AUTODESK UNIVERSITY 2014

SM6178-L – Wind Stress Study for Designers

Kevin L. Marchant – D3 Technologies, Speaker
Dave Graves – Autodesk, Co-Speaker
Jonathan den Hartog and Jim Swain - Lab Assistant's

SM6178-L This class will work through the process of using Simulation CFD and Simulation Mechanical software to generate engineering data throughout the product development process in order to optimize the design. You will discover that you don't need to be an analyst or expert in finite element analysis (FEA) or computational fluid dynamics (CFD) to use the Simulation software as many engineers still believe today. We will walk through the entire process of setting up and running simulations that examine how the effects of wind may directly affect mechanical properties, performance, and product design.

Learning Objectives

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

- Learn how to gather the relevant information to change the path of a design early in the project
- Learn how to create a usable design study of the product being engineered
- Learn how to export the results of the simulation process with a clear, understandable method
- Learn how to take your design through many stages of the simulation technology within the Autodesk offerings

About the Speaker

My engineering life in a nutshell, I've worked in most major design fields before currently working at D3 Technologies. I'm a certified Inventor and AutoCAD professional as well as leading two different Autodesk User Groups in Kansas City (I know, I'm a geek) I am One-Third of the D3 Technologies Simulation team. I am also our 3D Printer Specialist and am enjoying that growing industry and Technology. On a personal level I'm a father of two wonderful children who keep me active in life and I am a husband to a fantastic woman and my best friend. I enjoy many outdoor activities and am Adventure/Fun (OCR) runner.

Dave is a Technical Specialist with Autodesk currently working on the MFG Team. He has had positions in Design and Manufacturing and has over 10 years' experience with the Autodesk Simulation CFD products. He is very passionate about the Digital Prototyping solutions from Autodesk. In his spare time he enjoys running, golf, and spending time with his family.

Kevin.Marchant@d3tech.net Dave.Graves@Autodesk.com

Introduction to CFD Simulation

What is CFD? (Computational Fluid Dynamics) CFD is a process of using numerical methods and algorithms to solve and analyze problems that involve fluid flow with a fluid being a liquid such as water or a gas such as air or Hydrogen. For example water through a valve or air over a wing.

Why do we run CFD (*Computation Fluid Dynamics*) simulation on our designs? We run a simulation to find out what is happening in our design from thermal (Heat) and flow standpoint. Simulation allows us a peek/full view inside a world that would otherwise be invisible to us without taking it apart. It allows us to adjust the design and review what that changes made without a call to the machinist or out to the lab to rework the product.

Gathering the Relevant Information:

When we physically test a product, we need criteria to test to, simulation is no different. When we prepare to run a simulation we can only do so with the correct inputs that will create quality results of our design. There are many ways to gather the required information, some of that is easily available from the manufacturer of the item or part(s) in use. Others may require a test if your pushing a product to a new level of use. For our examples in the lab we are using fictitious numbers to get a more compelling result in our company.

Model Preparation

When you are preparing your model to be digitally tested using CFD software you do not need every minor detail in your components, in some case's you do not even need all of the components in the design. Even with today's technology in computing power complex model detail may generally have little effect on the heat or air flow of your design, yet may have a major effect to the simulation time of the model. By updating geometry to an appropriate level of simulation detail, your model will get great results in less time.

Bring the Geometry into CFD

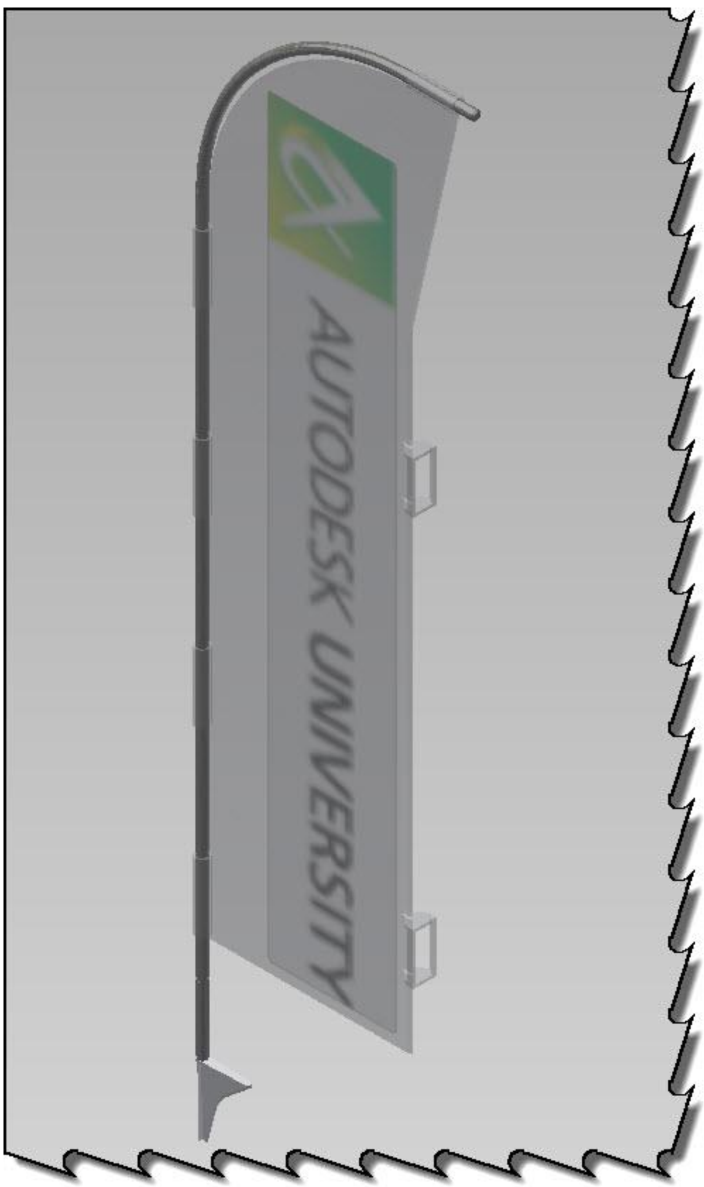
There are some Autodesk Lab tools (Sim Studio) that have a direct link as well as tools to help with the simplification and Fluid Volume creation; it's an option especially when you are dealing with a design that you need to tweak to get a smooth and efficient simulation. However the more popular tool and the one we are using today is Autodesk Inventor Professional 2015.

Inventor 2015 and Autodesk Simulation 2015 work seamlessly together, one click and you are on your way. With the selection of *Launch Active Model* Autodesk Simulation CFD is booted and we begin our journey to a look at natural powers of the world, and the dangers of unknowns in our designed products.

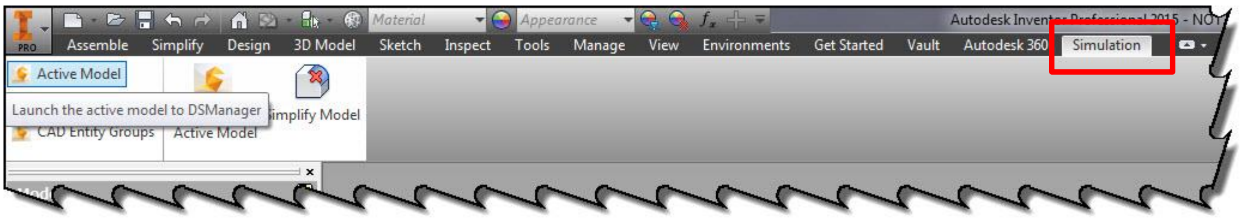
Lab Exercise: Advertisement Banner

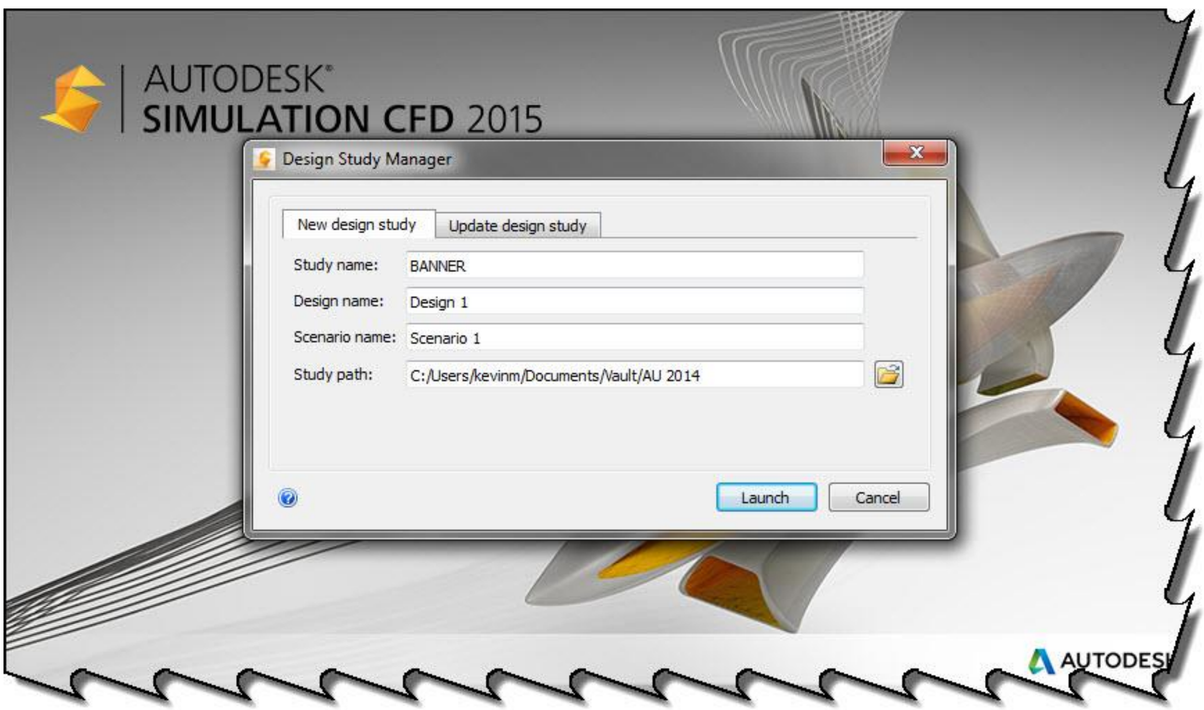
What kind of wind can our Banner withstand before it becomes a danger to passerby: Let us find out!

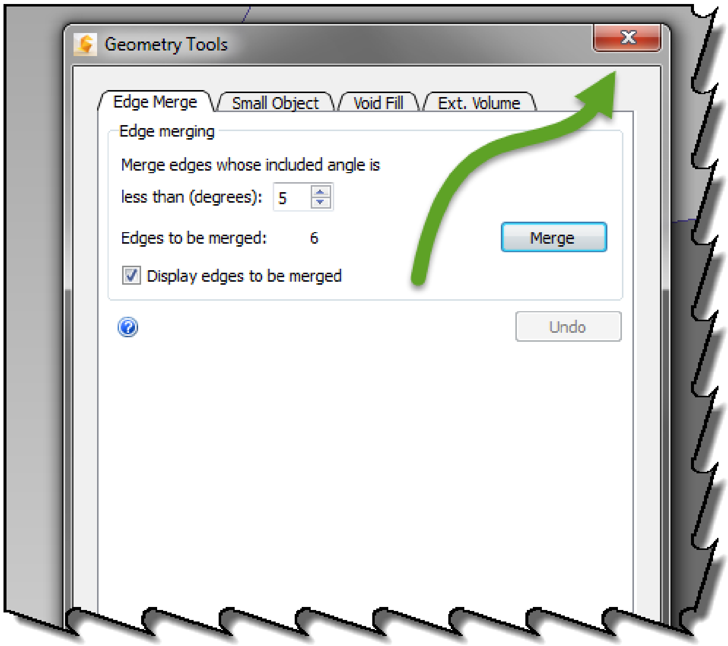
Open the BANNER.iam out of the workspace folder:



We need to create an environment for our wind to exist, or in our case turn that environment on; the visibility is turned off in our Inventor assembly file. Launch the model to Autodesk Simulation CFD



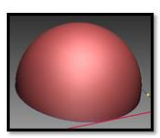


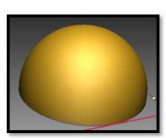


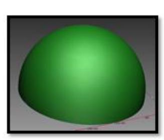
Close the *Geometry Tools* dialog box and the *Output Bar (Instructor Demo)*

CFD Navigation:

View controls in Autodesk Simulation CFD
Shift +MMB - rotation (orbit)
Middle Mouse Button *scroll* - zoom
Middle Mouse Button *hold* – pan
Ctrl + MMB – Blank/Hide
Ctrl + Wheel – Blank “Undo/Redo”

Selected 

About to Deselect 

Ready to Select 

Selection Colors

Assign Materials

Assigning material is no different than specifying physical part properties in Inventor. Typically for Simulation CFD the two main material types are solids and fluids.

Material Types:

*Solid – Solid Materials such as Steel, Aluminum, ABS (Polycarbonate)

*Fluid – Liquids or Gasses (Air, Water, Blood, Nitrogen, etc.)

Additional material types consist of:

Resistance (Used to represent things like Perforated Plates, Filters, etc.)

Pump/Fan/Blower (Allows you to input Blower/Fan Curves on simplified model)

Compact Thermal Model (CTM) (Two Resister Thermal Model primarily used in Electronics)

Printed Circuit Board (Allows users to specific layer thickness and material to generate lump sum properties)

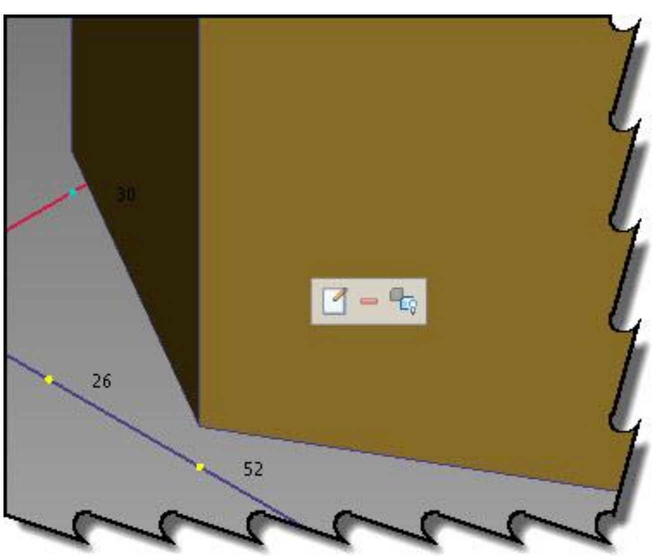
LED (Simplified model for LED application)

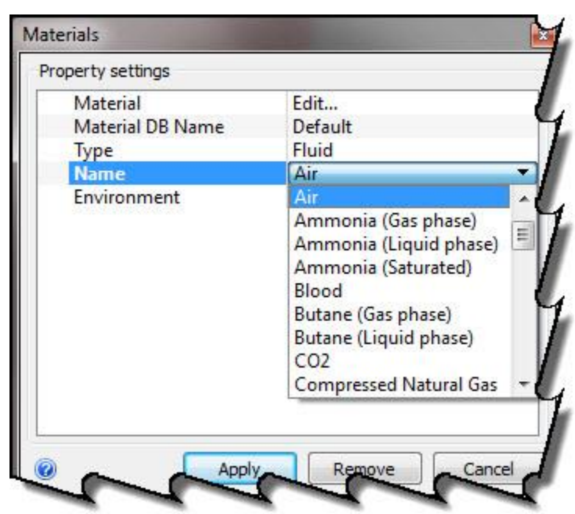
Thermoelectric Cooler

Heat Exchanger

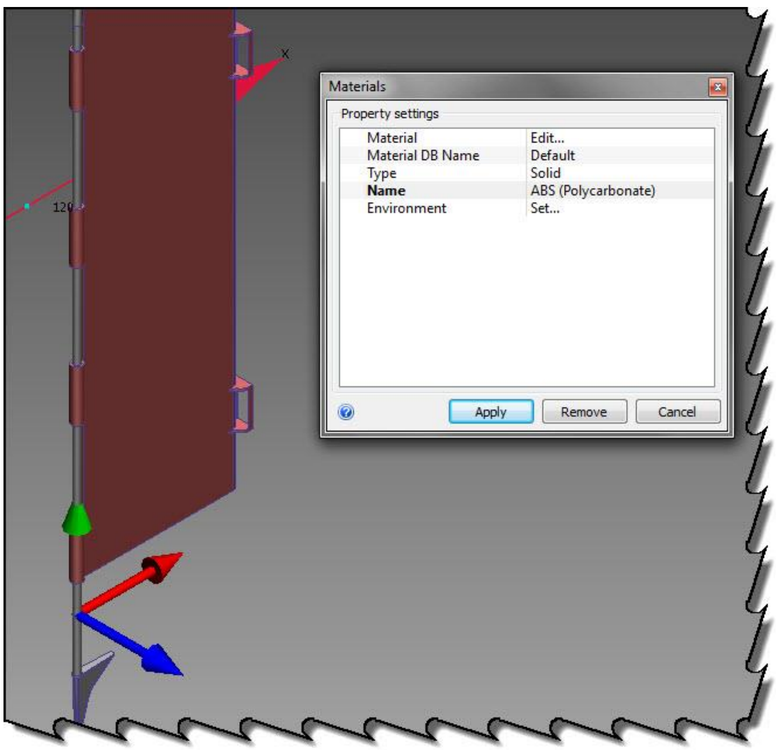
**Materials we are going to use in today's exercise.*

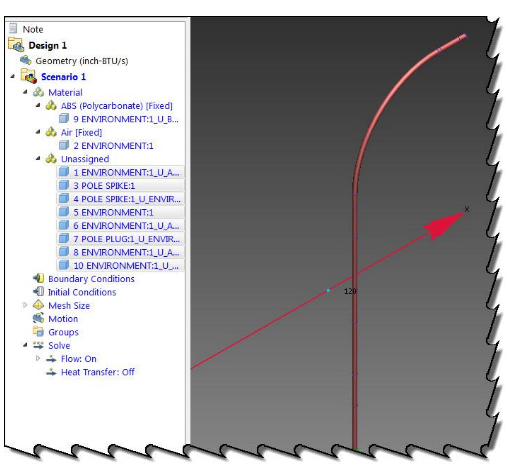
Select the large Hexagon and Apply a Fluid Type of *Liquid*, Name of *Air*:

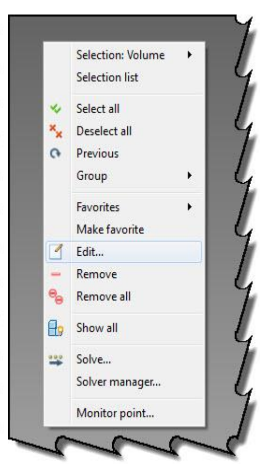


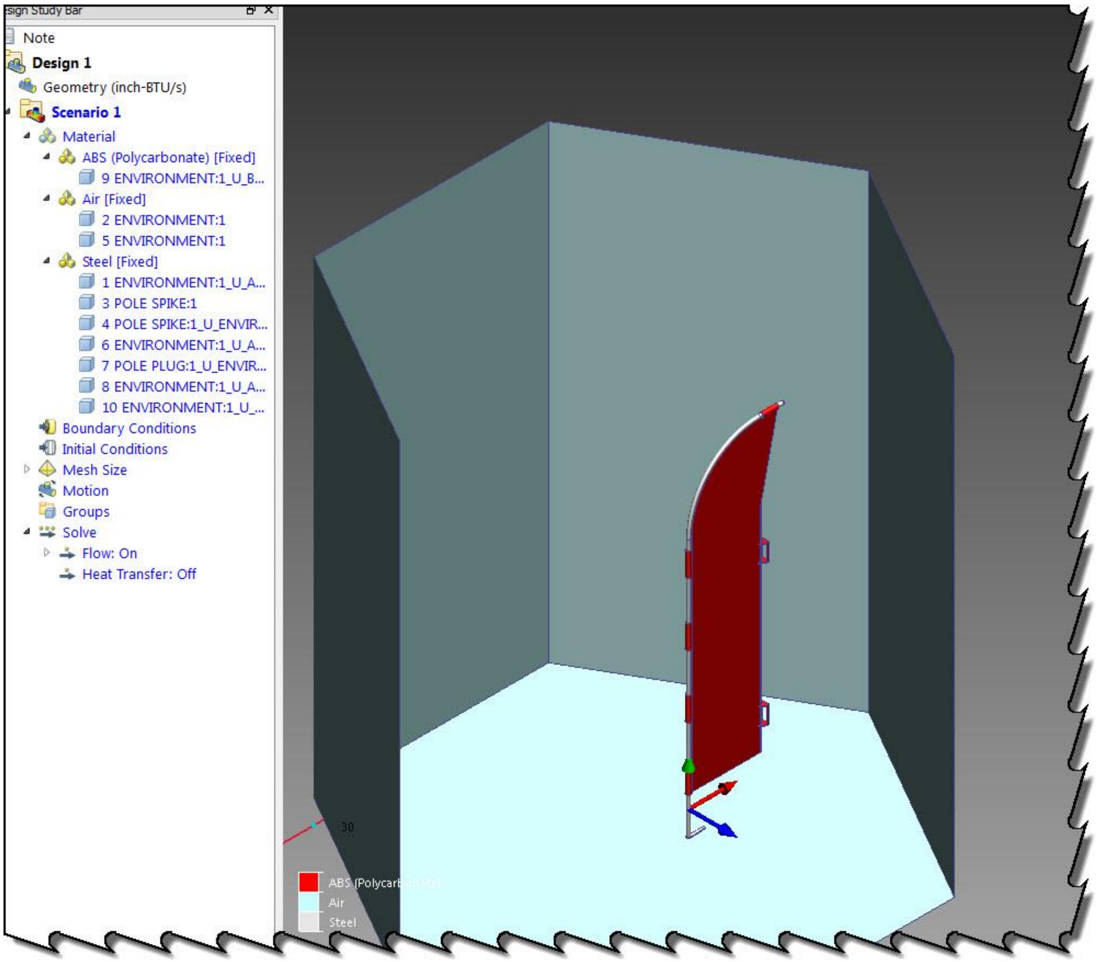


Let's work our way through our Assembly Model adding the Materials as shown. As we apply materials to our model we can see our list of Materials grow. (Instructor Demo)









Boundary Conditions

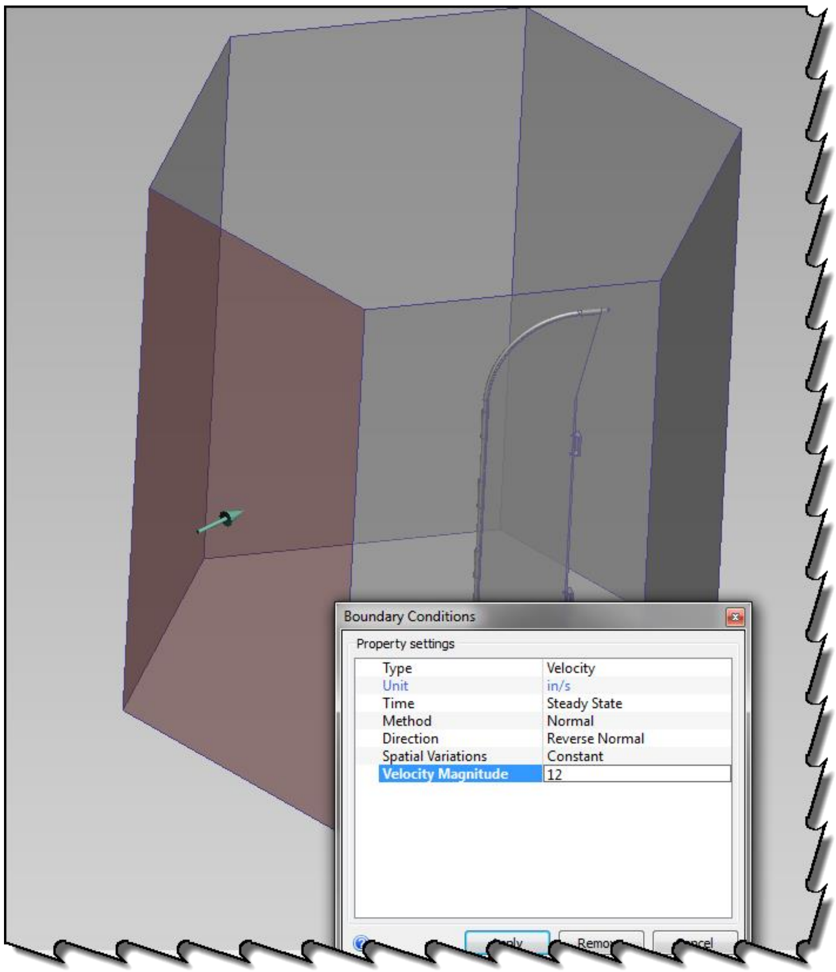
Boundary Conditions are used to specify operating conditions or constraints of a model. Some examples include things like known pressure drop, ambient air temperature, heat dissipation and flow rate. If you are setting up a physical test, this would be the conditions you are using in that test.

We will be dealing with air movement and force for our simulation this afternoon. On some (most) of our surfaces we will be applying a prescribed velocity.

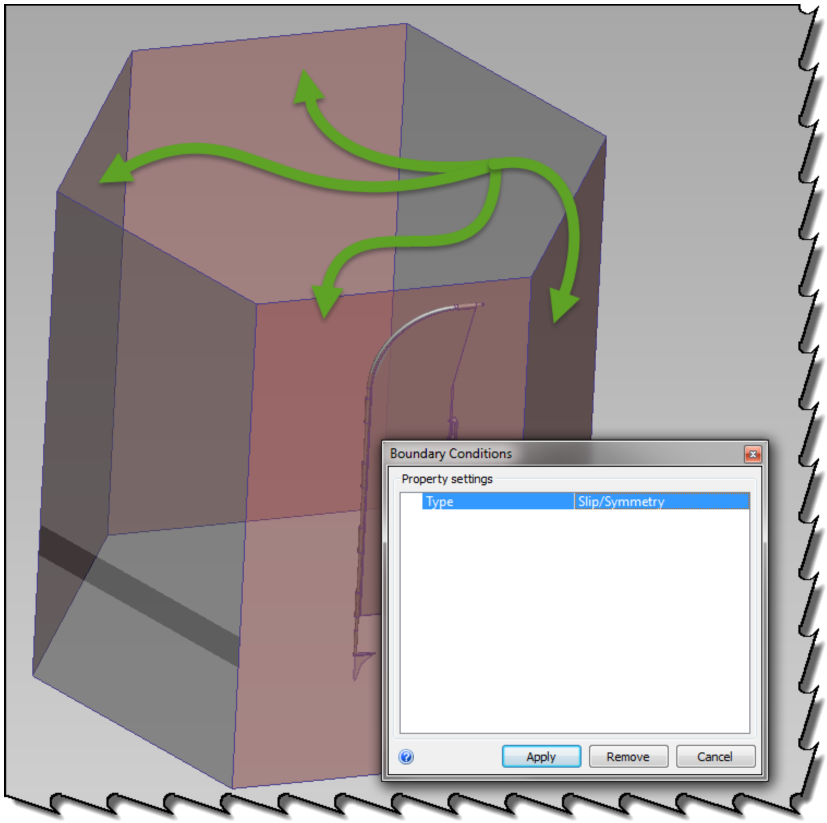
Assign the boundary conditions to the various surfaces (Velocity, Slip/Symmetry, and Pressure)

Applying a 0 pressure just lets the software know that air flow can enter or exit through that opening depending upon the calculated conditions. For this problem, we are assuming the Banner is an open, unobstructed area and open to atmosphere. For other type simulations users can specify a pressure drop or other boundary conditions.

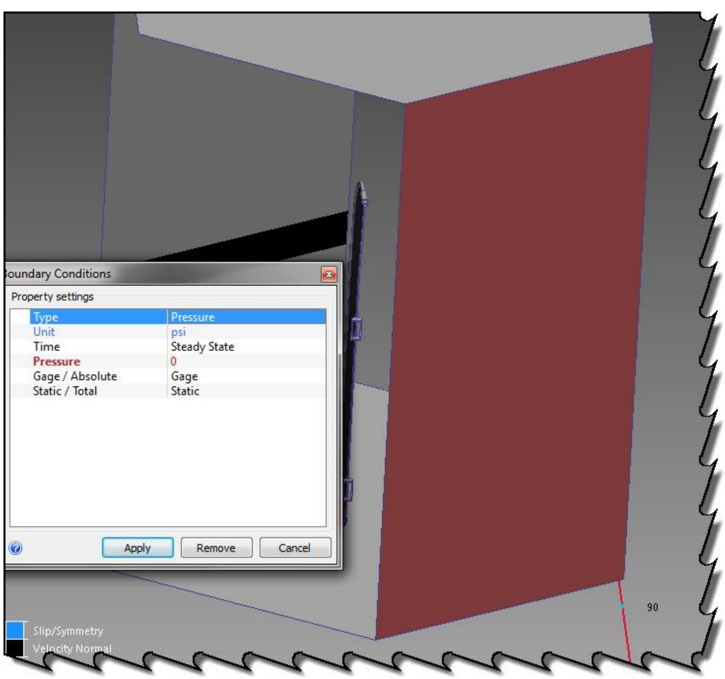
Assign Velocity to the front panel of the Hexagon with a speed of 12 mph.

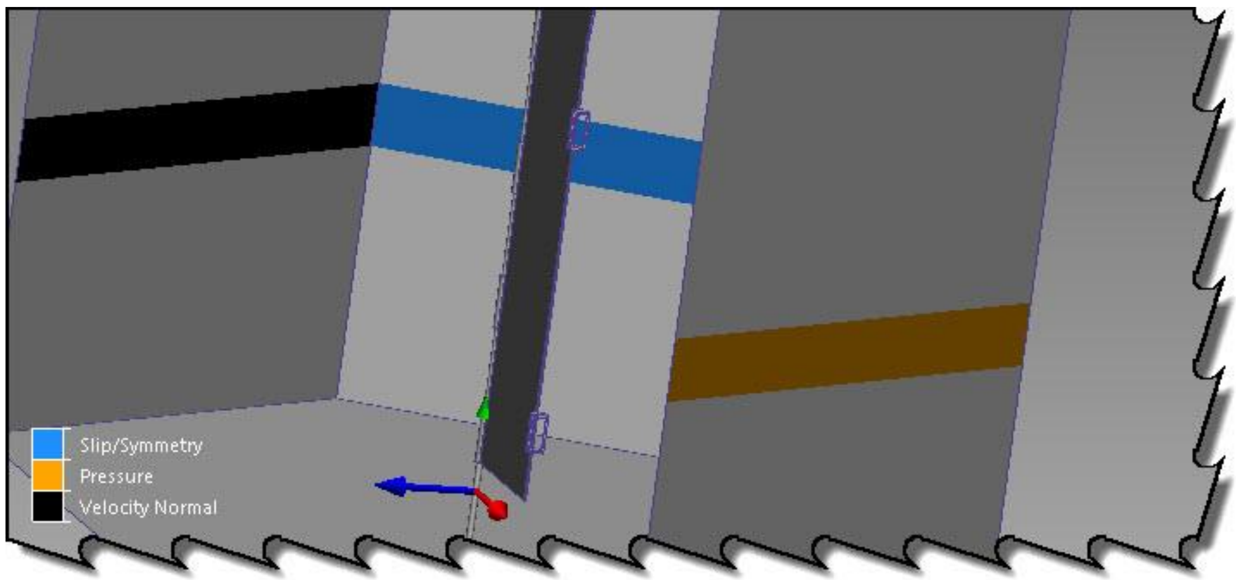


Apply the Slip/Symmetry to 4 of our 6 walls, this will allow the air to slide along the walls without changing any properties, speed and or force.



Lastly we will provide the Air a location to exit, Apply a 0 (Zero) Pressure to the back side of the Hexagon, opposite of the Velocity Boundary Condition.



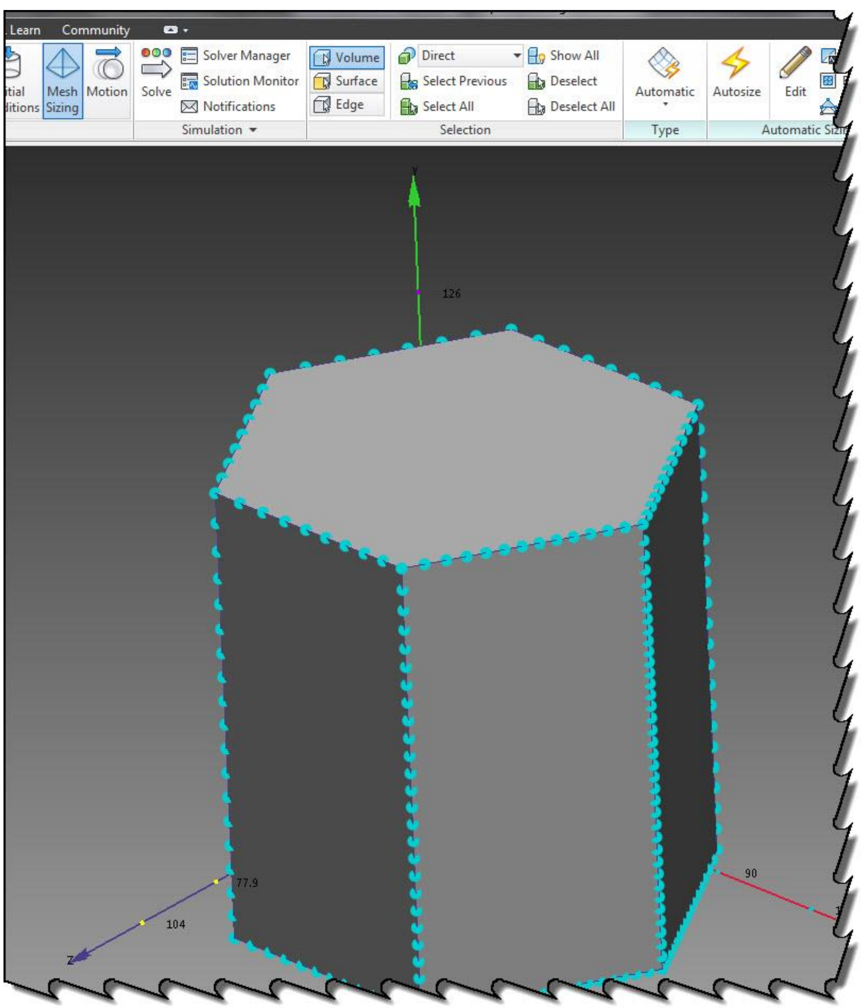


Model shown with Boundary Conditions applied as seen with the colored stripes on the surfaces.

Mesh

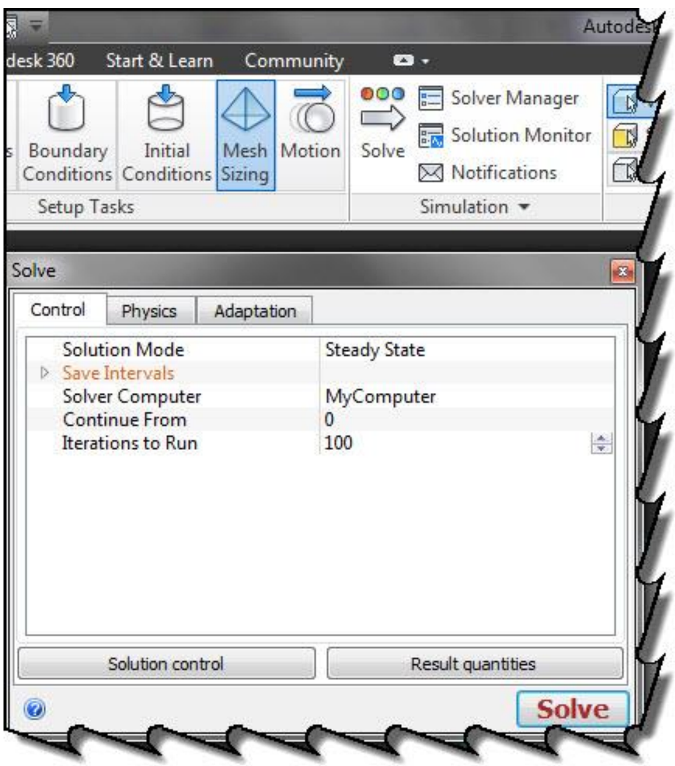
Meshing has traditionally been a challenging process. Many tools require users to choose element types and provide manual input to mesh size. While Simulation CFD does have a manual method, most users take advantage of the Automatic Mesh capabilities. The mesh is basically the mathematical version of the geometry. The Automatic Mesher looks at several things such as edge length, curvature, surfaces, to generally determine an appropriate mesh. The more mesh elements, the more closely it can be related to the actual model, however the more mesh, the longer the simulation takes to solve. Additionally, there is a point where adding more mesh does not change the results significantly so you will be running a model longer for no change in results. There are tools we have to analyze this but for this exercise we will use the default mesh settings.

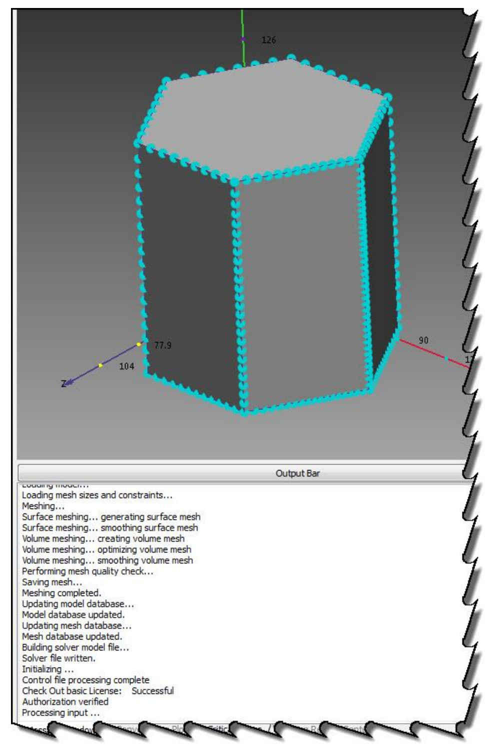
Let CFD do to the work (Autosize) however, when you have components that are not necessary to the simulation, you can control the detail of the mesh that is applied, or exclude whole components to avoid unnecessary computational time.

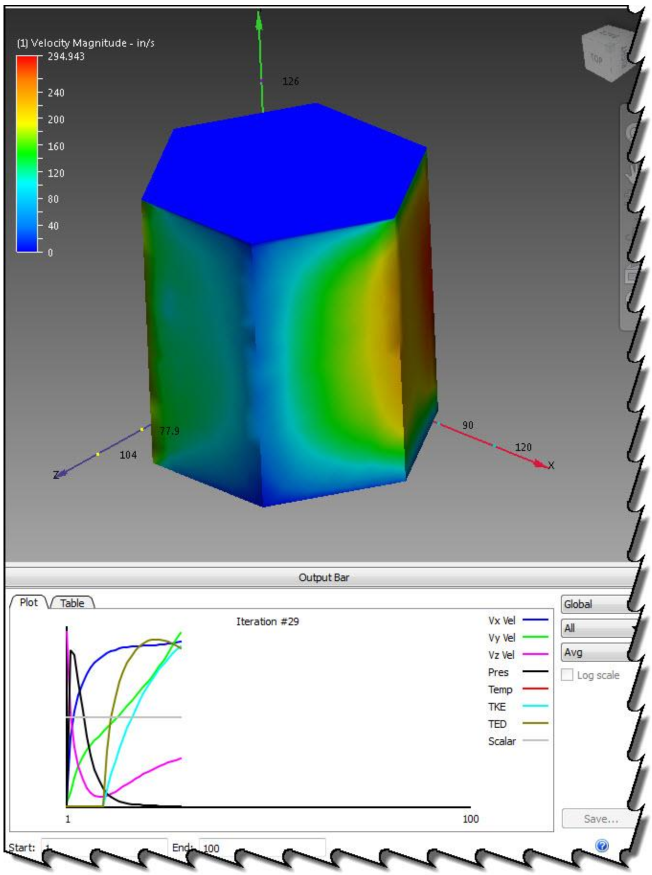


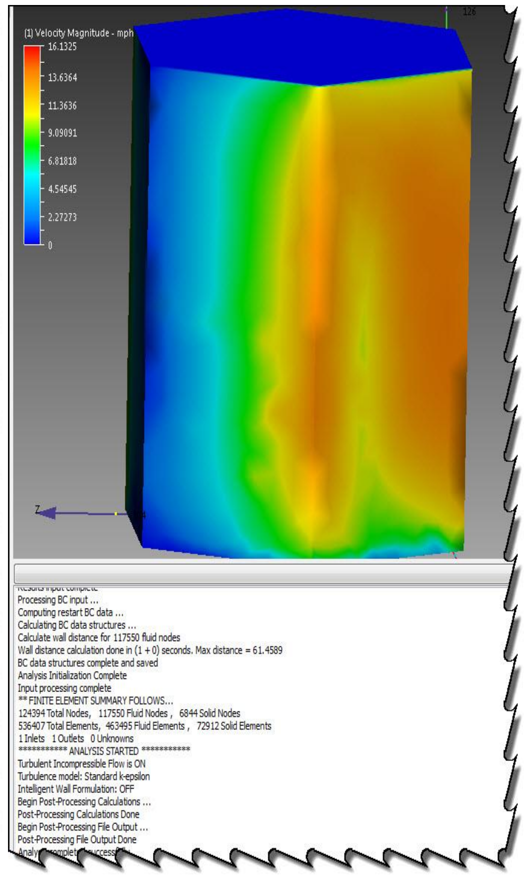
Solve:

The solver is where you tell the simulation what it is that you are trying to determine. Are you solving flow only for a valve simulation, or do you want to include heat transfer? Is heat radiation important? Are you looking to solve something with respect to time (Transient) or is a steady state simulation appropriate? For this exercise, we will calculate with the default 100 iterations.



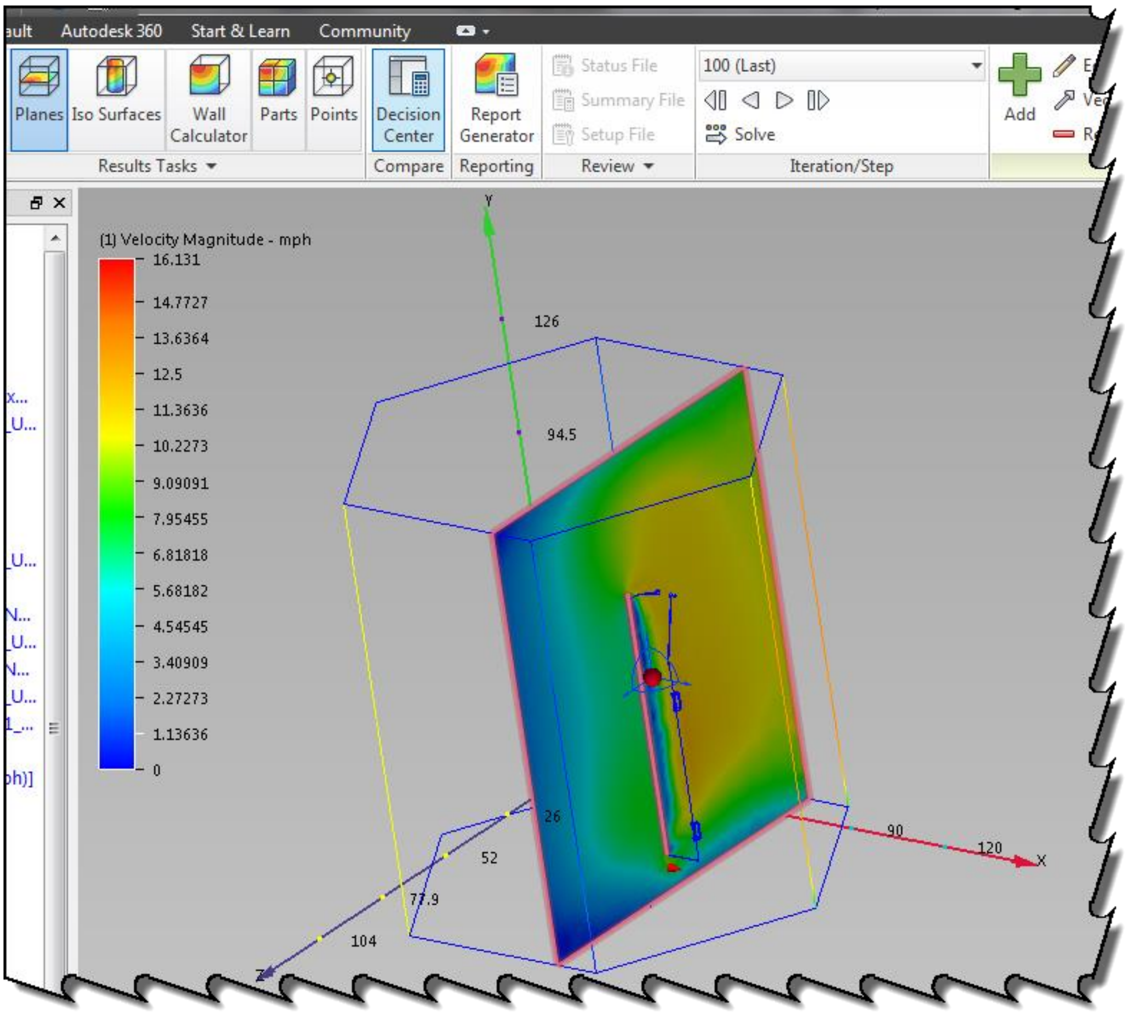



(1) Velocity Magnitude - in/s
294.943
240
200
160
120
80
40
0
126
77.9
104
90
120
Z
X
Output Bar
Plot
Table
Iteration #29
Vx Vel
Vy Vel
Vz Vel
Pres
Temp
TKE
TED
Scalar
1
100
Global
All
Avg
Log scale
Save...
Start:
End:
100


(1) Velocity Magnitude - mph
16.1325
13.6364
11.3636
9.09091
6.81818
4.54545
2.27273
0
126
Z
Processing BC input ...
Computing restart BC data ...
Calculating BC data structures ...
Calculate wall distance for 117550 fluid nodes
Wall distance calculation done in (1 + 0) seconds. Max distance = 61.4589
BC data structures complete and saved
Analysis Initialization Complete
Input processing complete
** FINITE ELEMENT SUMMARY FOLLOWS...
124394 Total Nodes, 117550 Fluid Nodes , 6844 Solid Nodes
536407 Total Elements, 463495 Fluid Elements , 72912 Solid Elements
1 Inlets 1 Outlets 0 Unknowns
************ ANALYSIS STARTED ************
Turbulent Incompressible Flow is ON
Turbulence model: Standard k-epsilon
Intelligent Wall Formulation: OFF
Begin Post-Processing Calculations ...
Post-Processing Calculations Done
Begin Post-Processing File Output ...
Post-Processing File Output Done

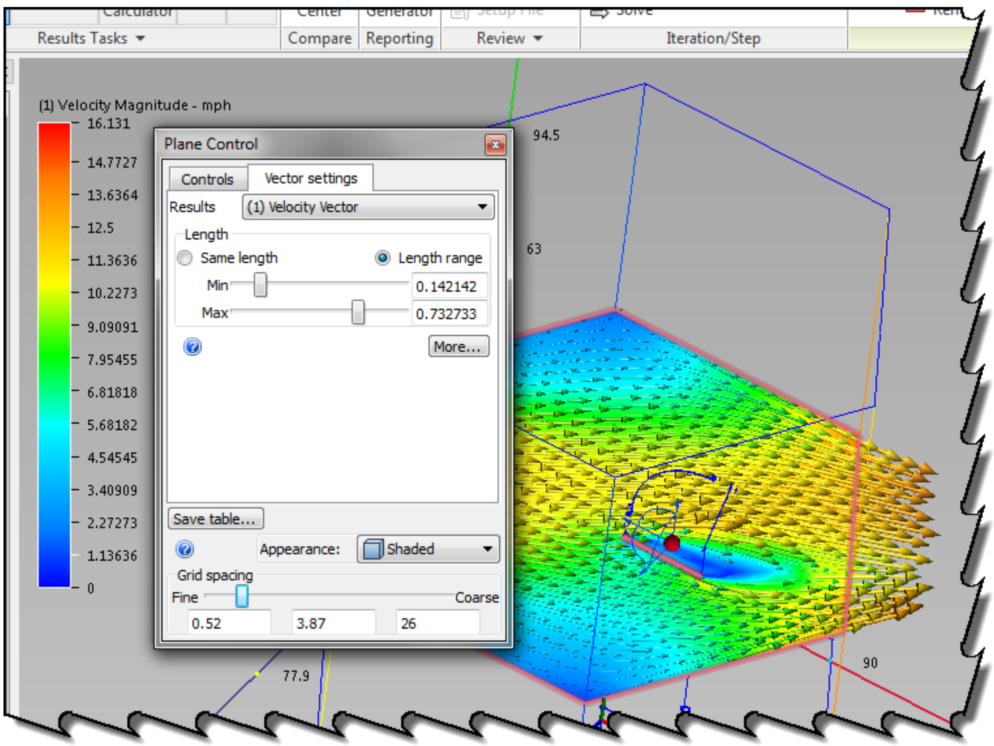
Results:

The advantage of using a Simulation CFD tools is the ability to gain insight into how and why your design is operating a certain way. This part of the exercise will demonstrate how you can interrogate the results both qualitatively and quantitatively.

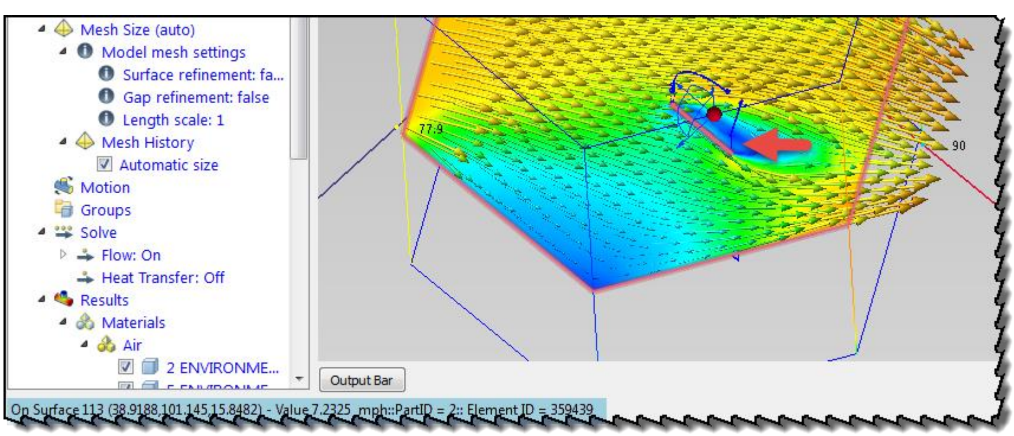


Adding Planes from the Results tab using the “Add” button in the ribbon will generate a default plane in the model witch provide insight. Left Mouse click on the plane to adjust orientation, using the arrows to slide its location.

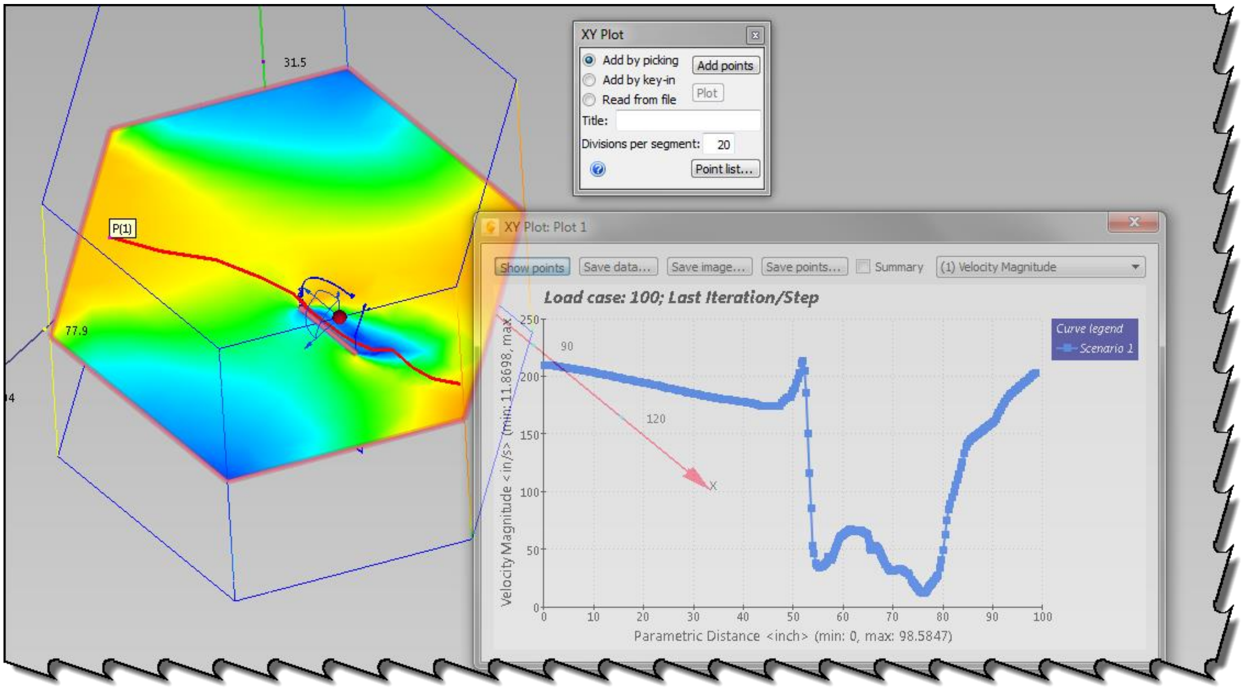
Velocity values can be shown on the model. Planes can allow you to investigate and explore specific location on the model. You can visualize the movement with the Vector Arrows to help show the direction of the fluid moving.



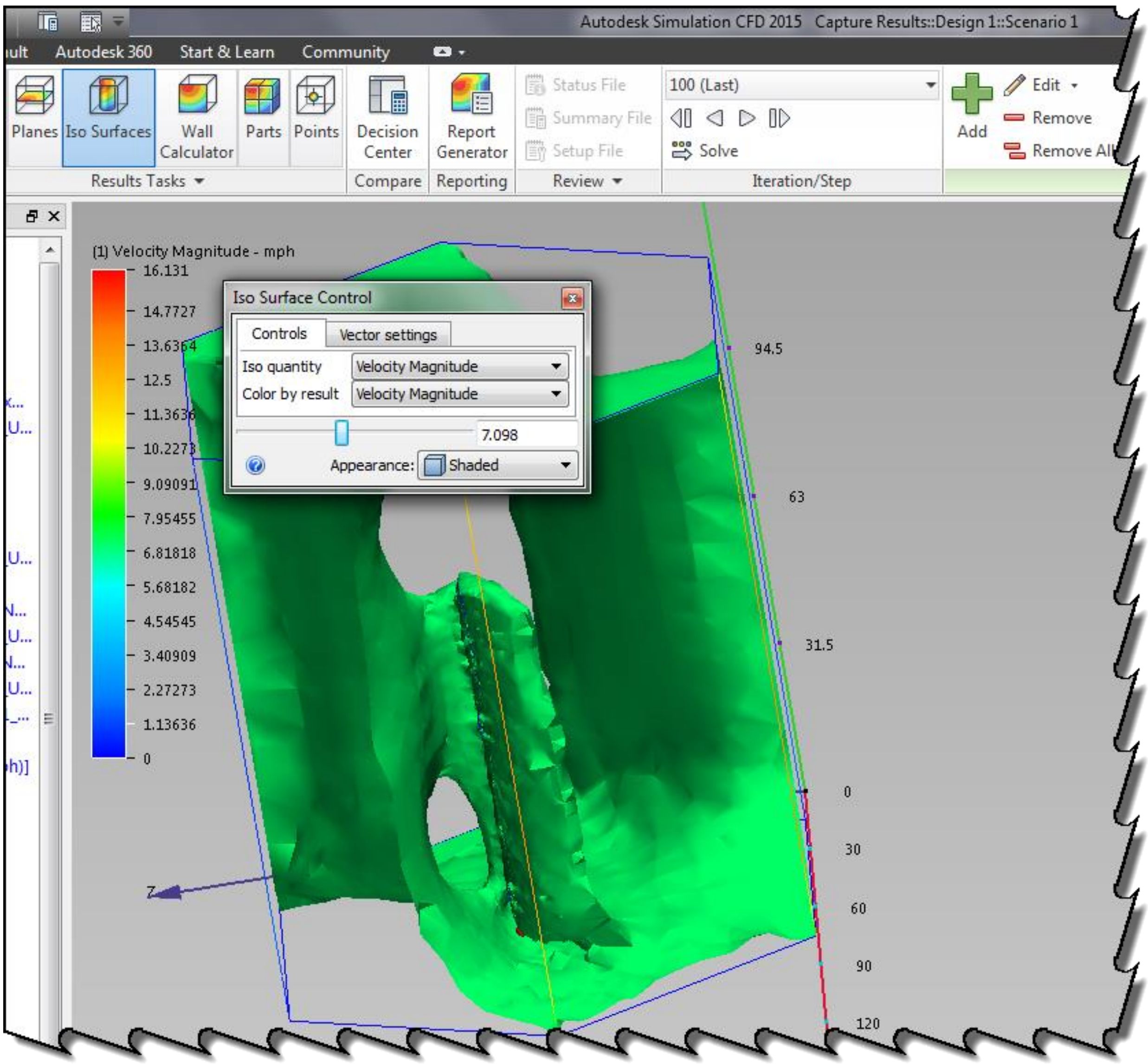
When we want specific locations of quantitative results we hold down CTRL + SHIFT and mouse where we desire.



When we need quantitative results across a profile we can plot them on the XY Plot from a plane. When you start the "XY Plot" command on the ribbon, you will "Add by picking" and choose the points you want to track. When you have your points selected the "Plot" Button will be available and when selected you will see your XY Plot appear as shown.



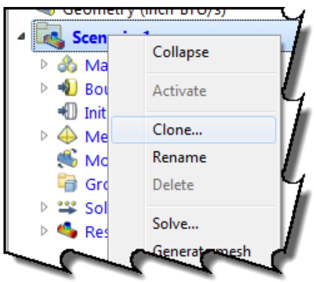
An **isosurface** is a three-dimensional analog view of an isoline. It's a surface that represents points of a constant value (e.g. pressure, temperature, velocity, density) within a specific volume of space. When you want to see the total area that consists of one sample of result, you can easily view that with an isosurface within Autodesk CFD 2015 by selecting the Iso Surfaces icon and then hitting "Add" button. You can then choose the specific parameter that you want displayed.



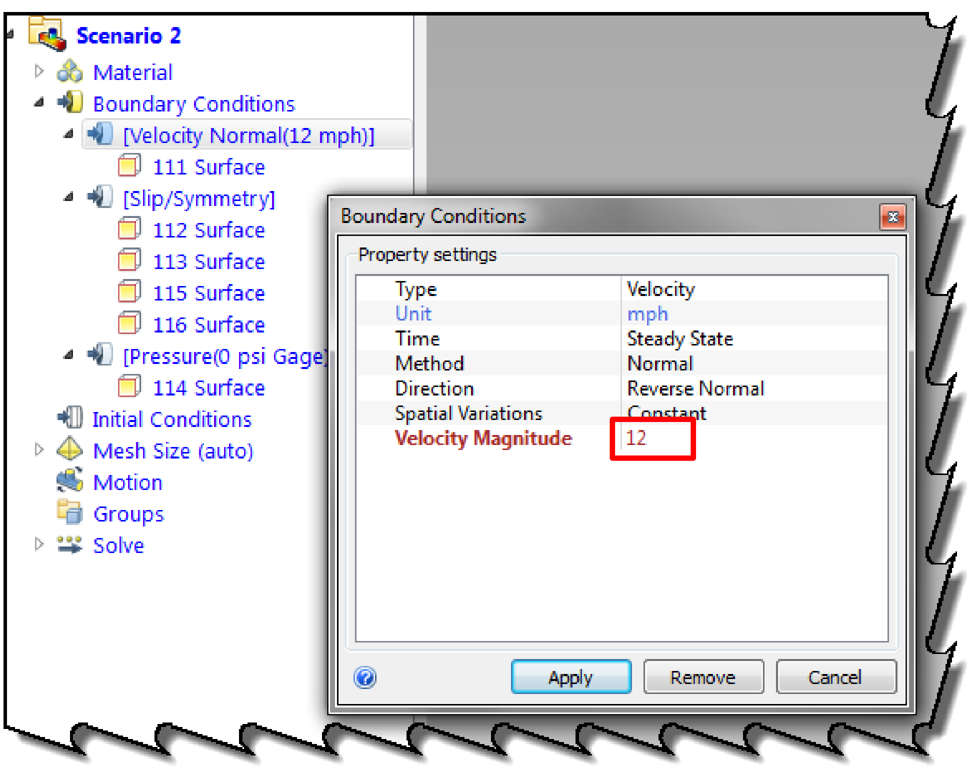
Design alternative

We have just learned how our initial design is going to perform. By leveraging the decision center in the Simulation CFD application, we are going to determine if we can make the design better by adding additional vents. This process is automated and leverages all of the previous work you have done earlier simulations.

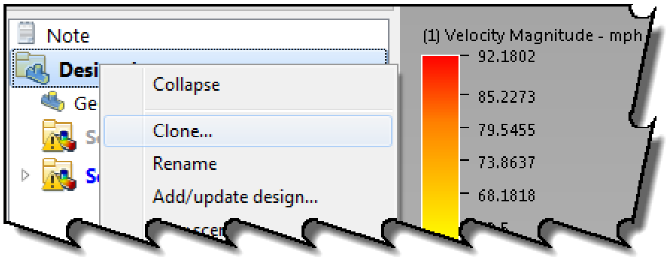
What's next? Let's change some things around and optimize the design without ever building a single physical proto-type.



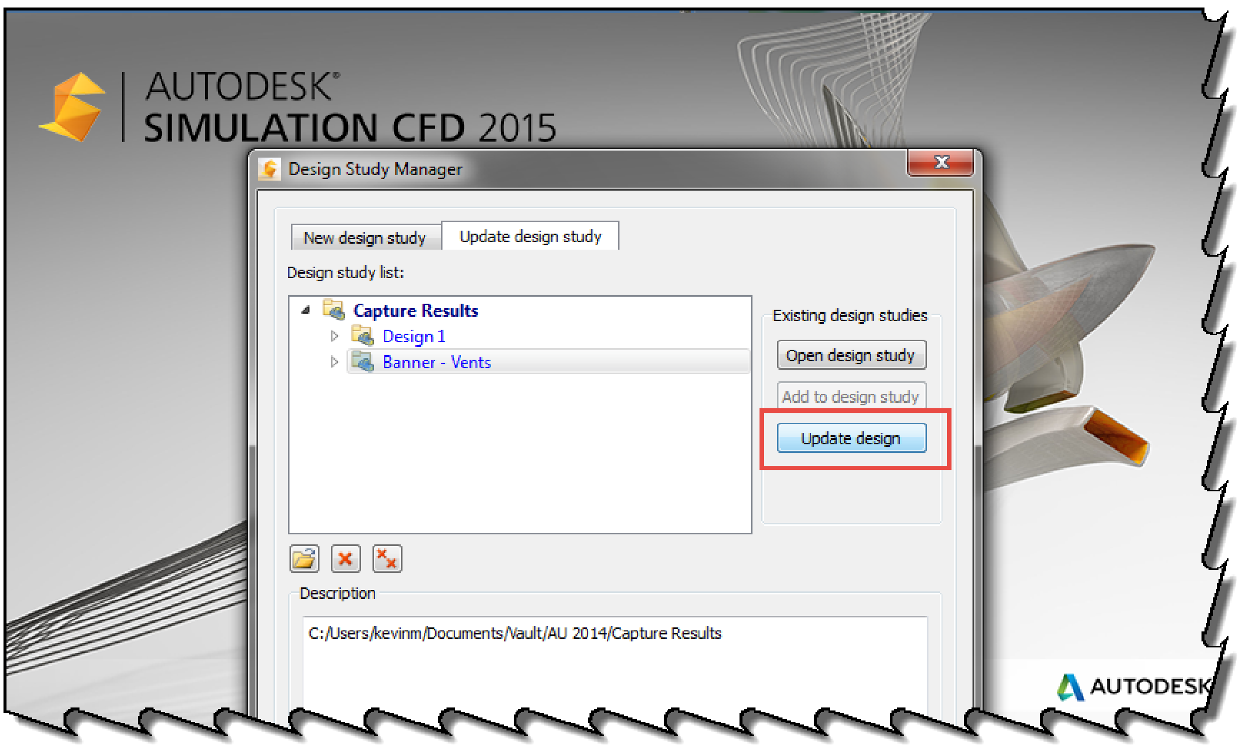
Clone Scenario 1
Change your Boundary Condition on Velocity from 12 to 60 mph
Solve again from 0 to 100



Now we will clone the design. As shown below, create a design with the name of "Banner – Vents" hit OK.

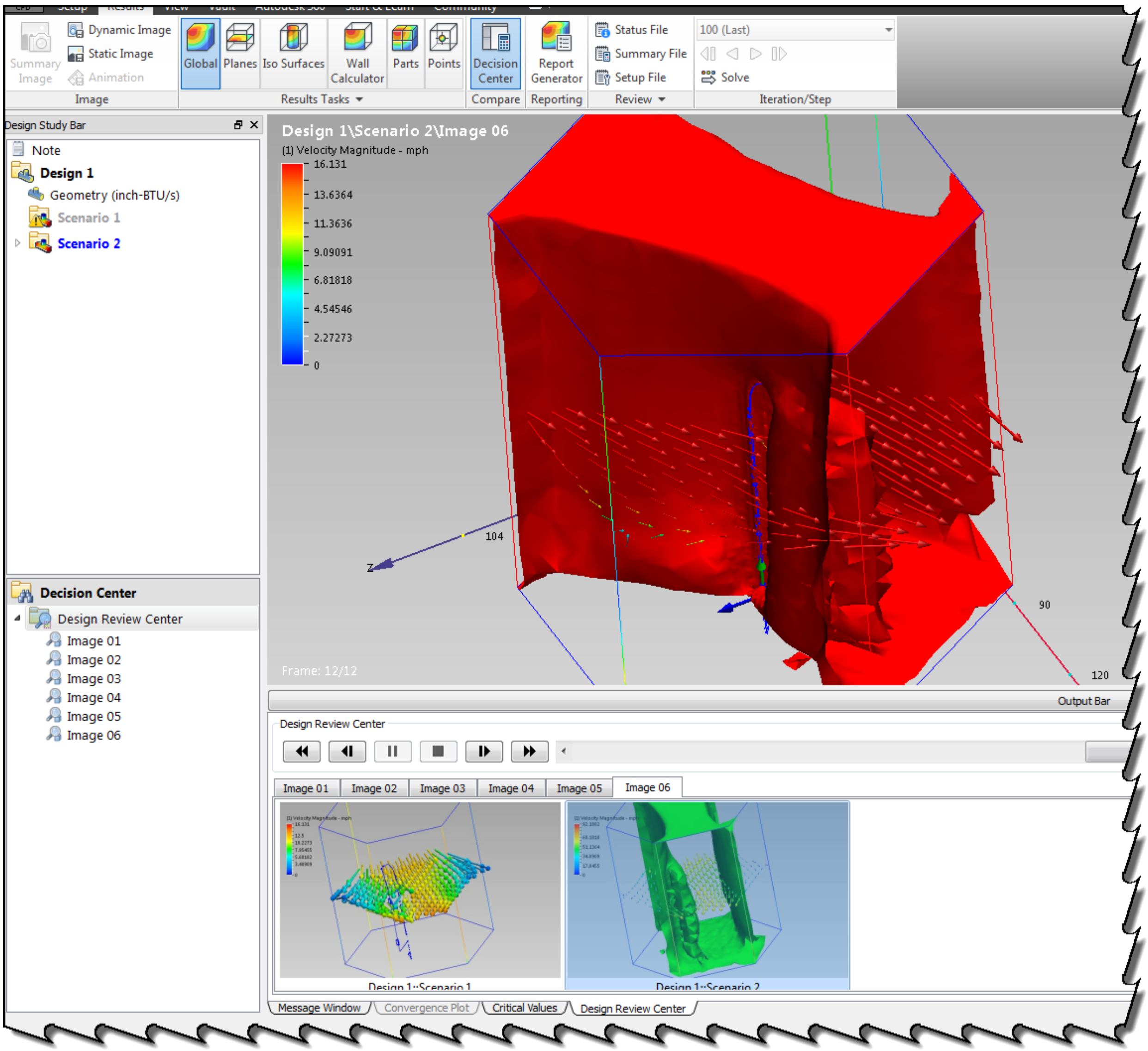


Now we need to bring the modified geometry into Simulation CFD. In Inventor 2015 activate the BANNER.ipt in the BANNER.iam file and move the End of Part down the bottom of the model tree. You will see 2 vents appear in the assembly. Follow the same steps as before only "Update design study" instead of the "New design study" as shown below.



With the updated design in Simulation CFD you can perform another digital analysis with the new geometry in the model. Same steps as before using the Solve button be sure to continue from iteration 0 (zero).

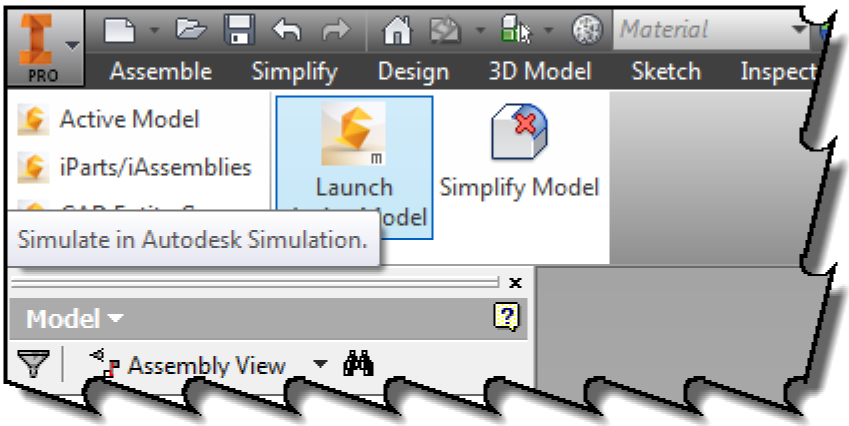
Decision Center, where we can compare our designs and the results quickly and easily using summery images generated during results visualization.



Compare Multiple Views of different simulations for a clear understanding of the side by side performance changes. Now you can go out and innovate your design!

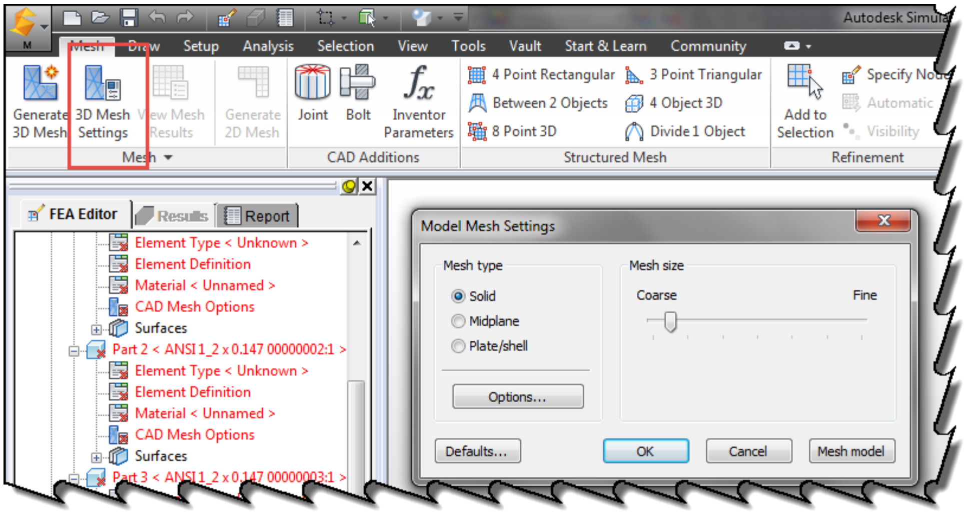
Now that we understand the stresses and forces generated by the wind, let's try to understand how it affects the mechanical performance of our banner. De-select the visibility of the Environment.ipt in the BANNER.iam.

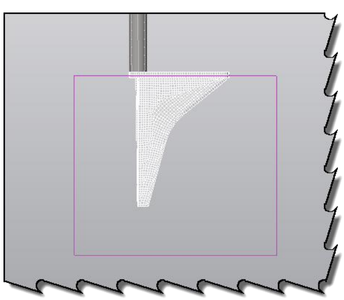
Let's send the model over to Autodesk Simulation Mechanical and directly import the results from our CFD simulation. You will need to close your CFD model out to accomplish this step.

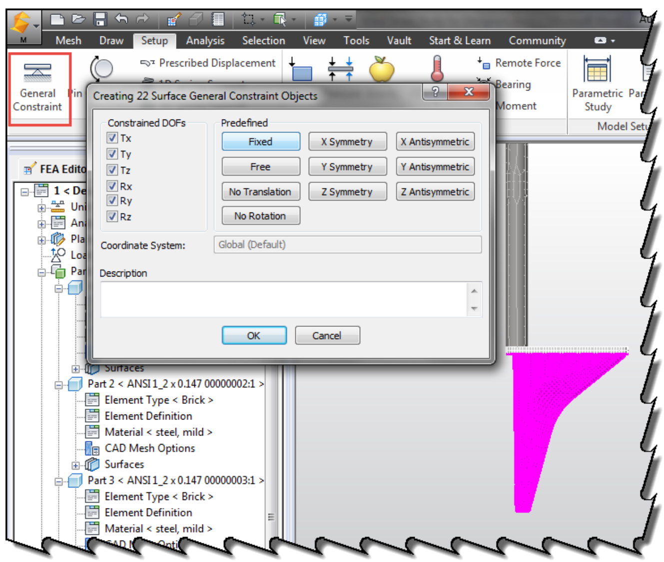


When you have the model open in Simulation Mechanical, you will need to make some adjustments to Materials and Constrains.

First we need to clear out some part(s) that are not needed / relevant to our study as shown in this case we need to suppress the air volume.

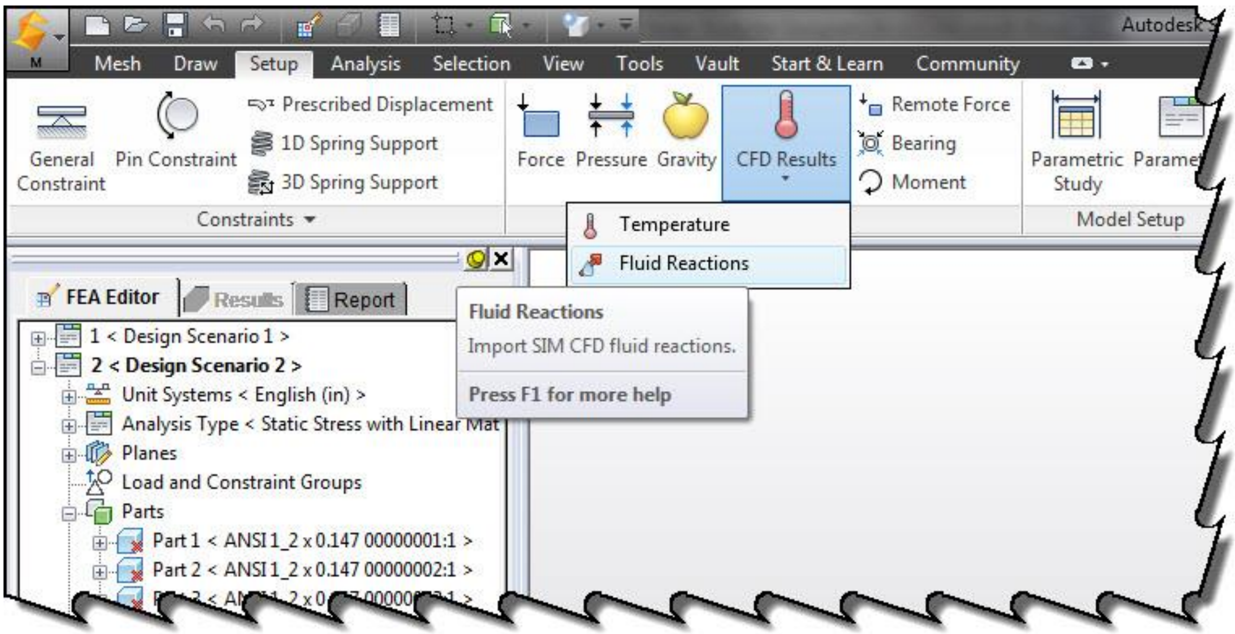


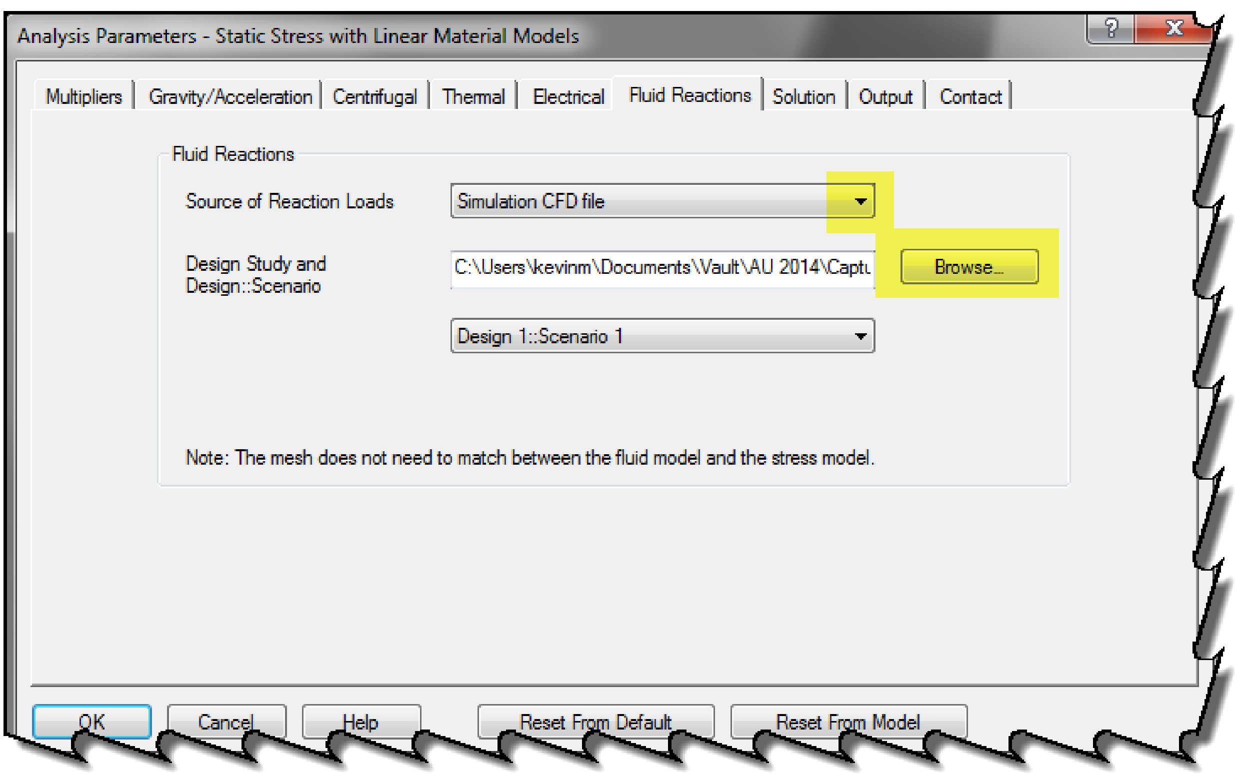




Window select the geometry as shown above, the surfaces to be held in place by solid material (Dirt, asphalt, Concrete etc..) and apply a General Constraint of *Fixed* and choose “OK” and fix them in place.

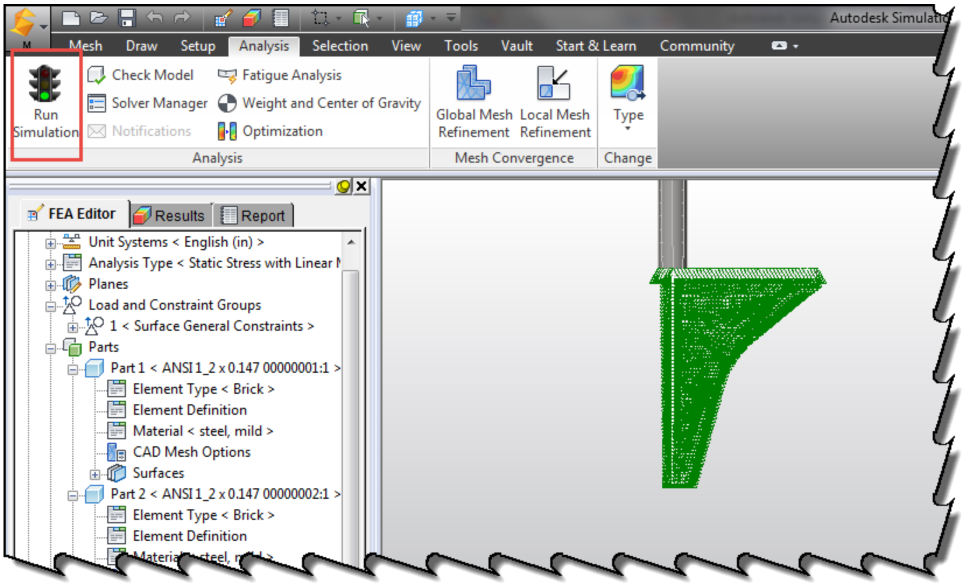
Now That we have Simulation Mechanical setup, we will bring in our results from our CFD Study.





When you Browse for the .cfdst file that will be in your working class folder, you will see the results for both Scenario's select *Scenario 1* and then **OK**

Now we are ready to Run Simulation!



(An archive file with previously run results is included in your working class folder)

Let's explore the results that Simulation Mechanical has found. If you would like, feel free to explore design solutions.