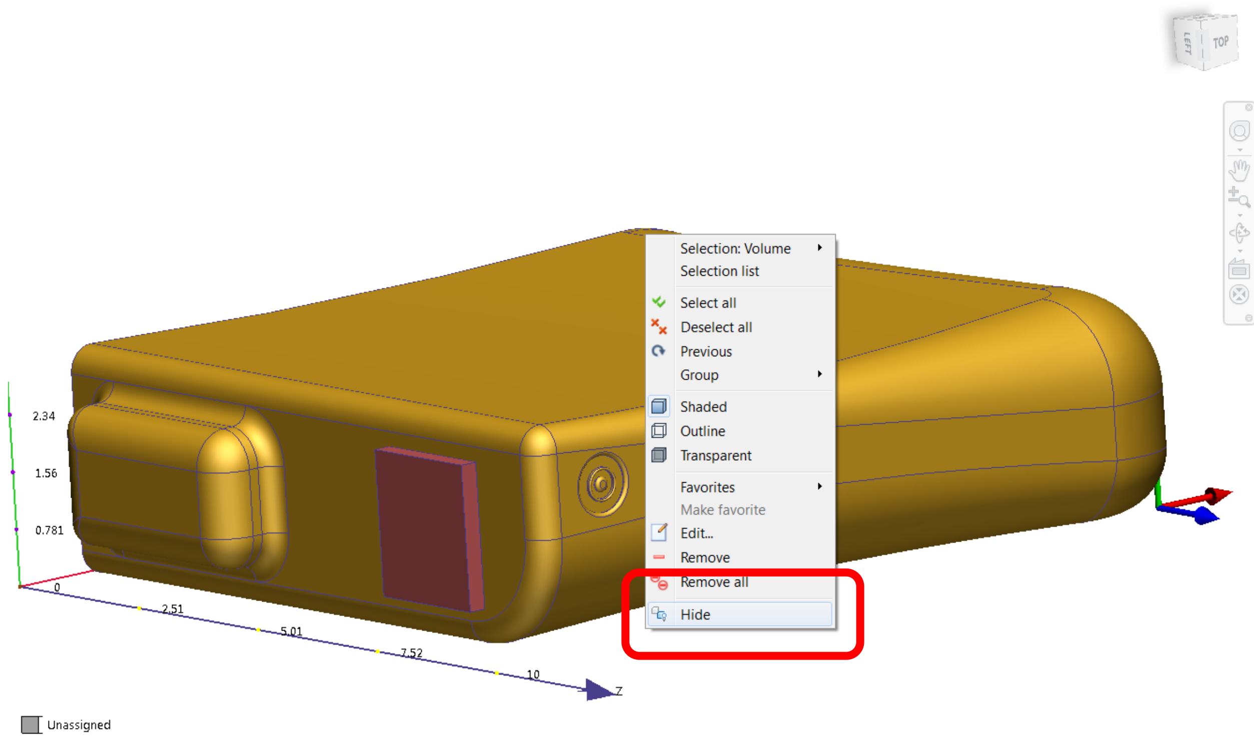
While simulation CFD has dedicated viewing controls, and can be set to mimic the controls in various CAD packages. By default, the View box behaves the same in Simulation CFD as it does in Inventor. You can also use the mouse wheel in the same as in Inventor.

Hiding and Showing Objects

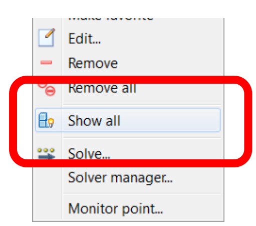
A challenge for new users is often how to change the display of objects in the graphics window. Let's step through this for hiding some objects.



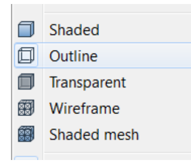
- Make sure the selection mode is set to **Volume**.
- Click on an object, right click on and select **Hide**.



- To display already hidden objects there are two choices:
  - Right click in an empty part of the graphics window and select **Show All**.
  - Hold down the Control key and roll the mouse wheel. This will scroll through the hidden object display list.



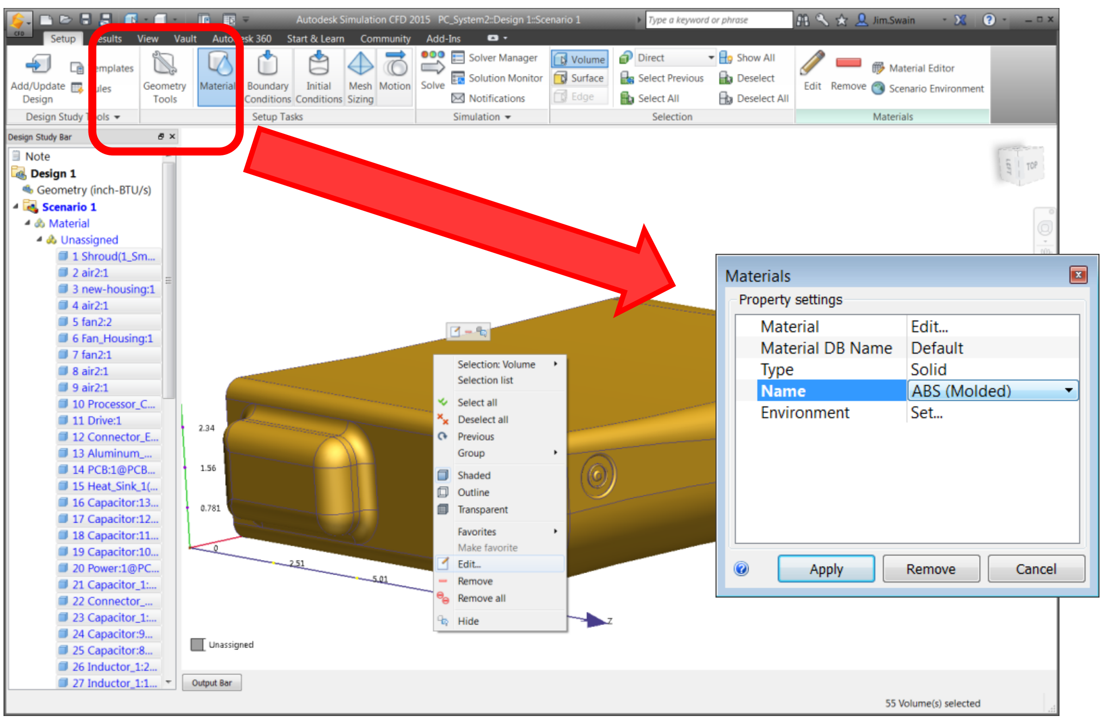
- Individual volumes can also be selected, then have their display style set to Shaded, Outline, Transparent, Wireframe, or Shaded Mesh.



Assigning Materials

Components come into Simulation CFD with no assign material properties. These can be assigned individually by picking an object and editing its material or by using rules to automatically assign materials to objects based on criteria. Since there are a large number of objects in the electronics enclosure scenario only a few will be assigned materials manually. The rest will be assigned via rules.

Choose the general category of material, such as solid or liquid, then choose the specific material in that category.



A useful technique when manually applying materials is to apply the material, then hide that object. This makes it easy to pick the next object on the screen.

Assigning Boundary Conditions

Boundary conditions are added the same way as material conditions, but are often added to surfaces instead of volumes. Therefore, be sure to double check the selection mode before adding boundary conditions.

Assigning Initial Conditions

Initial conditions are used in transient analyses to assign starting conditions, such as the initial temperature of an object. For this lab we won't be assigning Initial Conditions as all analyses will be solved for steady state conditions.

Assigning Mesh Sizing

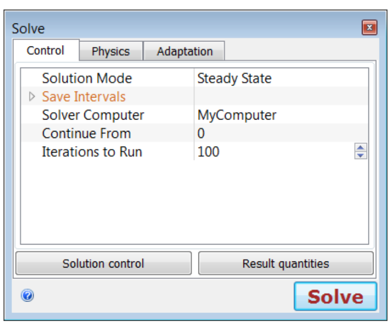
The mesh sizing can be adjusted in areas of interest or areas that have relatively large or small geometric features. For the scenarios in this lab we won't be adjusting the mesh sizing.

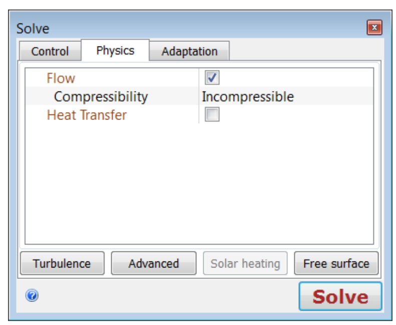
- ## Run the analysis

Once the setup tasks are completed click the Solve tool button to set the physics of the analysis. The choices include whether heat transfer or only fluid flow are being considered, and if transient conditions or only steady state results are of interest. After the choices are made, click the Solve button to begin the analysis itself.

Once the analysis is running the Solve button will be replaced by a Stop button. Clicking on the Stop button will stop the analysis once it has finished calculating the results for that iteration.

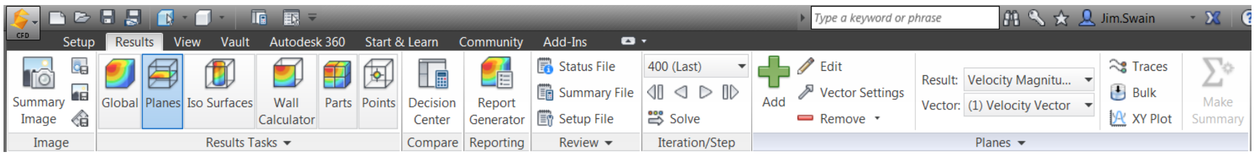
A new analysis will start at Iteration 0. An analysis can be stopped at any time, and restarted. For example: this may be done to turn on heat transfer once a forced convection flow pattern has been established.





- **Investigate results and compare different design scenarios**

After one iteration has completed you can go to the Results tab and begin reviewing the analysis results. (Don't expect a lot from Iteration #1.)



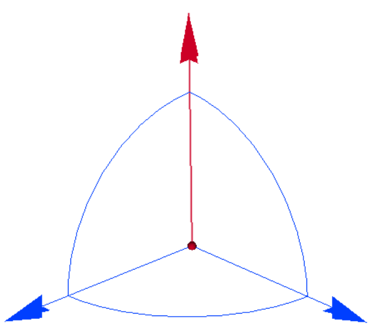
While there isn't space to show all of the methods of investigating the results of an analysis here are some of the major tools:

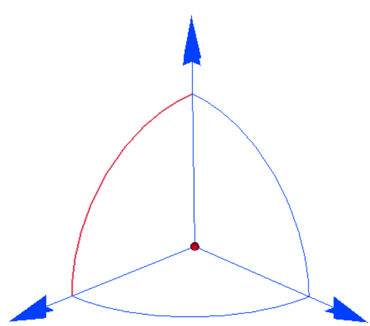
Global

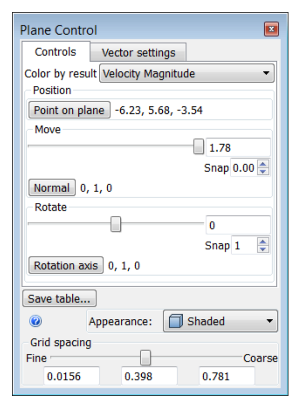
The Global results are always shown. Typically they will show temperatures or velocities. Global vectors can also be displayed. These are useful for showing the fluid velocities and directions within the analysis.

Planes

Information can be shown on one or more planes slicing through the analysis. Planes can be added, then repositioned and reoriented using the coordinate system handles. A planes final position can be tweaked using the Edit tool.

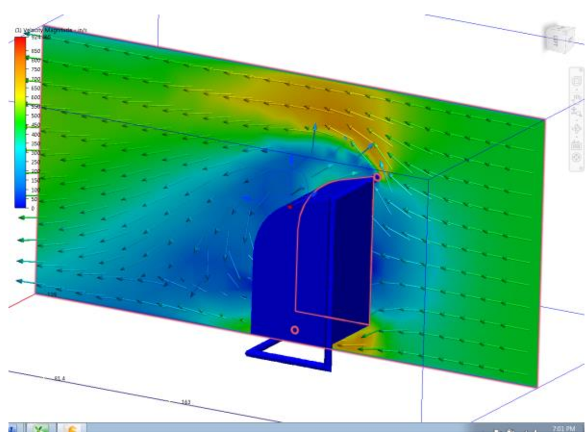


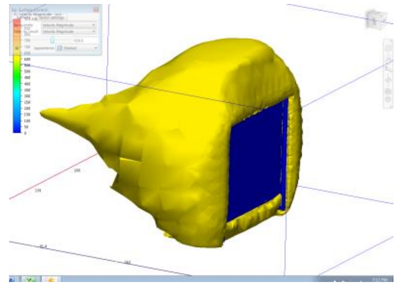




In addition to showing information via color a plane can also display vectors. This is very useful for showing the out of plane velocities through an analysis.

Hint: Be sure that the fluid volume isn't hidden when showing velocities. It is often helpful to turn the display of the fluid volume to **Outline**.



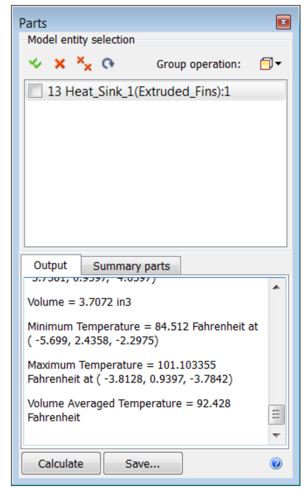


Iso Surfaces

Iso surfaces will show all surfaces that have the same property, such as all areas that have the same velocity in a fluid.

Wall Calculator

The wall calculator tool can be used to find heat fluxes, forces, pressures, etc., acting on a single face or an entire volume.



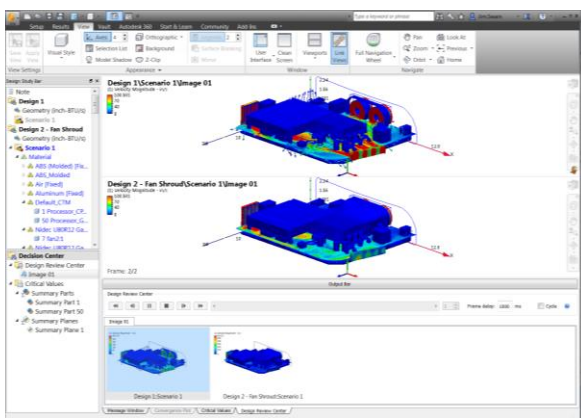
Parts

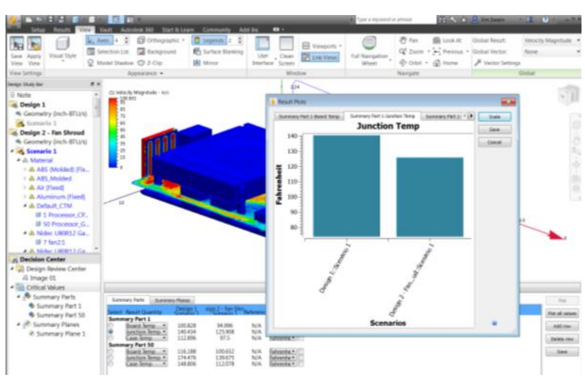
The Parts tool is useful for showing specific information about a part, such as its temperature.

Decision Center

Decision Center is a separate browser window that can be used to compare information from various designs and scenarios within a single analysis.

Planes, parts and views can be marked as summary items. The results for the summary items can then be shown across the designs and scenarios in the study.





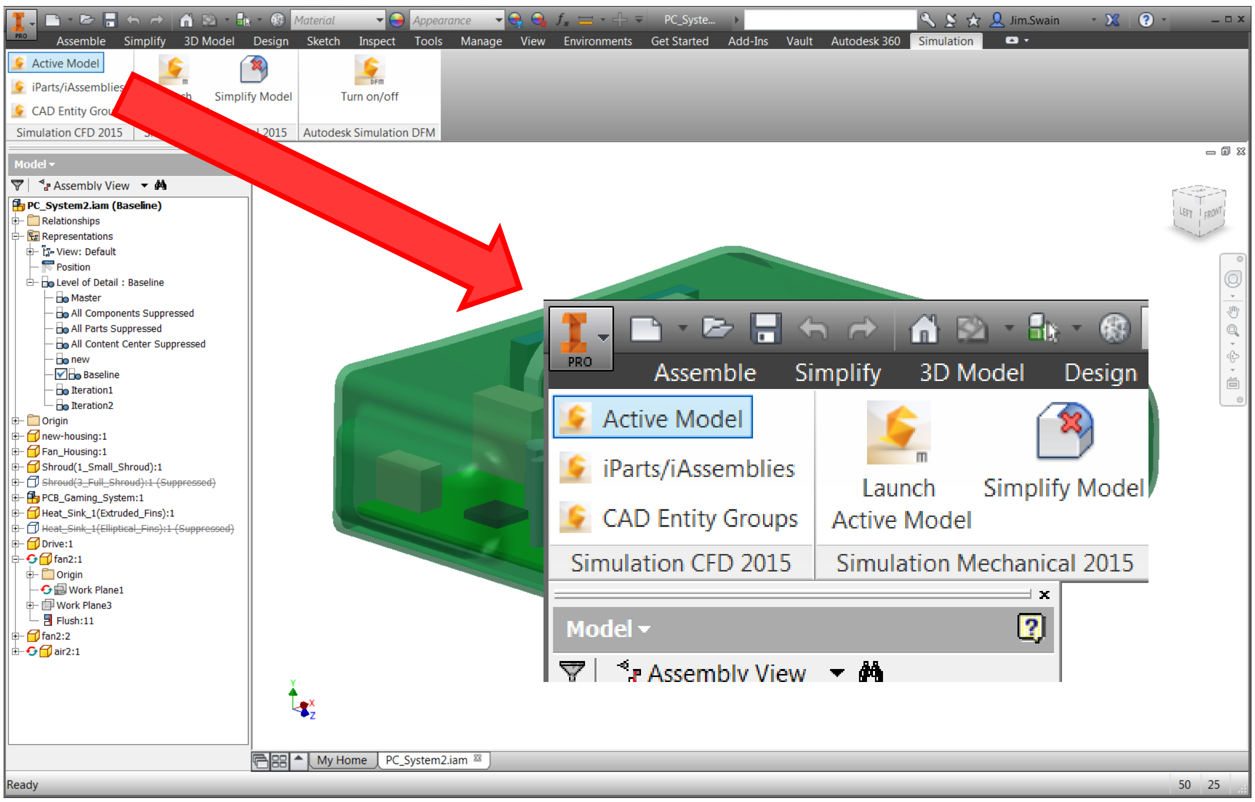
Ok, now for the fun stuff…

- **Scenario 1 – Electronics Cooling**

Open the Inventor File

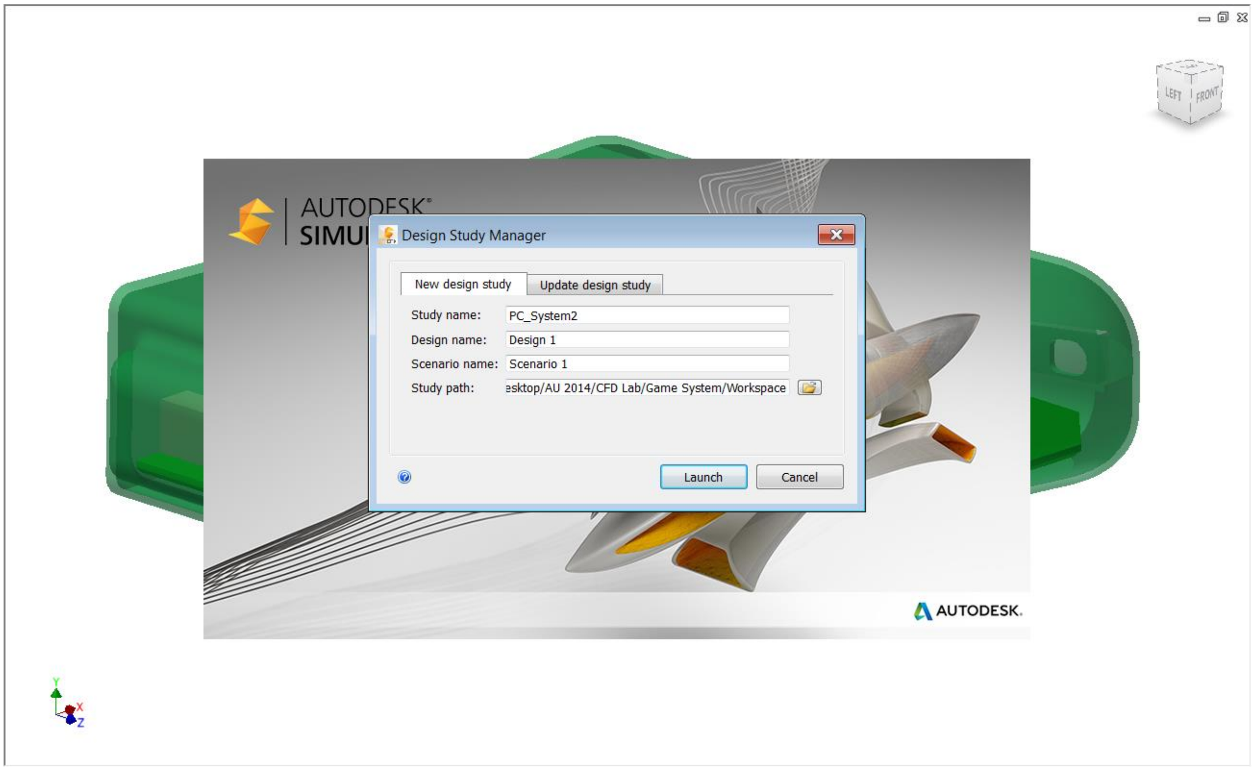
1. Start Inventor and check that the project file is set to **SM5852-L.ipj**.
2. Open the **Electronics Enclosure.iam** file.

Send to Simulation CFD

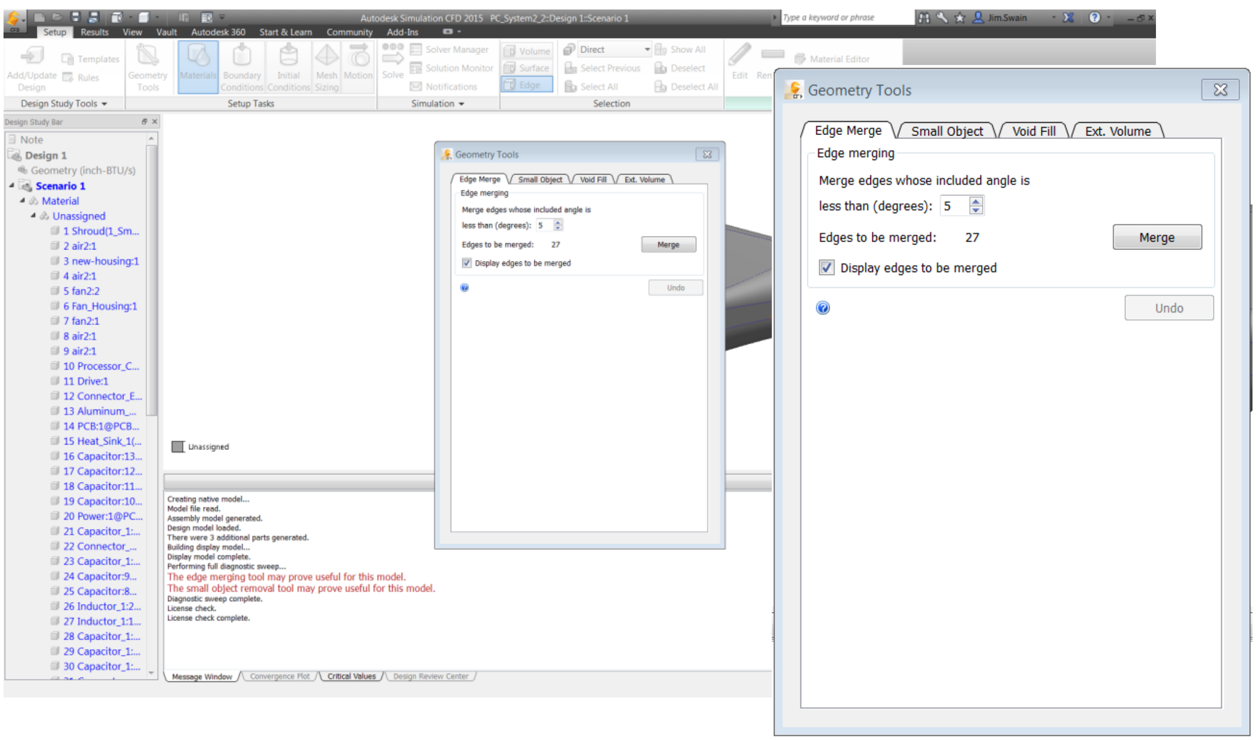


3. Choose the **Active Model** tool from the **Simulation CFD 2015** panel of the **Simulation** tab.
4. Click **OK** in the Save dialog box.

Apply Geometric Tools



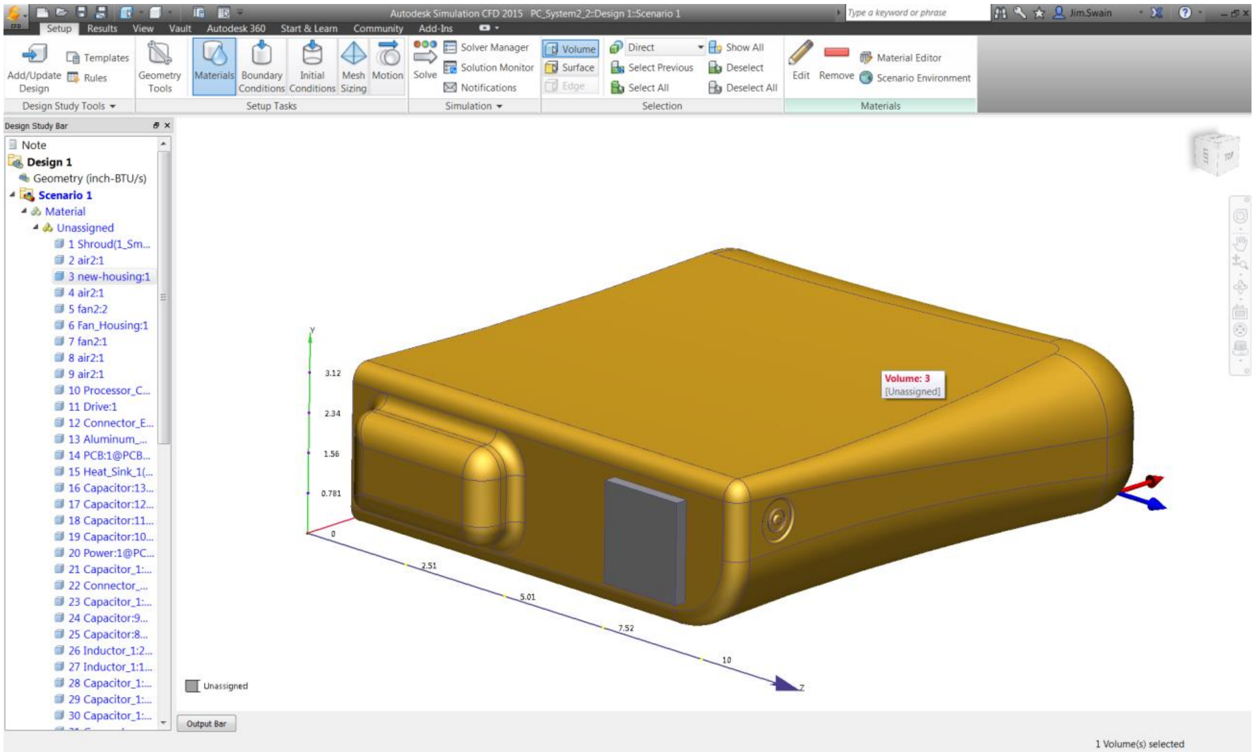
5. Click in the **Launch** button on the Design Study Manager dialog box. This will create a new design study with the default name.



6. Select **Merge** in the Edge Merge tools tab of the Geometry Tools dialog box.
7. Select Remove in the Small Object tools tab.
8. Close the Geometry Tools dialog box.

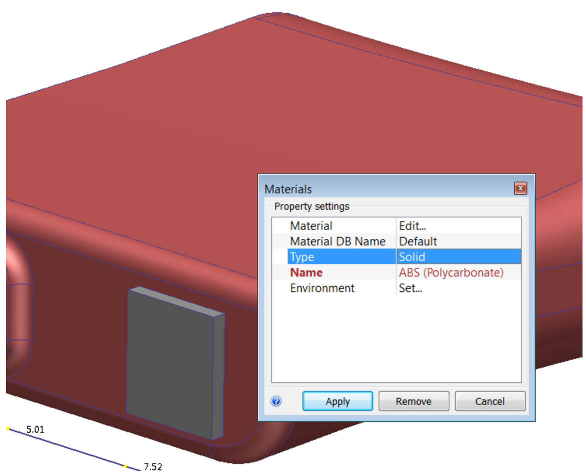
Assign Materials

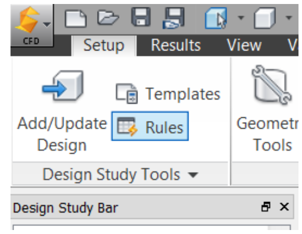
The **Material** tool is automatically activated once the **Geometry Tools** dialog box is closed. Currently none of the CAD parts have any assigned materials.



9. Click on the outer case of the model. Pick the **Edit** tool from the Ribbon, the in-canvas menu, or the right click pop up menu. Change the material to **Solid** and **ABS (Polycarbonate)** and **Apply**.

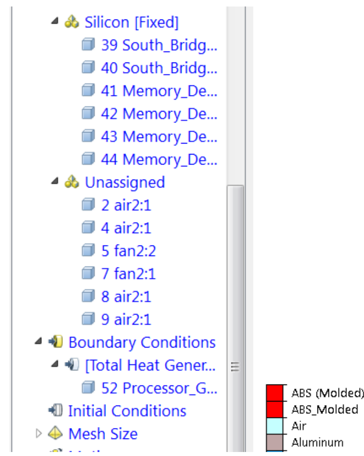
10. Right click again on the case and choose **Hide**.





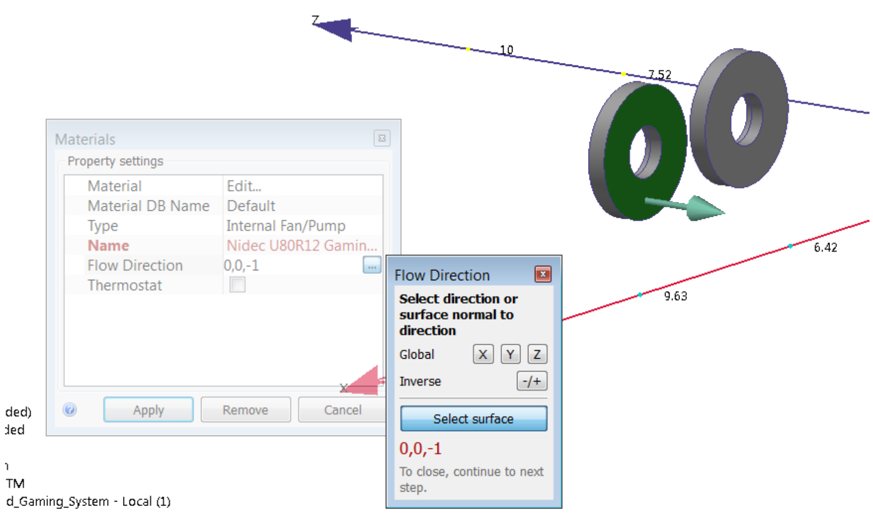
Apply Rules

11. To speed material assignment, choose the **Rules** tool from the Design Study Tools panel.
12. Use the **Import rules…** tool to read the rules from the **SM5852-L.usr** file in the lab's top level folder.
13. Choose the **Select all** button, then the **Apply now** button in the Rule Manager dialog box.
14. Close the Rule Manager dialog box.



Several volumes are still not assigned a material.

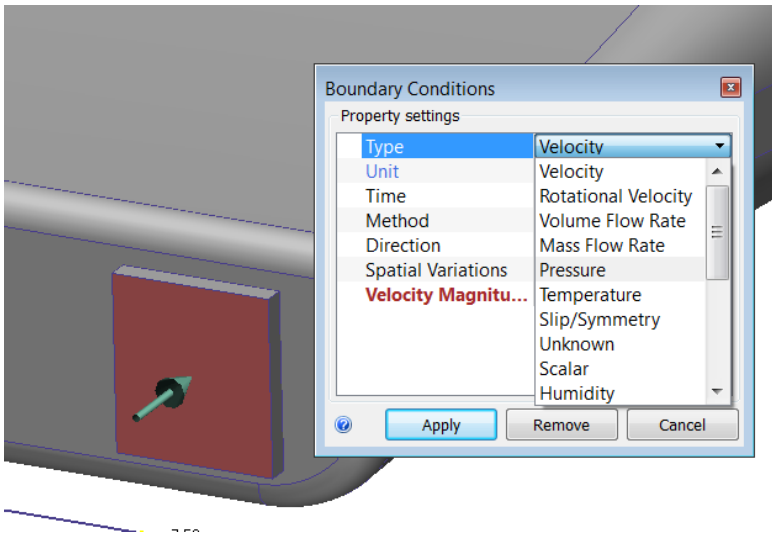
15. Highlight the volumes named air in the Design Study Bar and change their material to **Fluid**, **Air**.
16. Highlight the volumes named fan and change their material to **Internal Fan/Pump**, **Nidec U80R12 Gaming System**.
17. Click the ellipses button next to the **Flow Direction** field, then the **Select surface** button. Choose the outside facing face of the ring that represents the fan volume. This will set the flow direction to 0,0,-1. Then pick **Apply**.



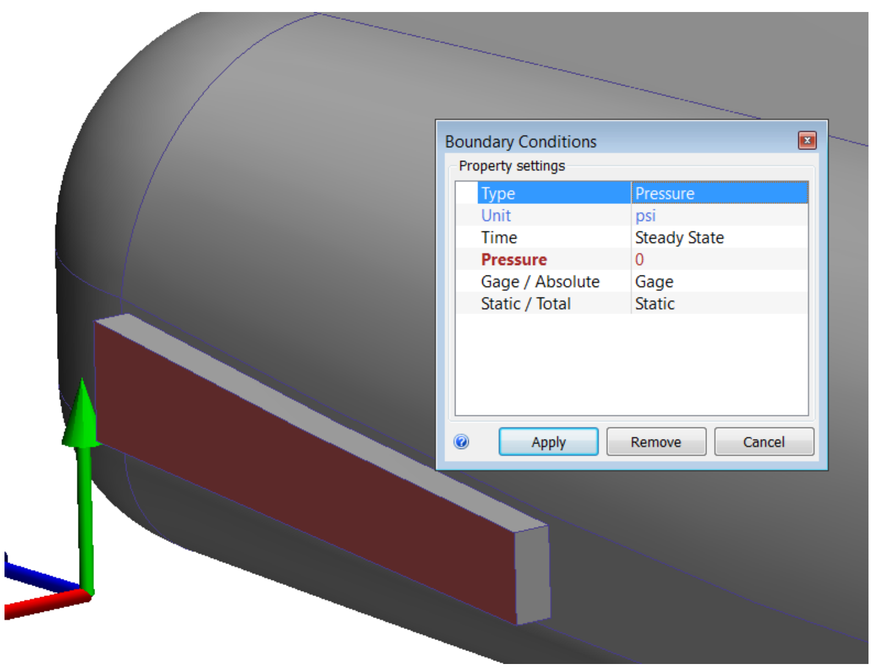
Assign Boundary Conditions

The inlet and outlet portions of the model need to have their surface boundary conditions set to a static pressure of 0 psi gauge.

18. Start the **Boundary Conditions** tool on the Setup Tasks panel.



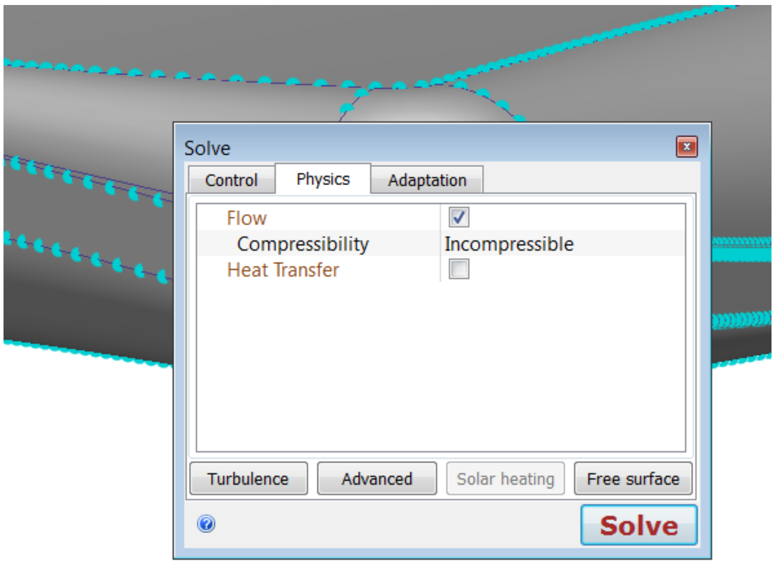
19. Select the outer surfaces of the air inlet and set the **Pressure** to **0 Gage**.
20. Set the outer surfaces of the inlet on the opposite face of the enclosure.



This is only a flow analysis. For a full thermal analysis, the heat generation of the other electronic components would be added and the air temperature of the incoming air would be specified.

Note: Don't run the Mesh Sizing tools on this model! The existing mesh settings allow the heat sink fins to mesh.

Run the Analysis



21. Pick the **Solve** tool, make sure only **Flow** is selected, and press the **Solve** button.
22. Once the analysis has started, and the Results ribbon tab is active, right click on the overall volume and set its display style to **Outline**.
23. Go back to the Solve dialog box and **Stop** the analysis.

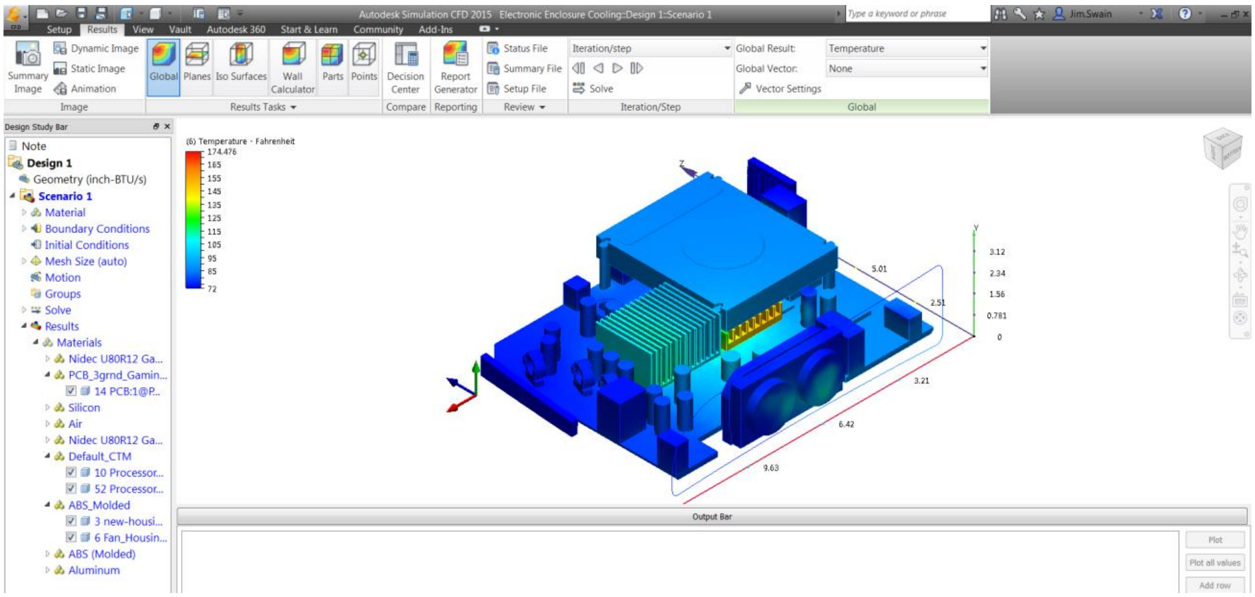
Once one iteration of the analysis has been completed you can start investigating the results. For the next section, you will open a study where the analysis has successfully completed.

Investigate the Results

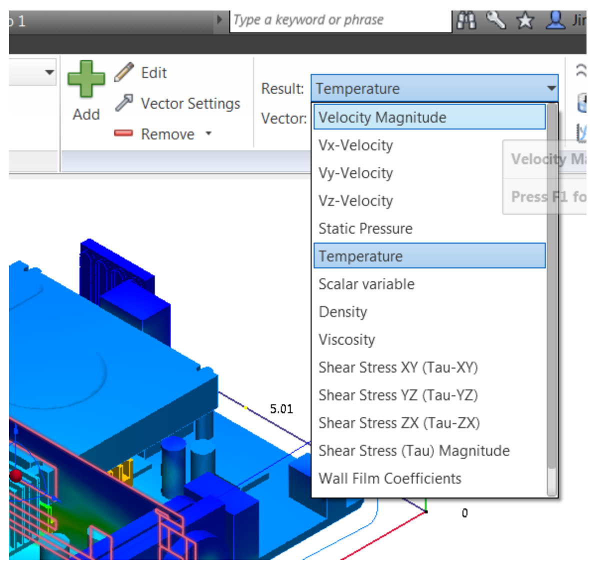
Open the Results File

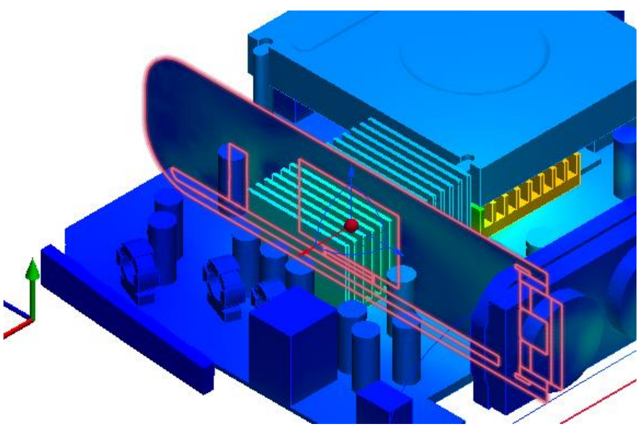
24. Use the **Open** command from the Application Menu, and open the **Electronic Enclosure Cooling_support.cfz** file. This will bring in results from a completed analysis.
25. Select the Results tab, and change the **Global** results to **Temperature**.
26. Right click on the main air volume and **Hide** it.
27. Hide the hard drive and the heat sink under it to see the temperature of the CPU.

28. Hold down the Control key and scroll the mouse wheel to unhide the heat sink, the hard drive and the air volume.
29. Right click on the main air volume and change its display to **Outline**.



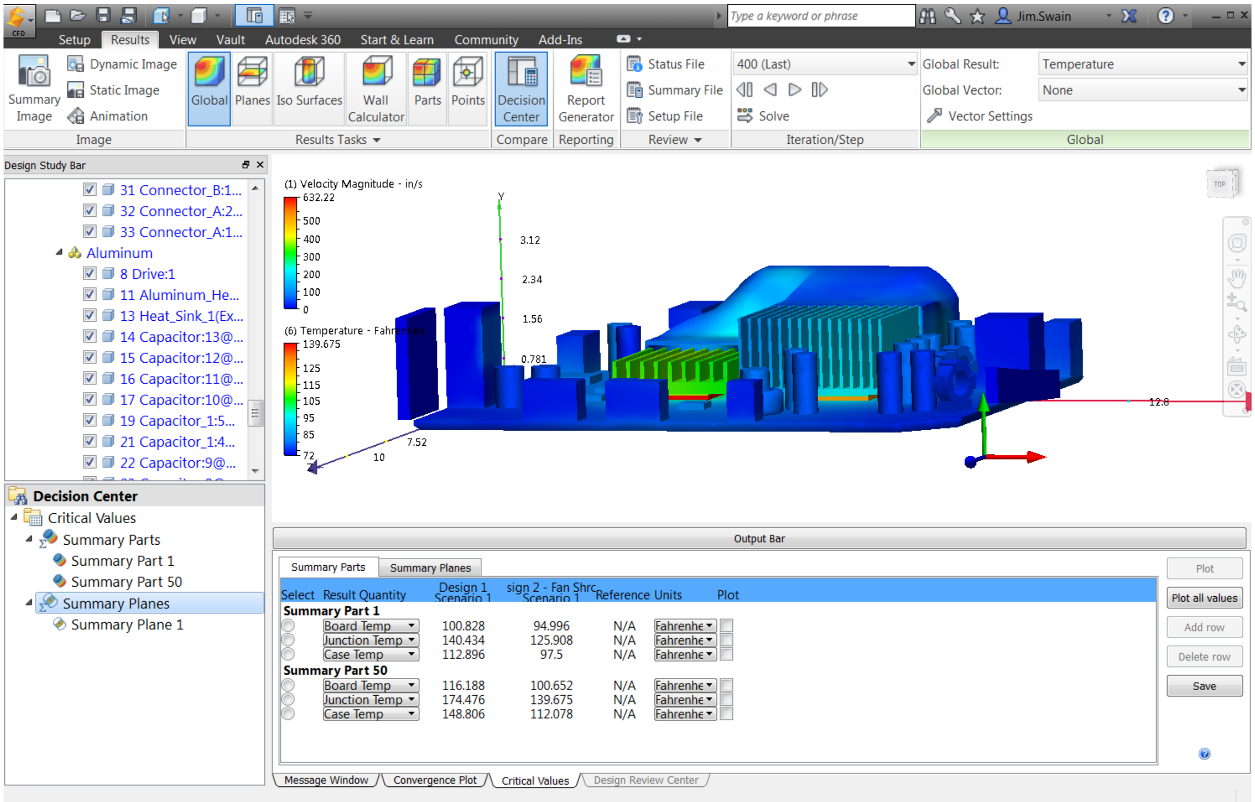
30. Select the **Planes** tool and click the **Add** button.
31. Change the display results for the plane to Velocity Magnitude.
32. Move and rotate the plane to explore the air flow around the fins.



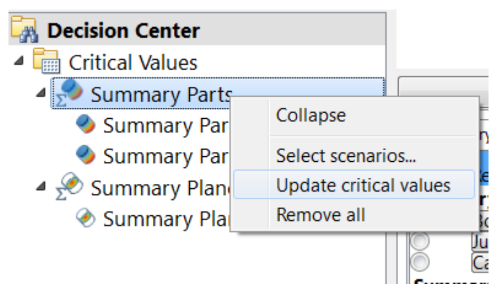


Notice how the air is bypassing the larger cooling fins. Next we will take a look at a design with a fan shroud directing airflow through the heat sink's fins.

Compare Results



33. Turn on the **Decision Center**.
34. Two parts are listed as Summary Parts, the CPU and the GPU. Right click on the Summary Parts header and pick **Update critical values**.
35. Since these are defined as Compact Thermal Model material parts the board, junction, and case temperatures are automatically calculated.



Notice how the temperatures are about 15° F lower for the revised design.

<u>End of the Electronics Cooling scenario.</u>

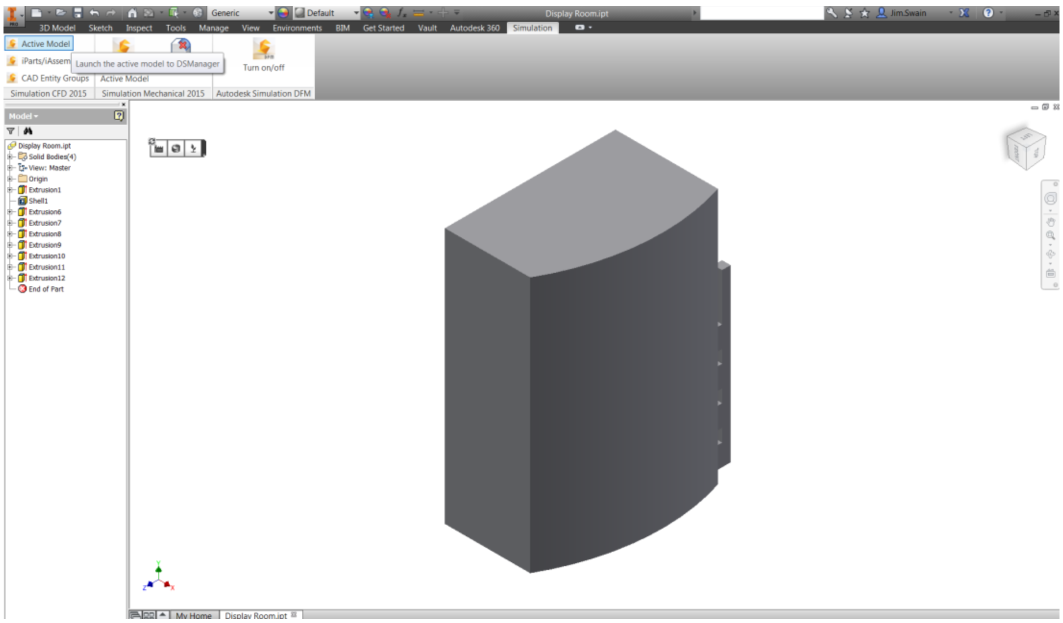
- ## Scenario 2 – Air Flow in a Room

Open the Inventor File

1. Start Inventor and check that the project file is set to **SM5852-L.ipj**.
2. Open the **Display Room.ipt** file.

Send to Simulation CFD

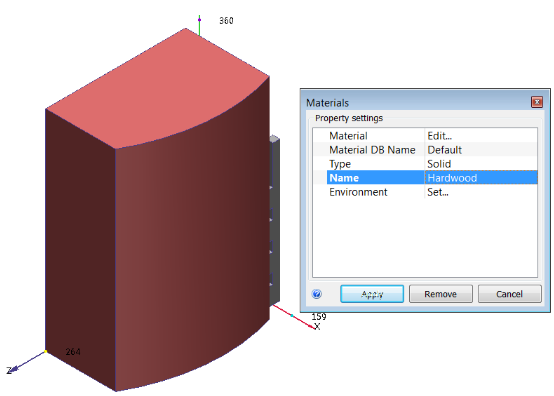
3. Choose the **Active Model** tool from the **Simulation CFD 2015** panel of the **Simulation** tab.
4. Click **OK** in the Save dialog box.



5. Click in the **Launch** button on the Design Study Manager dialog box. This will create a new design study with the default name.

Setup the Analysis

Note: it isn't necessary to apply any of the Geometry Tools to this model to get it ready for analysis.



Assign Materials

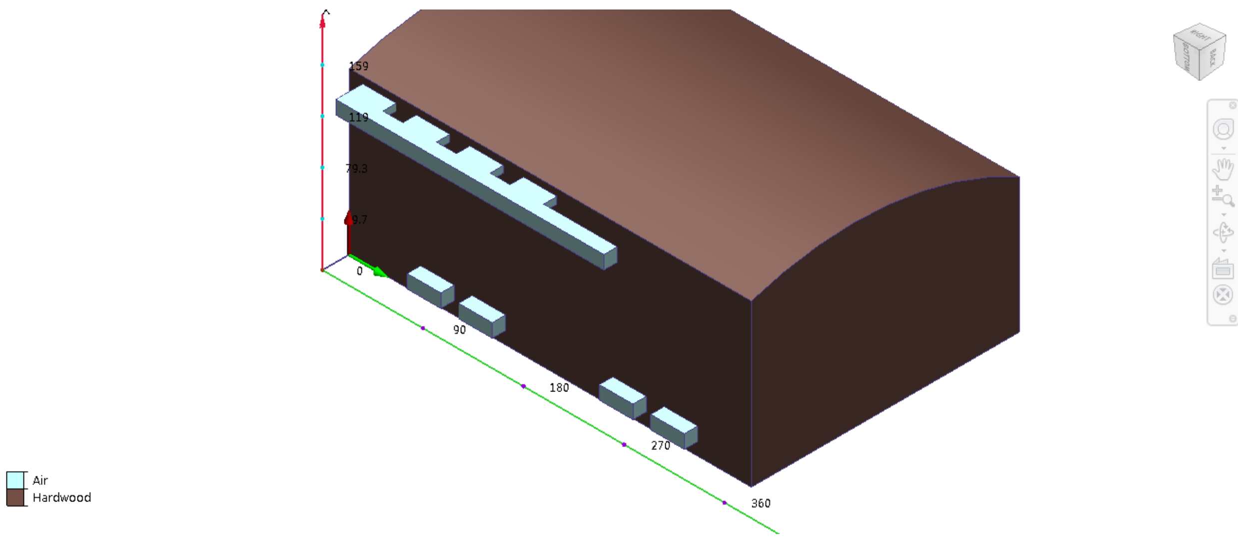
6. Click on the larger body of the model. Pick the **Edit** tool from the Ribbon, the in-canvas menu, or the right click pop up menu. Change the material to **Solid** and

Hardwood, and then click **Apply**.

7. Right click again on the same body and choose **Hide**. This will hide the exterior volume shell.

8. Repeat steps 6 and 7 for remaining volumes in the model. Change all these volumes to **Fluid** and **Air**.

9. Use the View Cube to change the view orientation to match the image shown here.

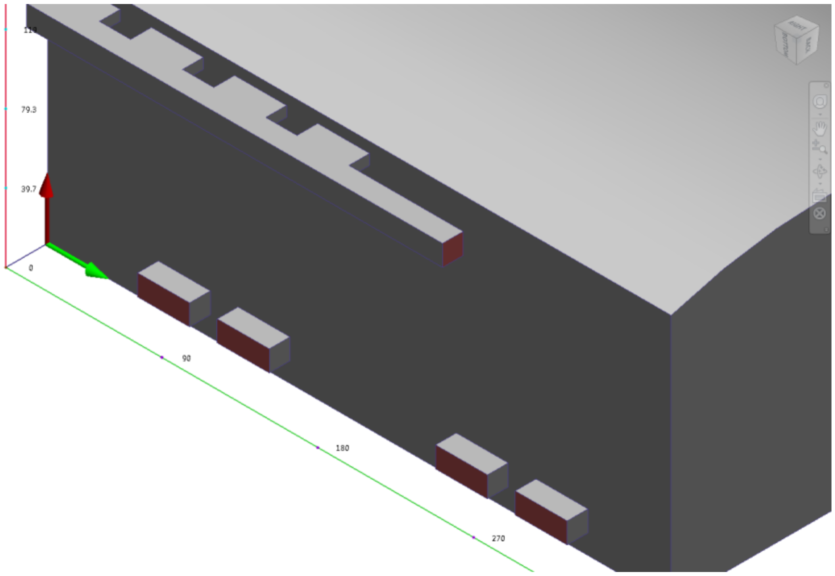


Assign Boundary Conditions

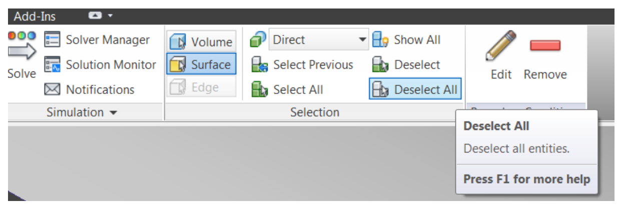
The inlet and outlet portions of the model need to have their surface boundary conditions set to a static pressure of 0 psi gauge.

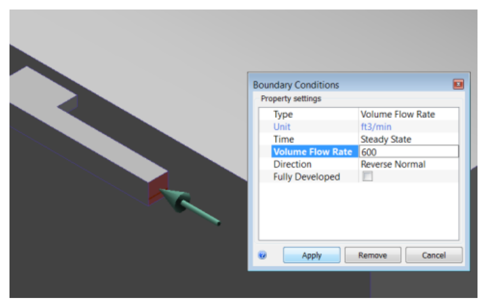
10. Right click in the graphics window and pick **Show all**.
11. Start the **Boundary Conditions** tool on the Setup Tasks panel.
12. Select the 5 faces shown and **Edit** their boundary conditions to have a **Pressure** of **0 Gage**.

Note: You may have to change the Selection mode to Surface in order to pick just those faces.

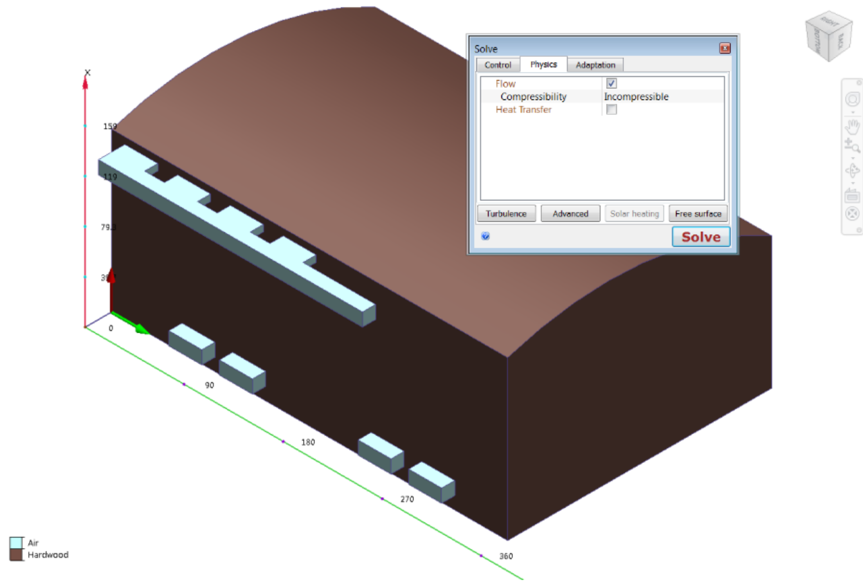


13. Pick the **Deselect All** tool from the Selection panel.
14. Select just the end of the volume shown, and Edit its boundary condition to have a **Volumetric Flow Rate** of **300 ft^3/min**. Click **Apply** to accept the edit.



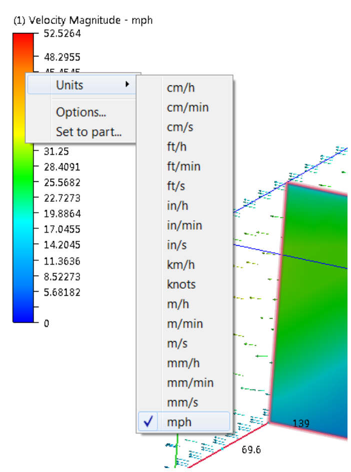


Run the Analysis

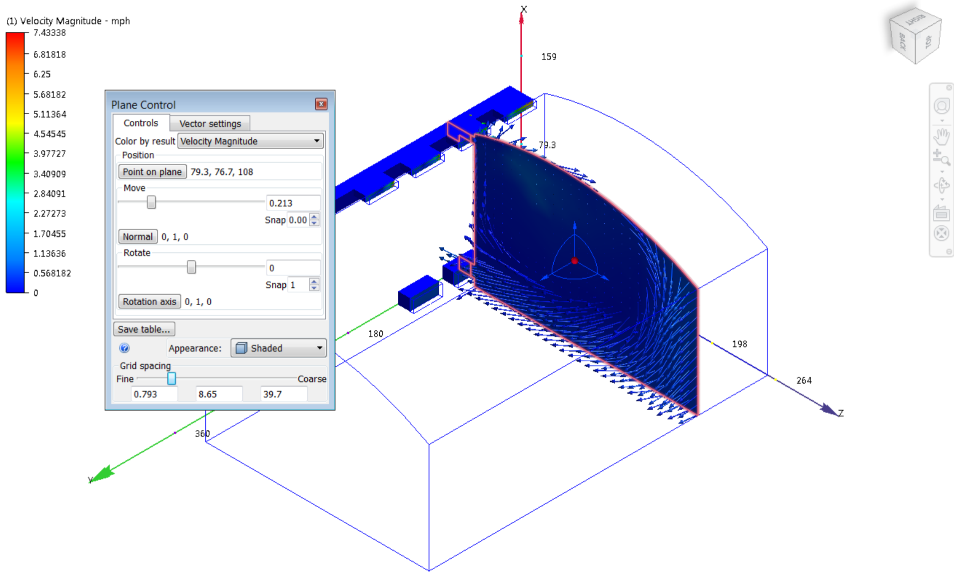


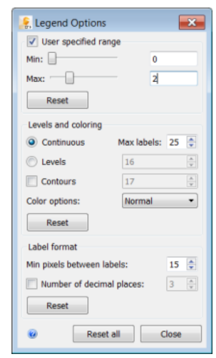
15. Pick the **Solve** tool, make sure only **Flow** is selected, and press the **Solve** button.
16. Once the analysis has started, and the Results ribbon tab is active, and a few iterations have run, right click on the overall volume and **Hide** it.
17. Click again in the same area and turn the display of the other air volume to **Outline**.
18. Go back to the Solve dialog box and **Stop** the analysis.

Open the Results File



19. Use the **Open** command from the Application Menu, and open the **Display Room_support.cfz** file. This will bring in results from a completed analysis.
20. Select the Results tab, and change the **Global** results to **Velocity Magnitude**.
21. Right click on the overall volume and **Hide** it.
22. Click again in the same area and turn the display of the other air volume to **Outline**.
23. Right click on the color bar and set the **Units** to **mph**.
24. Change the Global Vector: to **Velocity Vector**.
25. Select the **Planes** tool and click the **Add** button.
26. Change the display results for the plane to **Velocity Magnitude**.
27. Turn on the display of the **Velocity Vectors**.
28. Move and rotate the plane to explore the air flow in the room.





29. Right click on the color bar and change the Max to 2 mph. This will help see the actual velocity range within the room itself.

Notice how the air is recirculating around the room.

Compare Results

30. Use the **Open** command from the Application Menu, and open the **Room – Both Cases.cfz** file.
31. Double click on **Restricted Duct**.
32. Compare the circulation patterns in the two scenarios by adding planes to each as above.

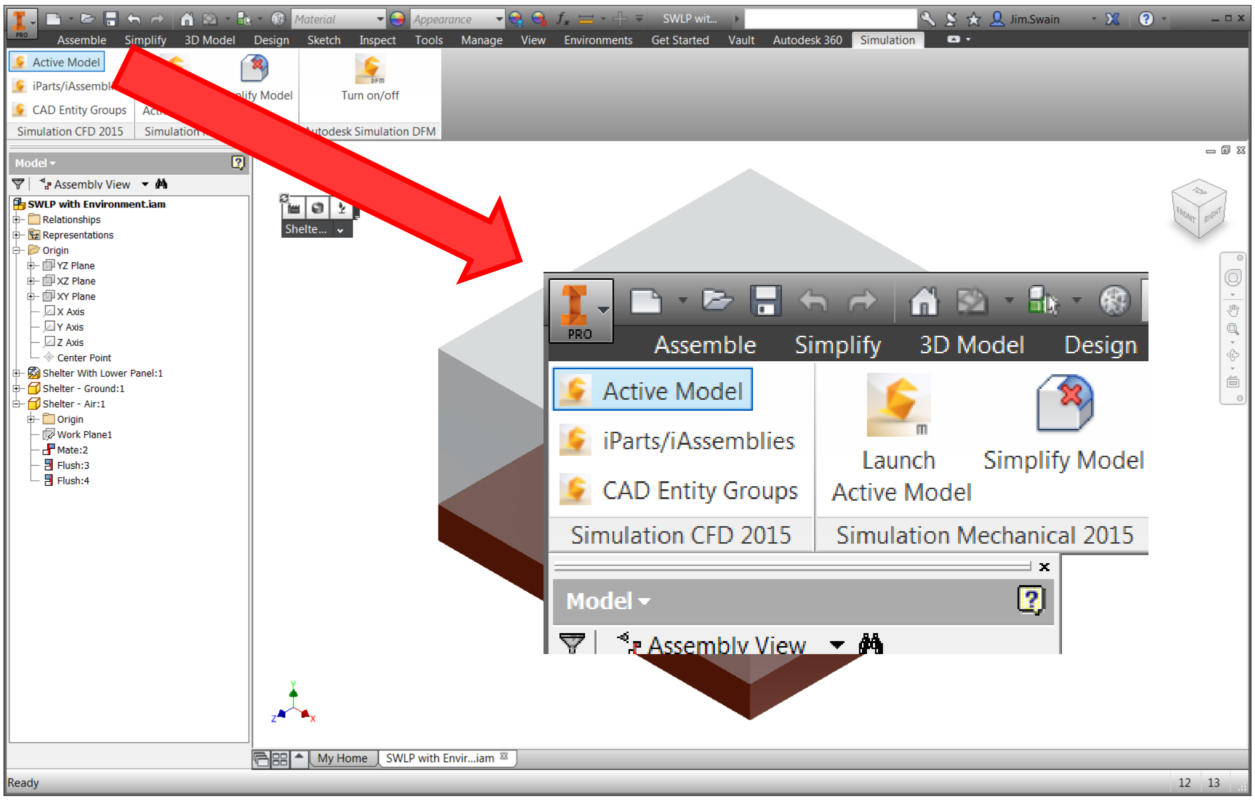
End of the Air Flow in a Room scenario.

- ## Scenario 3 – Wind Loading on a Structure

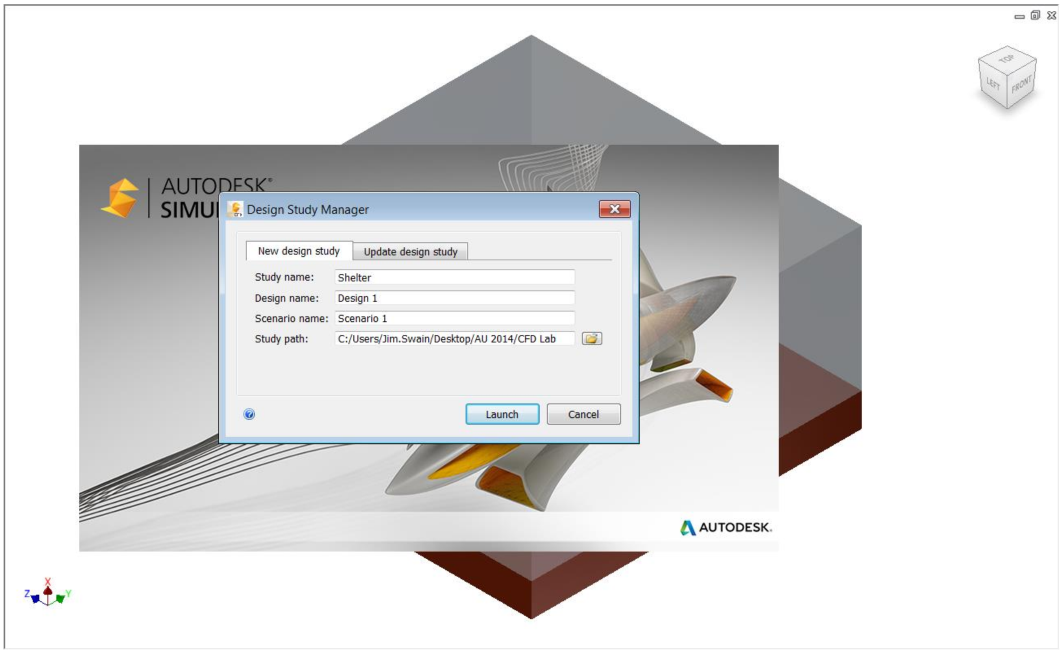
Open the Inventor File

1. Start Inventor and check that the project file is set to **SM5852-L.ipj**.
2. Open the **Shelter.iam** file.

Send to Simulation CFD



3. Choose the **Active Model** tool from the **Simulation CFD 2015** panel of the **Simulation** tab.
4. Click **OK** in the Save dialog box.



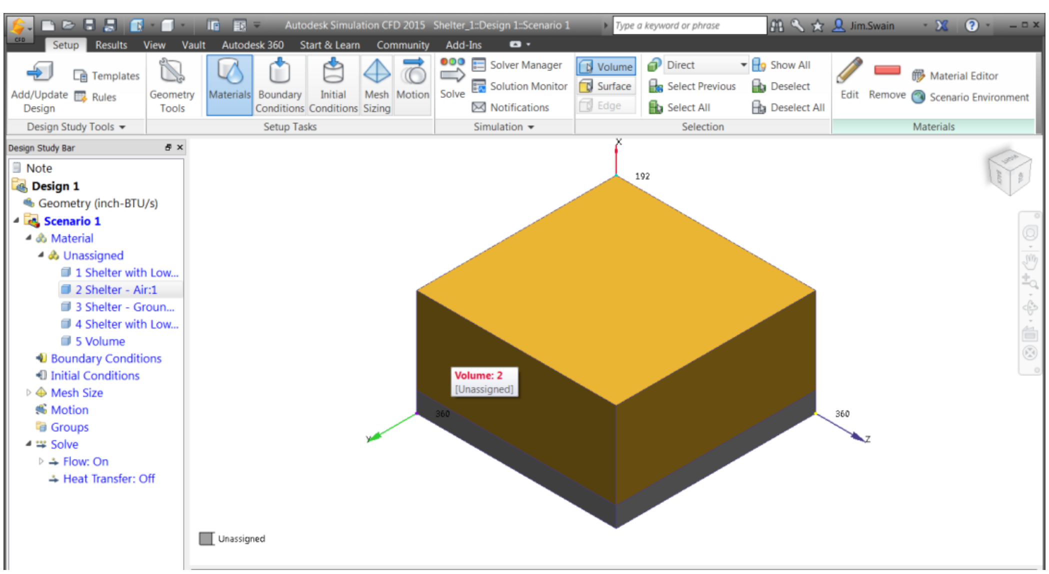
5. Click in the **Launch** button on the Design Study Manager dialog box. This will create a new design study with the default name.

Setup the Analysis

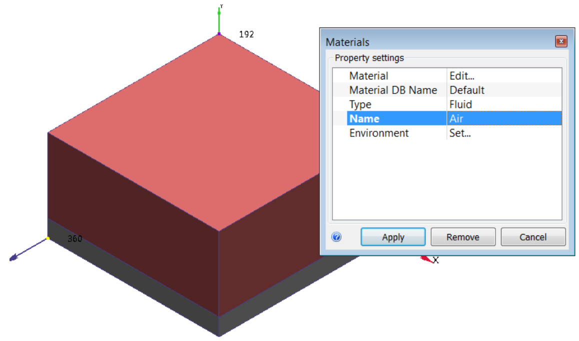
Note: it isn't necessary to apply any of the Geometry Tools to this model to get it ready for analysis.

Assign Materials

Currently the CAD parts don't have any assigned materials.



6. Click on the larger rectangle of the model. Pick the **Edit** tool from the Ribbon, the in-canvas menu, or the right click pop up menu. Change the material to **Fluid** and **Air,** and then click **Apply**.

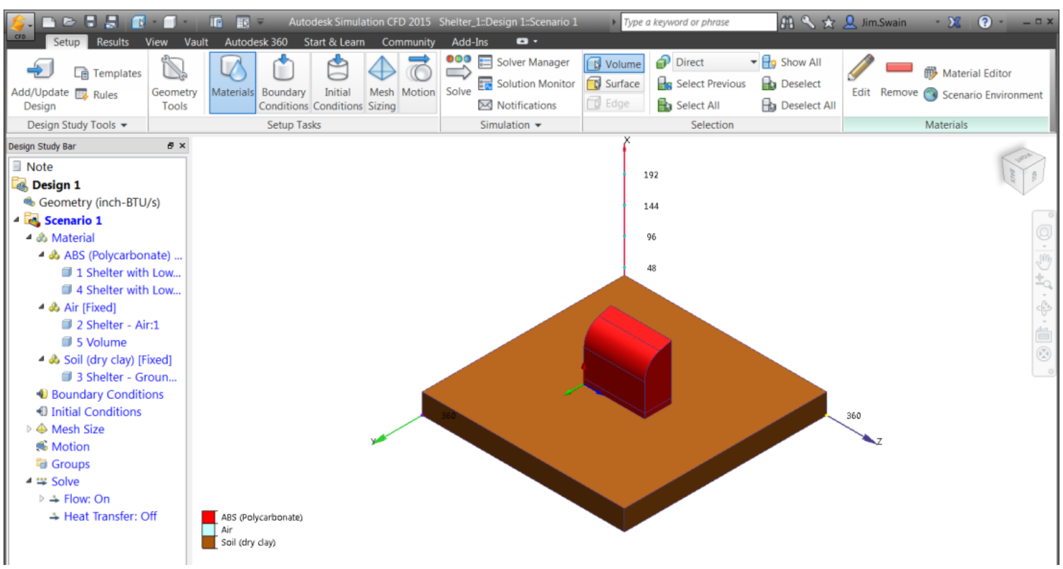


7. Right click again on the rectangle and choose **Hide**. This will hide the air volume shell.

8. Repeat steps 6 and 7 for the underlying air volume.

9. Highlight the volumes for the shelter. Edit the material to be the **Solid** named **ABS (Polycarbonate).**

10. Highlight the volume for the ground, and edit its material to be the **Solid** named **Soil (dry clay).**

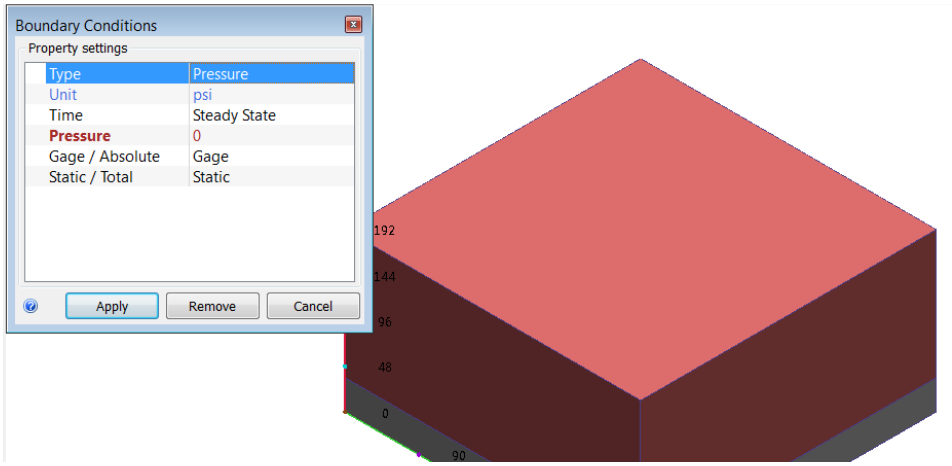


11. Right click in the graphics window and pick **Show all.**

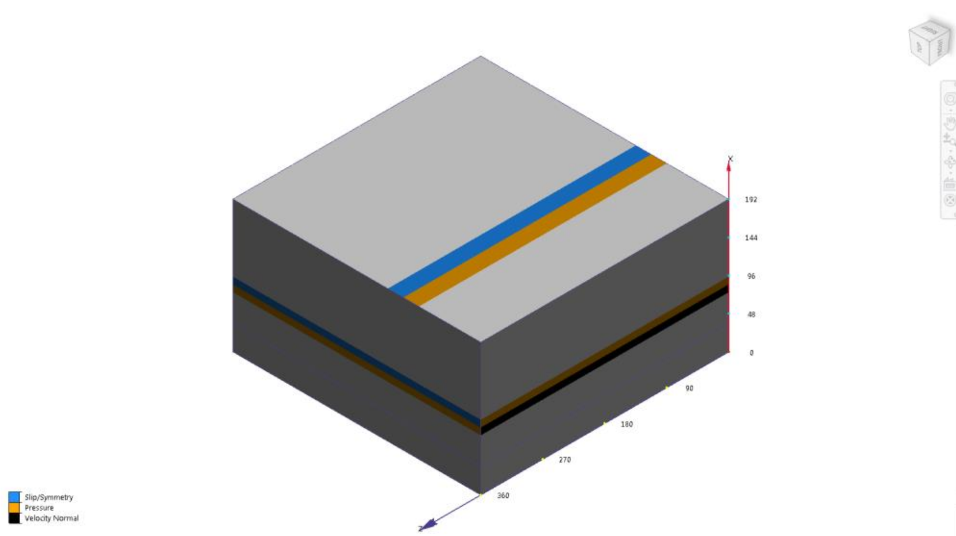
Assign Boundary Conditions

The inlet and outlet portions of the model need to have their surface boundary conditions set to a static pressure of 0 psi gauge.

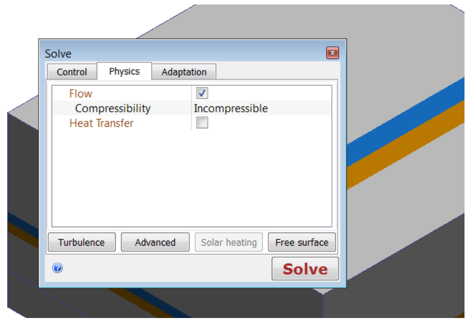
12. Start the **Boundary Conditions** tool on the Setup Tasks panel.



13. Select the outer surfaces of the outer air volume and set the **Pressure** to **0 Gage**.
14. Select the XZ face of the outer air volume and set the boundary condition to a **Velocity** of **35 mph**. Be sure to select the face so that the air blows into the open side of the shelter.
15. Select the left, right and top faces of the outer air volume and apply **Slip/Symmetry**.

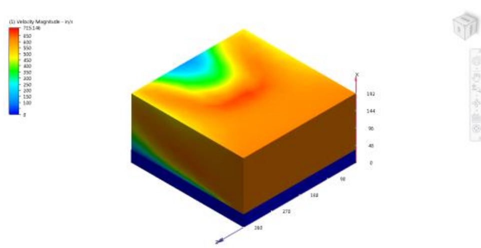


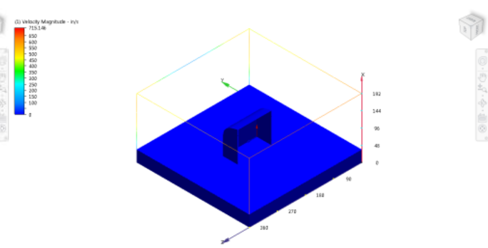
Run the Analysis



16. Pick the **Solve** tool, make sure only **Flow** is selected, and press the **Solve** button.
17. Once the analysis has started, and the Results ribbon tab is active, right click on the overall volume and **Hide** it.
18. Click again in the same area and turn the display of the other air volume to **Outline**.
19. Go back to the Solve dialog box and **Stop** the analysis.

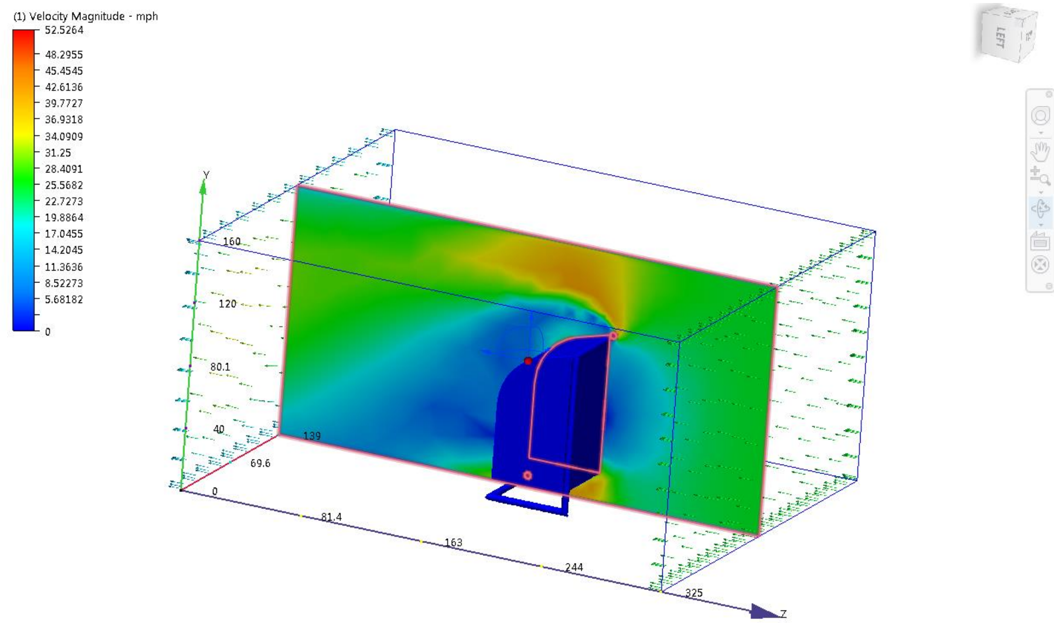
Once one iteration of the analysis has completed, you can start investigating the results. For the next section you will open a study where the analysis has successfully completed.

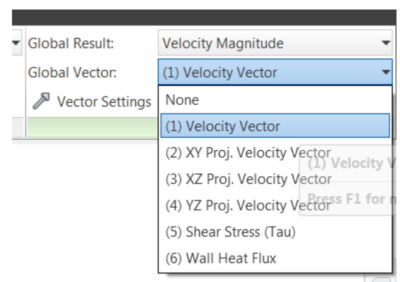




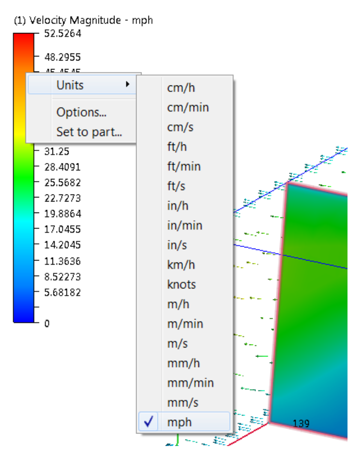
Investigate the Results

Open the Results File



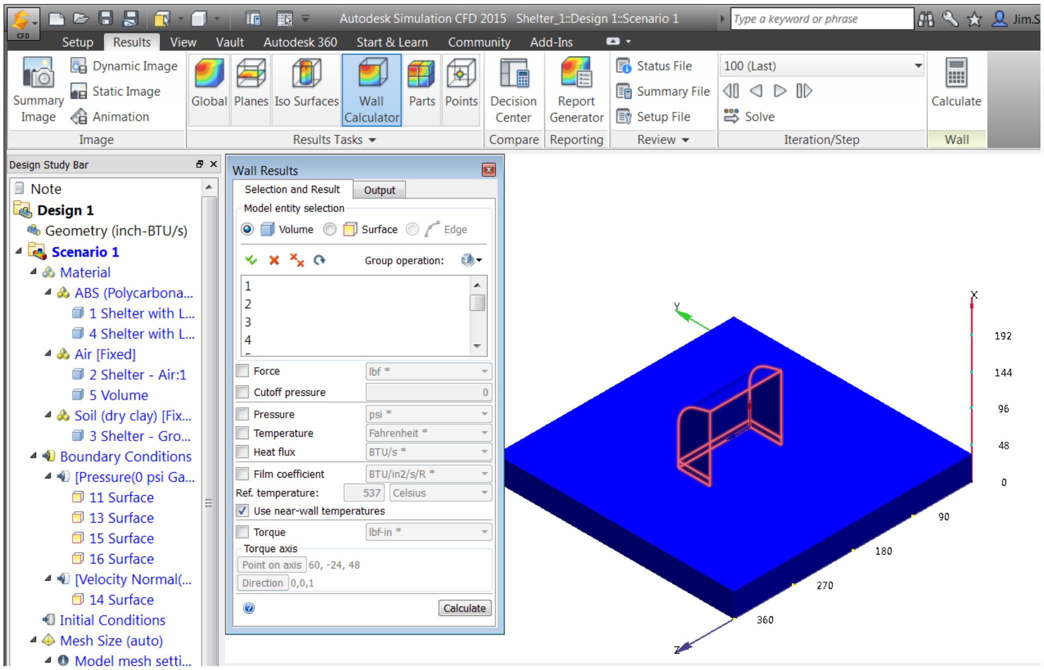


20. Use the **Open** command from the Application Menu, and open the **Full Shelter.cfz** file. This will bring in results from a completed analysis.
21. Select the Results tab, and change the **Global** results to **Velocity Magnitude**.
22. Right click on the overall volume and **Hide** it.
23. Click again in the same area and turn the display of the other air volume to **Outline**.
24. Right click on the color bar and set the **Units** to **mph**.
25. Change the Global Vector: to **Velocity Vector**.
26. Select the **Planes** tool and click the **Add** button.
27. Change the display results for the plane to **Velocity Magnitude**.
28. Turn on the display of the **Velocity Vectors**.
29. Move and rotate the plane to explore the air flow into the shelter.

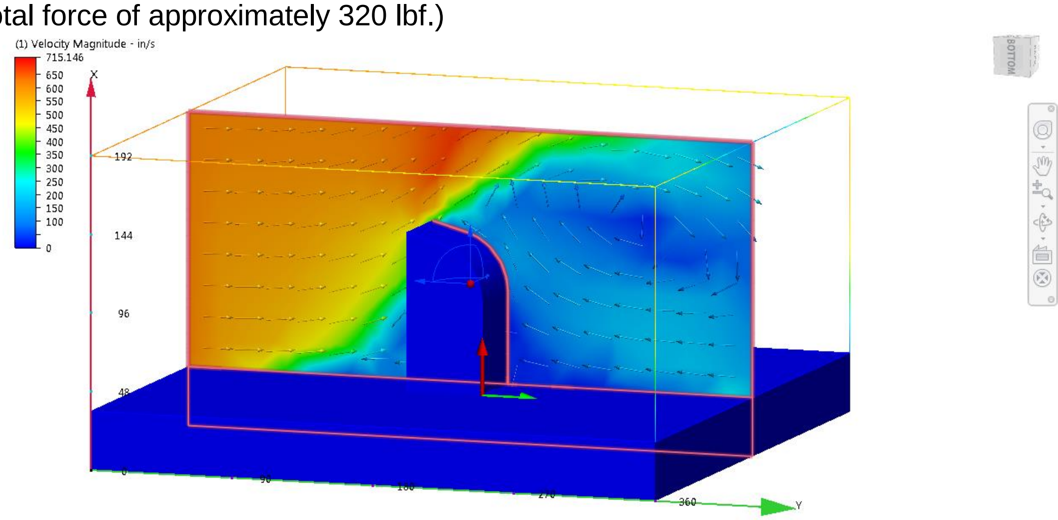


Notice how the air is recirculating around the backside of the shelter.

Forces on Walls



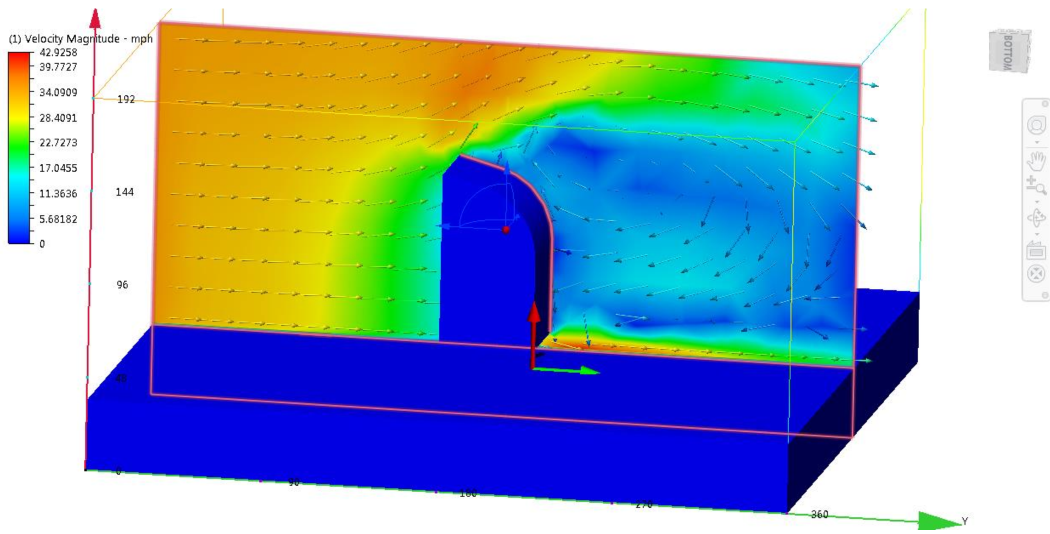
30. Start the **Wall Calculator** tool and set the Model entity selection to Volume.
31. Hide the two air volumes.
32. Select the shelter itself.
33. Click the check box to have **Force** calculated and click **Calculate**.
34. Scroll all the way to the bottom of the calculations and note the components of the forces on the shelter. (The result should be about 167 lbf in X, 273 lb in Y and -1 lbf. In Z, for a total force of approximately 320 lbf.)



Compare Results

35. Use the **Open** command from the Application Menu, and open the **Shelter – Both Cases_support.cfz** file.

36. Double click on **Scenario 1 of Shelter wo Lower Panel**.
37. Check the air flow patterns in the same manner as for Design 1. Note the high velocity area at the ground level opening.
38. Check the wall forces in the same manner as for Design 1. (The result should be about 192 lbf in X, 318 lbf in Y and -5 lbf in Z, for a total force of approximately 371 lbf.)



Notice how the loads are about 10% higher for the revised design.

<u>End of the Wind Loading scenario.</u>