BES469281

Fusion Electronic Cooling, not just for Electronics

Dave Graves
Autodesk

Learning Objectives

- Learn about the basic requirements for setting up a problem in Electronics Cooling in Fusion 360
- Learn about adapting Fusion Electronics Cooling for other applications
- Learn good practices for setting up flow problems not related to Electronics Cooling
- Discover different flow results within Fusion Electronics Cooling

Description

Fusion Electronics is an exciting new technology purpose build for Electronics Cooling. This session will explore practices for using the technology for other types of air flow problems.

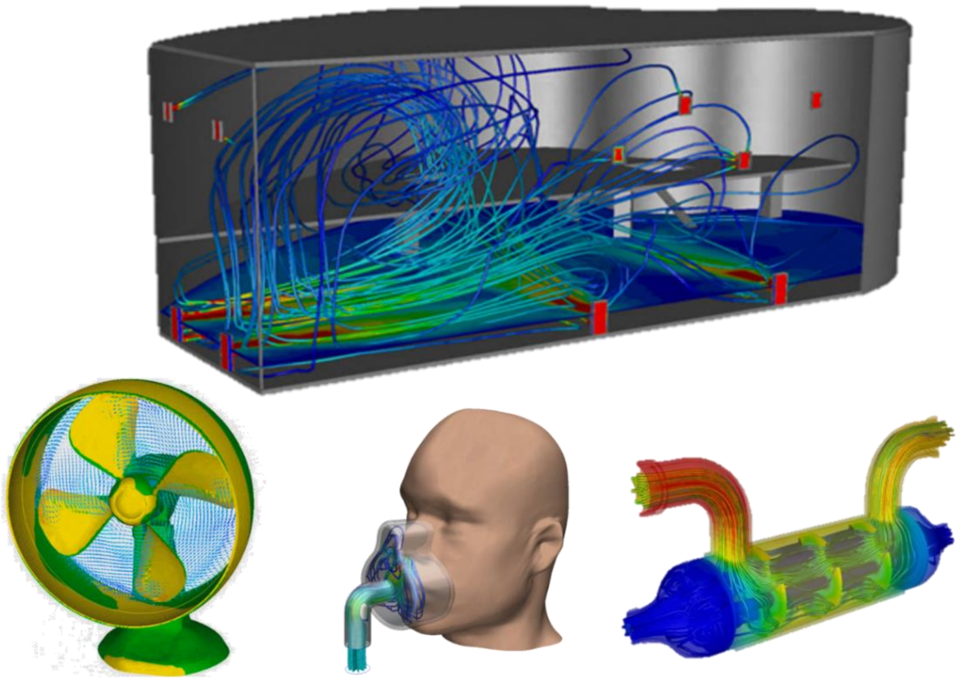
Speaker: Dave Graves

Dave is a Sr. Technical Specialist on the Autodesk Owners and Ecosystems team. He has a BSME from North Carolina State University and has spent time in Telecommunication industry before moving to the world of Simulation and CAD. At Autodesk Dave has worked with various manufacturing solutions including Fusion 360 and has been involved with the Autodesk CFD product for over 15 years.

Dave.Graves@Autodesk.com

Why

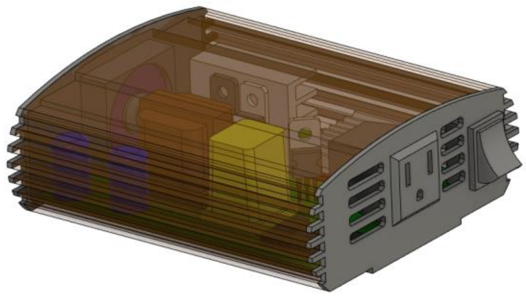
Understanding air flow during the design phase of project can help you make better decisions. This can include product designs like valves and manifolds, as well as building designs. The Autodesk CFD product can solve many types of flow and thermal problems. As Autodesk continues to develop the Fusion platform, Fusion Electronics Cooling (FEC) has been developed and is currently a 'Pre-View' technology inside of Fusion. While FEC is purpose built for electronics, this document will explore how the technology may be expanded to other industries and products.



CFD Examples

Basic Problem Set-Up in Fusion Electronics Cooling

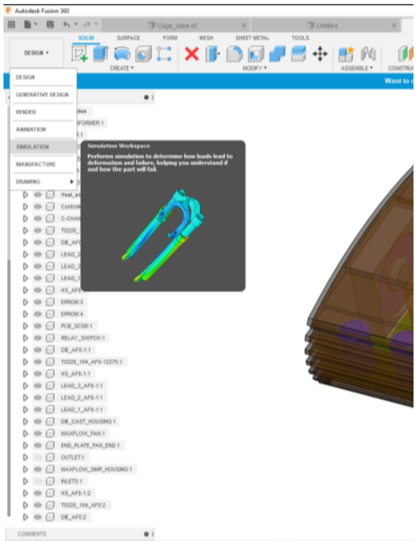
The following example will go through some of the basic steps in setting up a successful Fusion Electronics Cooling simulation. The first example is a simple Inverter example. It has a fan and vents to help with cooling the internal components. Additionally the housing itself has fins that help dissipate heat into the external environment.



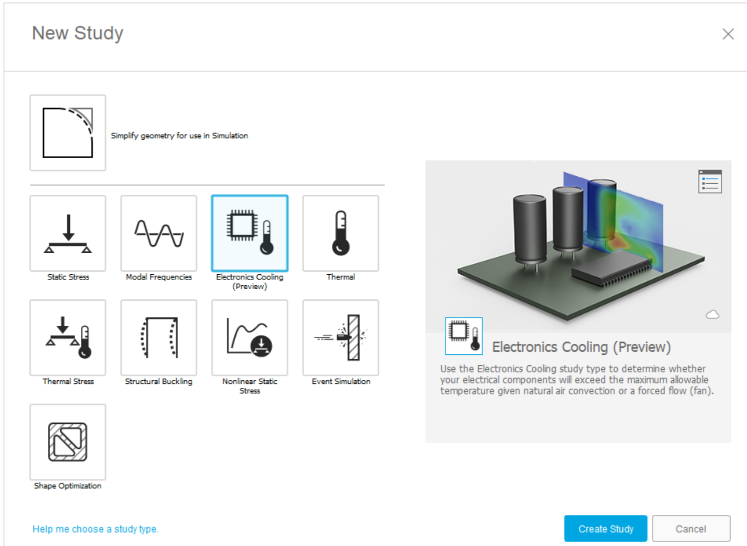
Inverter Example

Step 1

The first step is to lauch the model into the Simulation Workspace, then chose the Electronics Cooling (Preview) option from the menu.



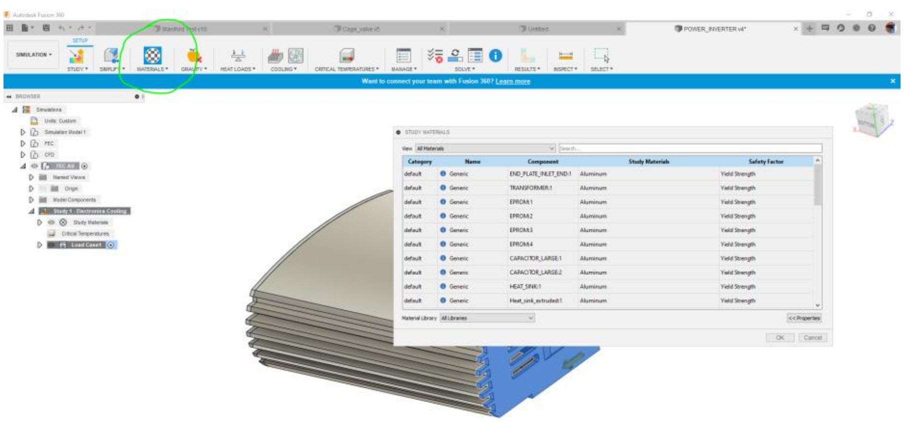
Simulation Workspace



Electronics Cooling Study Selection

Step 2: Assign Material

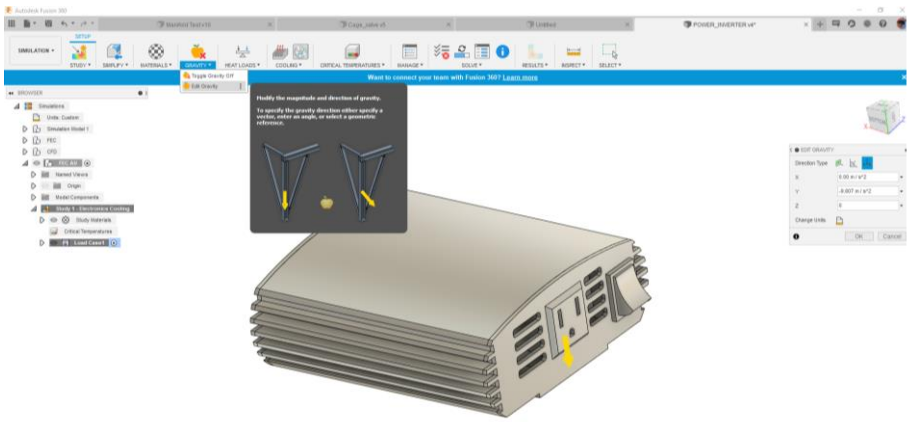
All of the solid parts in your model should be assigned a material. If they are already assigned in Fusion, the simulation will use those assignment.



Material Assignment

Step 2: Turn on/off Gravity

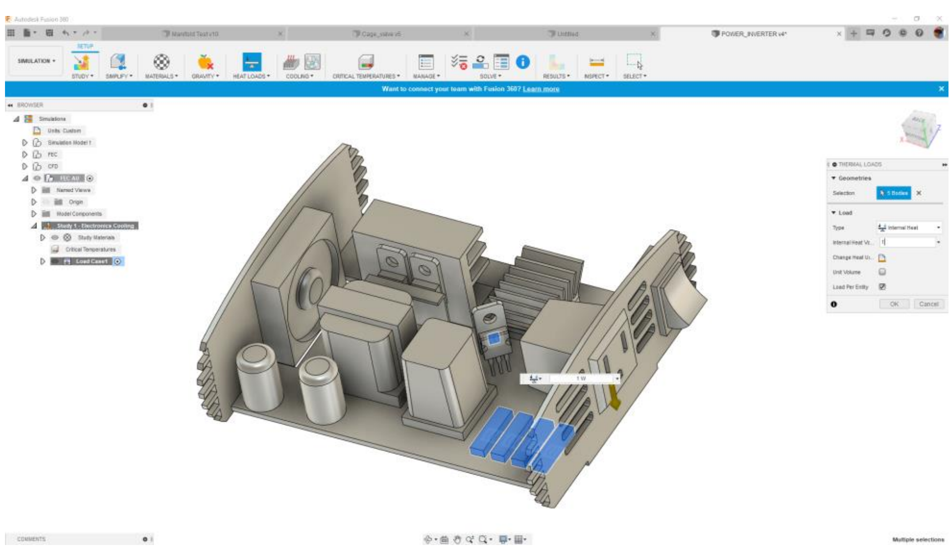
Depending upon the type of simulation you are doing, you can turn gravity on or off. If you chose to turn it on, you can specify the direction. Quite often with a force flow problem, you can keep it off but again it depends upon what you are trying to accomplish with your design.



Gravity Set Up

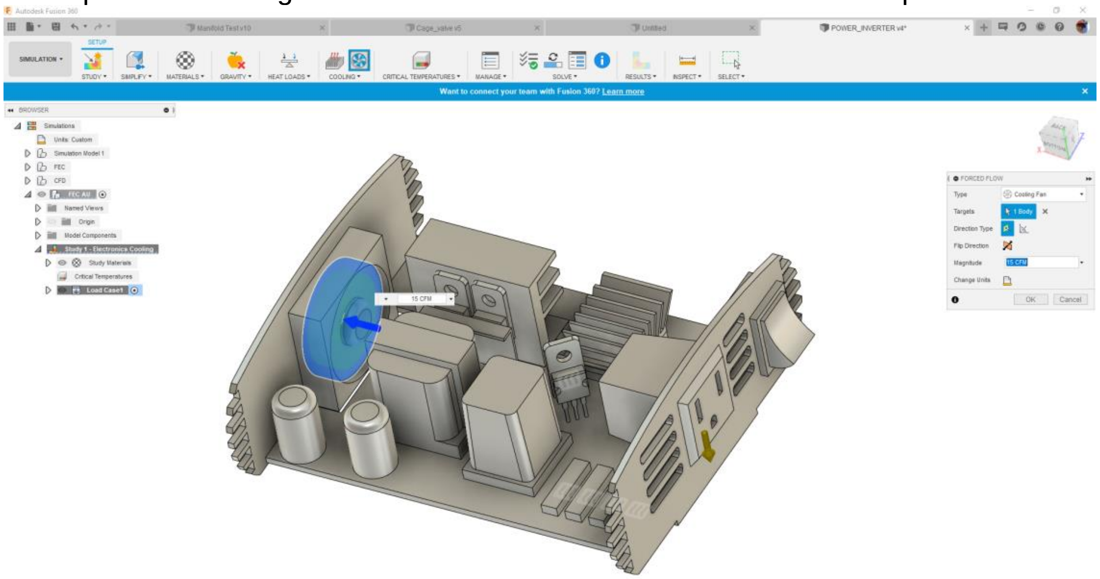
Step 3 – Assign Heat Loads

Assign values to the components that are dissipating heat.
NOTE: The FEC Solver requires that heat loads be assigned to at least 1 solid in the model.



Assign Heat Loads

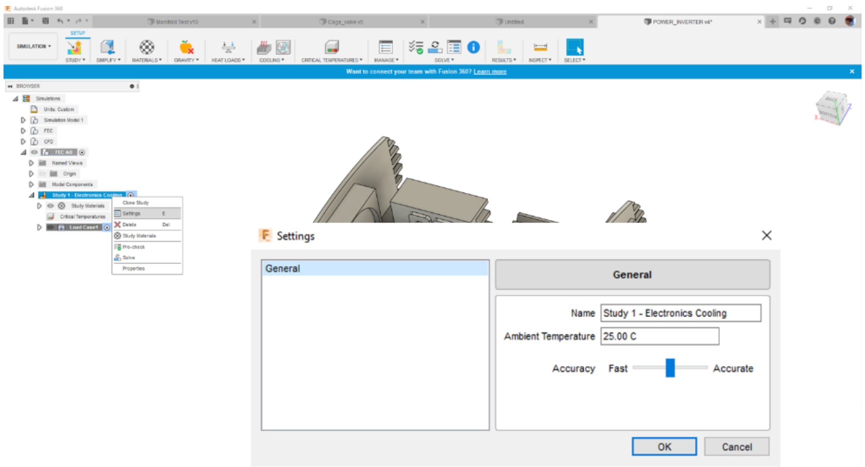
Step 4: Assign Fan Values
In this case we have a fan generating air flow to help with cooling. The FEC technology is capable to running natural convection simulation that does not require a fan.



Assign Fan Properties

Step 5: Set the Accuracy

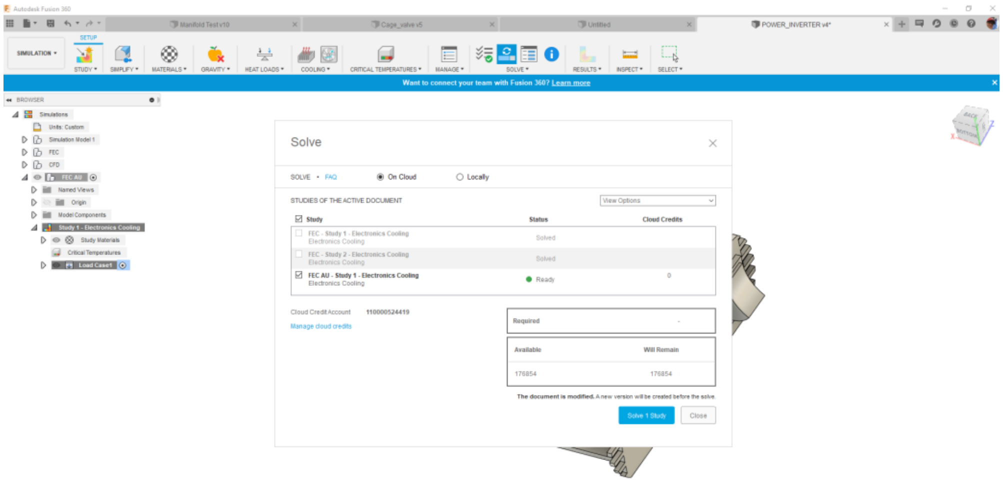
Under the settings tab, you can change the accuracy. For this model the default setting is appropriate. For the additional models shown in this document, the slider should be positioned all the way to the right (Accurate). This will take longer to solve but should get better results for non-FEC type solves.



Solver Settings

Step 6: Start the Simulation

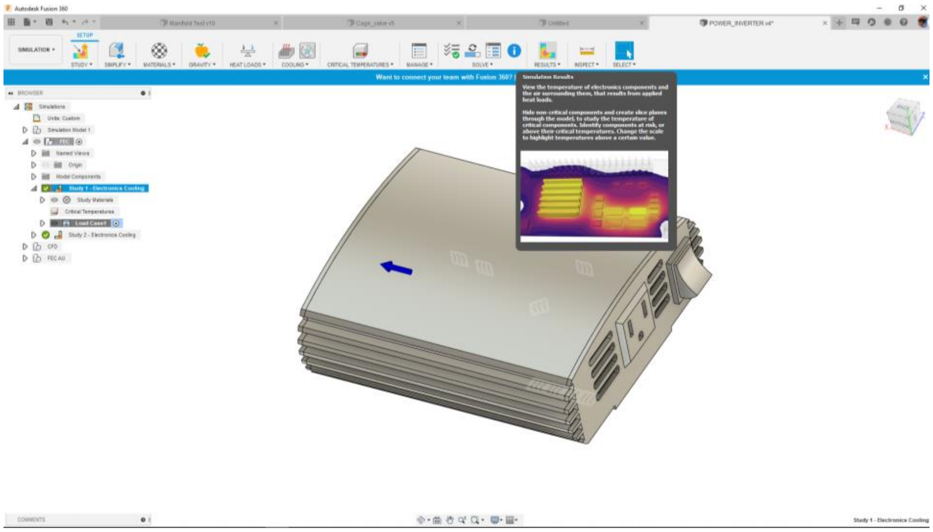
If the FEC solver has all of the information required, you will see a green check box. Currently all solves are done in the cloud, however no cloud credits are charged. Upon the comericialization of FEC, this will require cloud credit consumptions.



Start Cloud Solve

Step 7: View/Interogate the results

After the simulation is complete, the results tab will be visable. Currently the FEC product is only able to display temperature and velocity.
Note: There are no vectors for velocity



Enter Results Mode

Section Planes are another way to visual the results. These planes can be moved throughout the entire model. The image below illustrate the temperature profile just above the PCB.

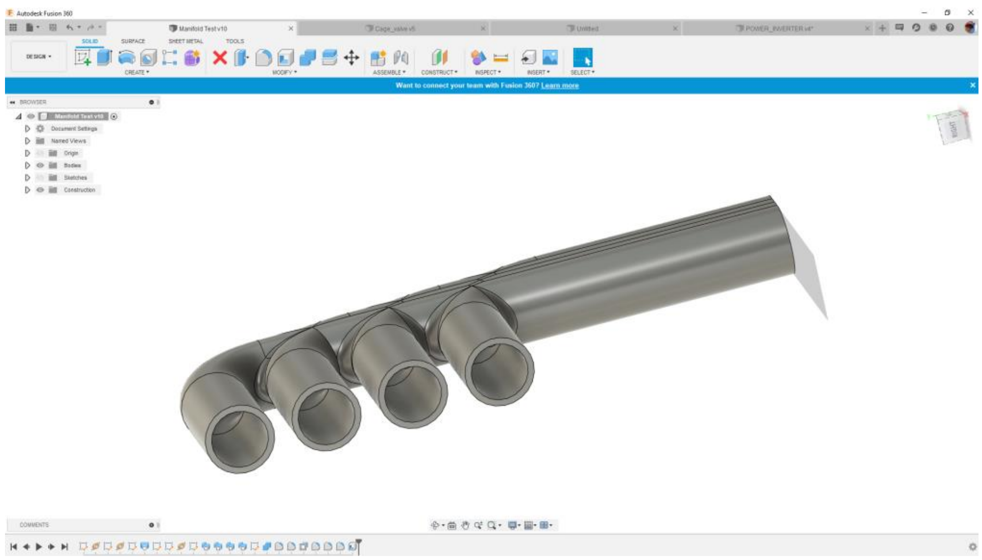
Sectional View Showing Temperature

Adapting Fusion Electronics Cooling for other applications

The next section will go through a couple of examples on how FEC could be used for ‘non-electronic’ examples. Again, FEC is not a full scale CFD tool, however there are ways to gain tremendous insight into product design leveraging this technology.

Manifold Example

The following is a simple manifold example. FEC was not developed to support this type of simulation so some changes in geometry need to to occur to support this. Even though the results visualization may be limited, they can still provide value to an engineer or designer.



Manifold Example

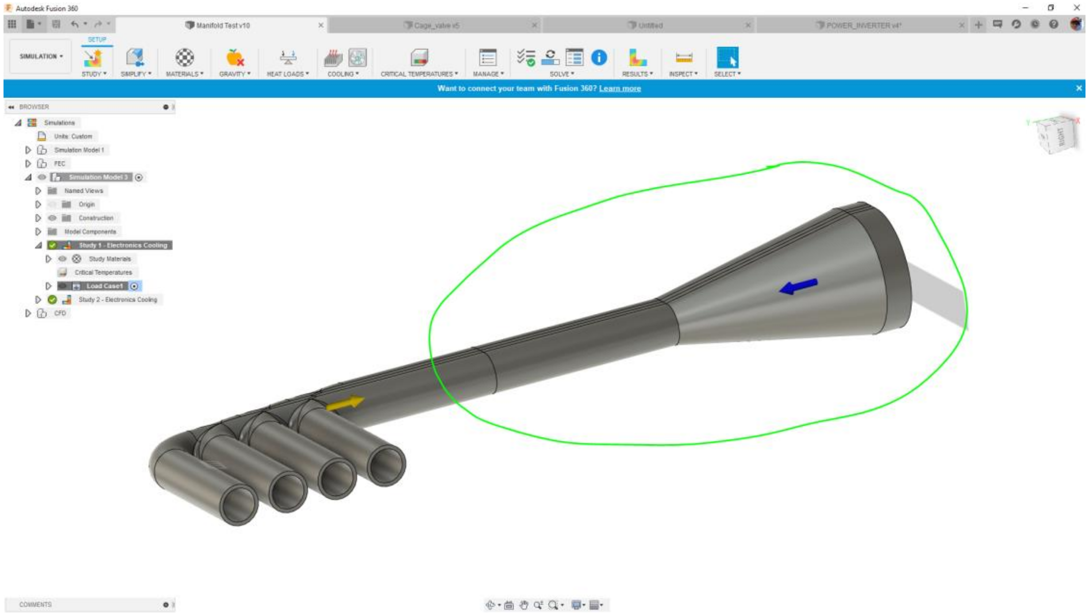
Step 1: Geometry Modification

With FEC, there is not currently the ability to specify a uniform velocity or even a flow rate on the model. Any air flow must be driven by a fan. The fan velocity profile that FEC generates include a center hub and is not a uniform output. To help product a constant flow rate through the inlet of the manifold, some geometry changes are required.

If you notice the model below, there have been some changes to help drive the flow.

1. The inlet has been extended to allow more time for the flow to develop
2. There is a taper created (Large to Small). By doing this the idea is that the high velocity from the outer diameter of the fan will be somewhat forced towards the center to help mitigate the effect of the fan hub.

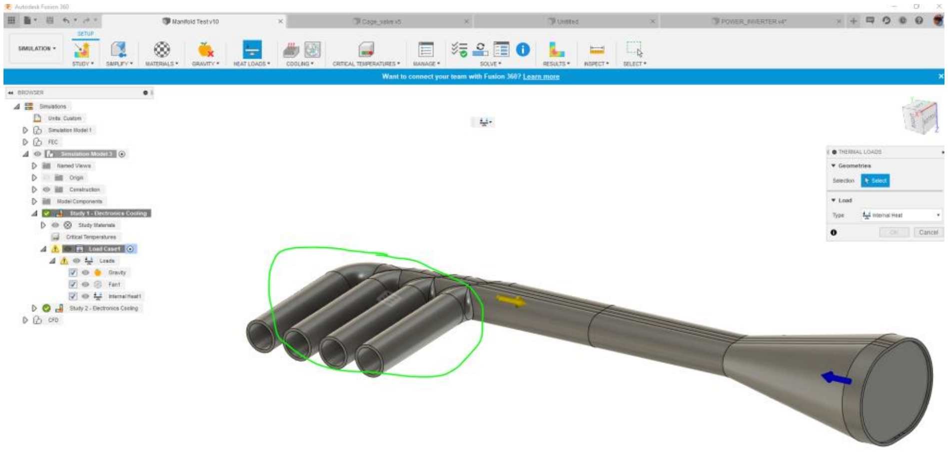
Again, FEC is not a full blown CFD tool, so these are some changes to help you get reasonable results.



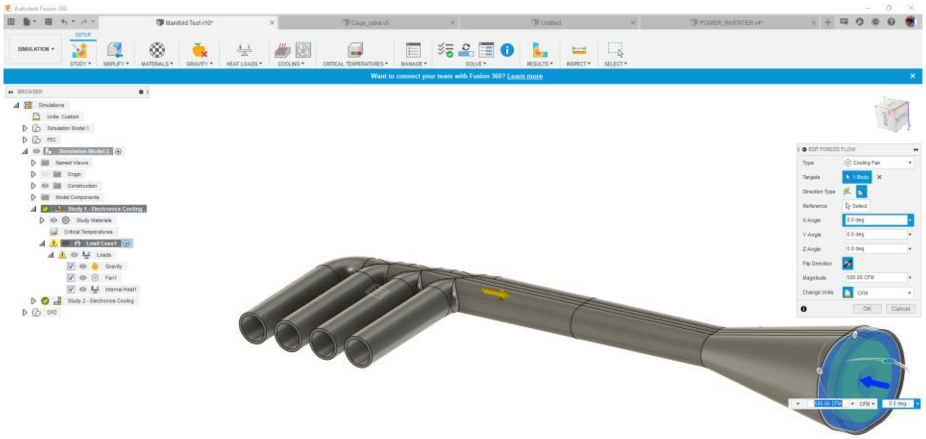
Geometry Changes to Manifold

Step 2: Assign Heat Load

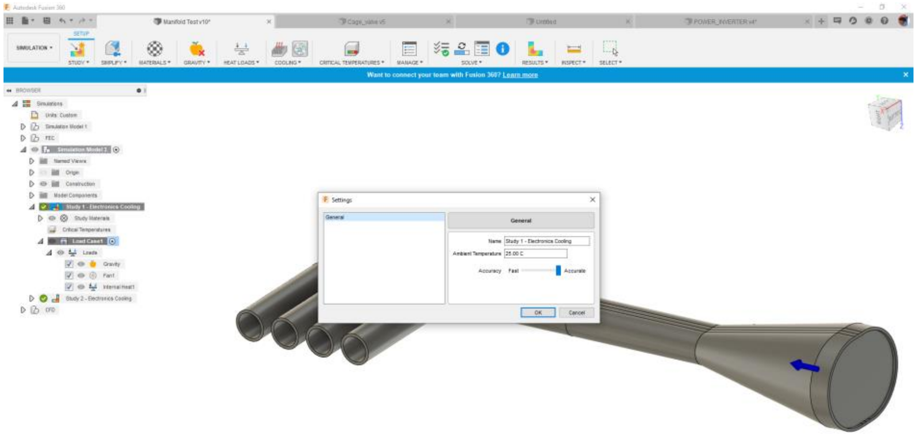
FEC will always require a heat load. In this case, we are not concerned with thermal so an arbitrary head load is assigned to the solid portion of the manifold.



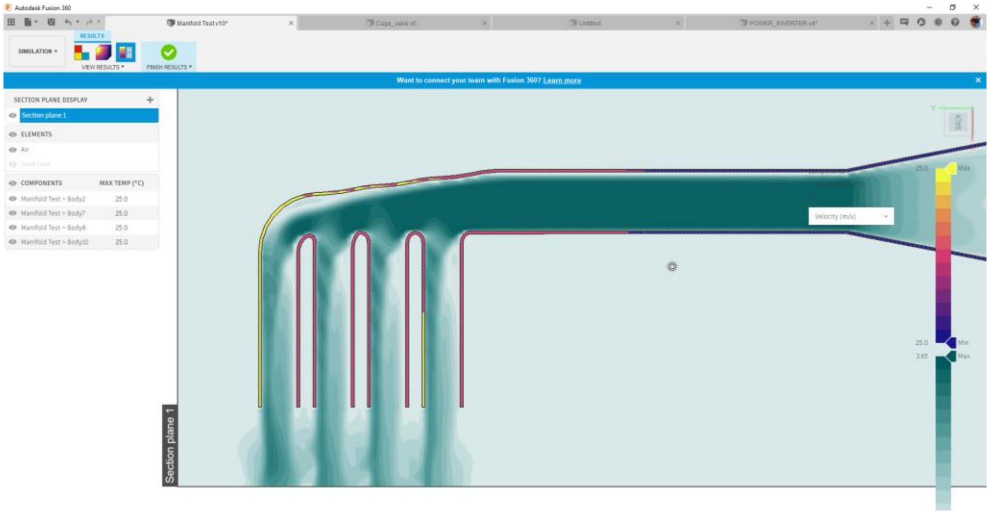
Arbitrary Heat Load



Fan (Momentum) Assignment



Accuracy/Simulation Settings



Velocity Results

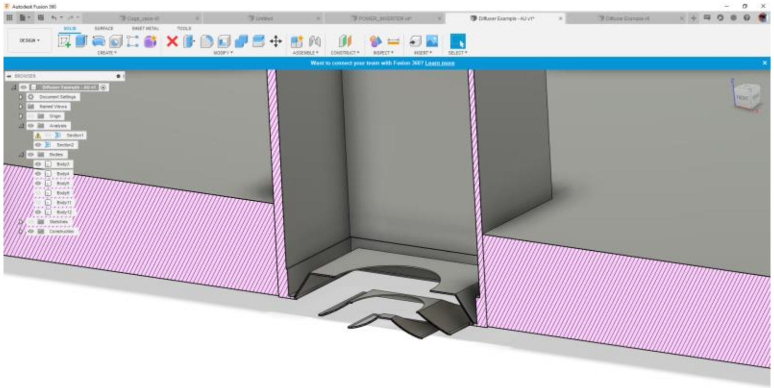
Diffuser Example

The next example is air flow through a diffuser. This may be valuable to understand flow patterns and how much coverage they might provide. Some of the challenges with this example is the thin geometry of the diffuser itself as they are typically constructed from sheet metal.

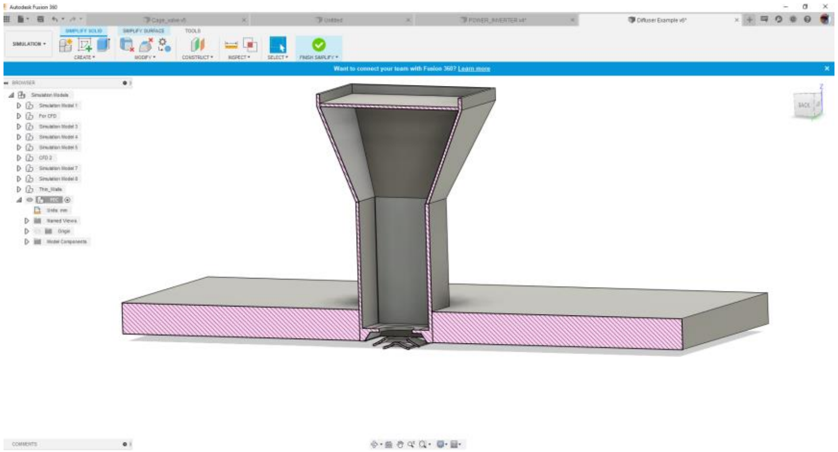
Step 1: Modify Geometry

You can see how thin the geometry is in the image below. Due to the way the FEC generates it's simulation model, there is potential for the thin walls of the diffuser to actually be ignored. One of the ways to help prevent this is to artificially increase the thickness. Again, while this may not be the exact representation of the diffuser, our goal is to gain insight

Additionally the inlet to the diffuser was modified in a similar manner to the manifold example above to make the fan larger than the ductwork. The ductwork transitions to the standard HVAC inlet vent in an effort help the flow develop, rather than take the profile of the fan.



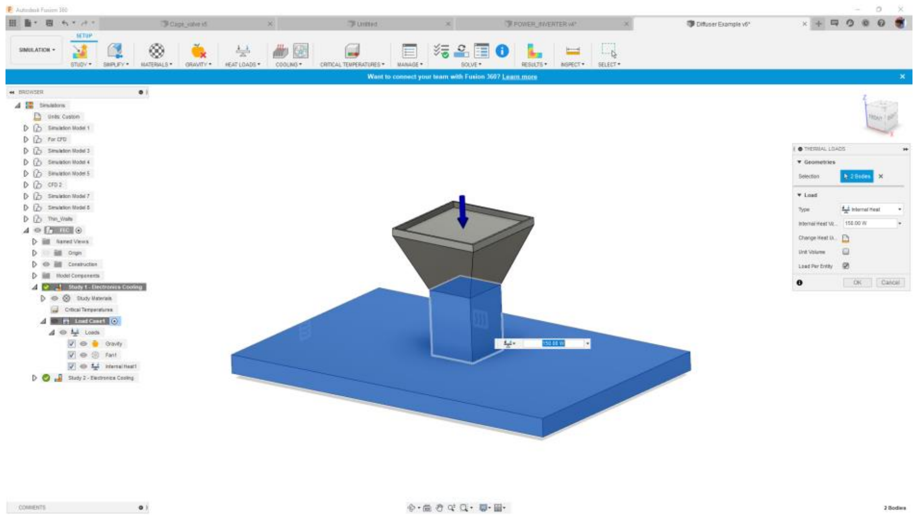
Diffuser Geometry



Diffusion Geometry Changes (Thicker)

Step 2 – Set up the model

Even though we are not concerned with heat generation, we need to specify and arbitrary load. In this case it can be ductwork and/or even the ceiling



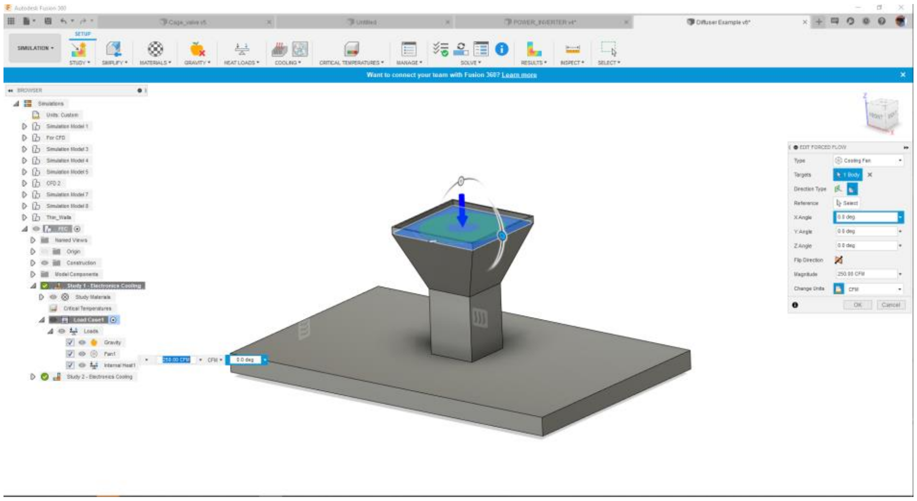
Arbitrary Heat Load

Much like the previous example we need a momentum source to generate flow. This is done by assigning the fan a flow rate.

Note: You can see how the fan will assume there is a hub and be circular in nature even though the ductwork and fan is square.

Since I'm not really worried about the thermal performance, there is no need to change any of the material assignments. The model should be ready to solve.

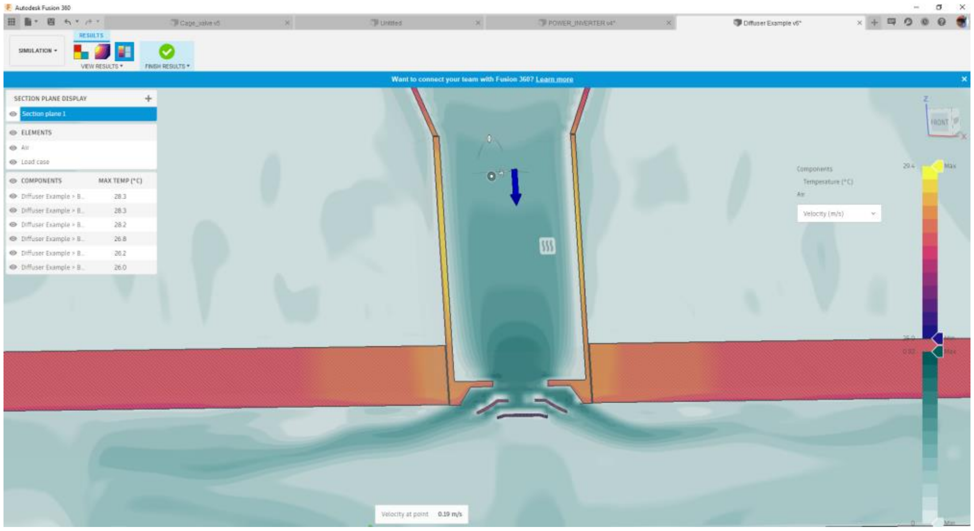
.



Fan Setup

Step 3: Interrogate the Result

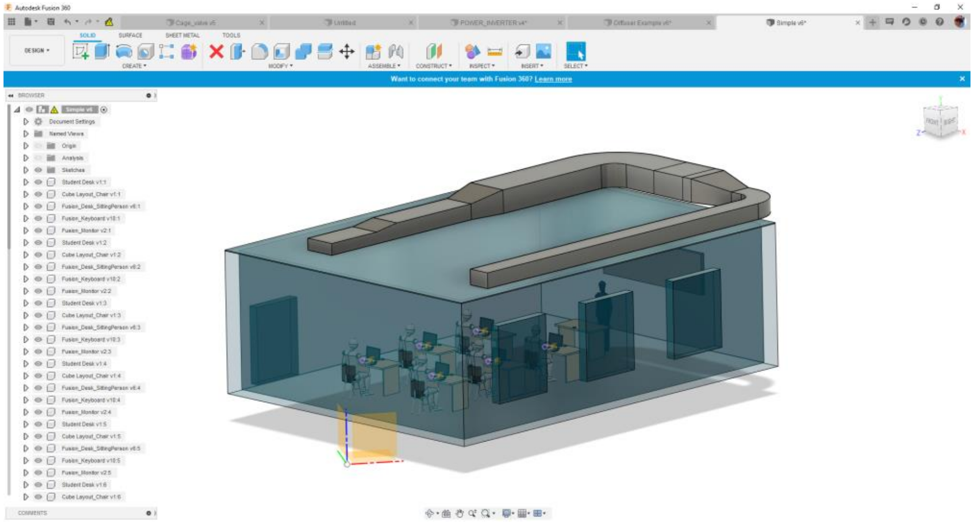
If you notice below, you can the velocity of the air going through the diffuser. This is best accomplished by adding a section plan to the view.



Diffuser Results

HVAC Example

The example below can help us understand basic flow and temperature of people and electronics in a small room. Due to the voxel based approach the FEC solver users, there may be challenges with larger rooms.

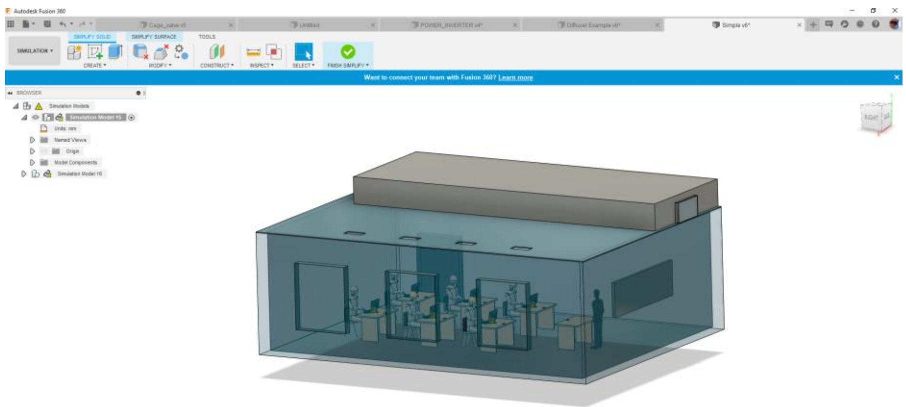


HVAC Example

Step 1: Geometry Modifications

In this model there were a few changes that need to made to get reasonable results.

1. The Diffusers need to be removed due to the thinness of their design
2. The inlet area of the HVAC system was thickened so the FEC solver could recognize the walls. This consisted of adding a rectangular box around the ductwork, then utilizing the combine command to remove the internal void of the ductwork itself.
3. A fan was added to prove a flow source

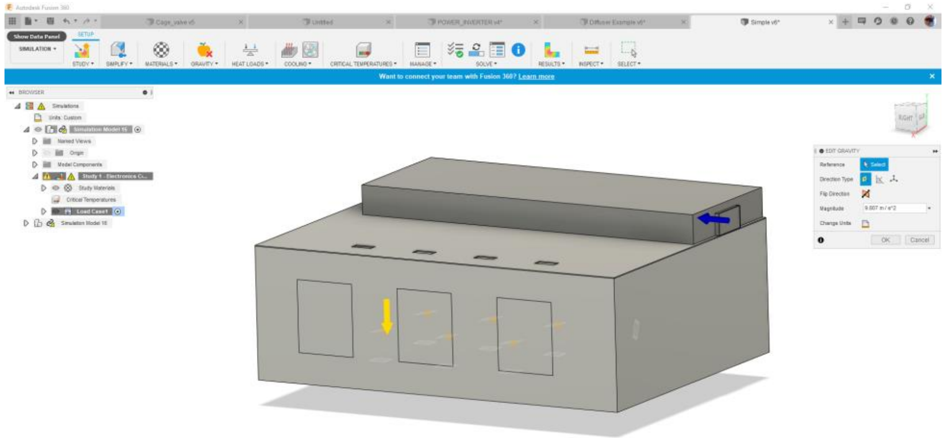


Geometry Modifications

Step 2: Set up the Model

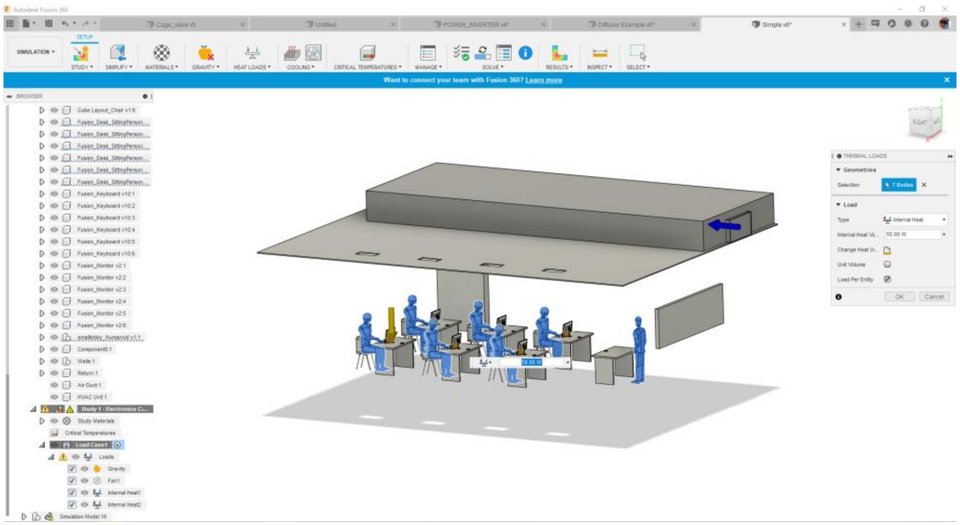
In this model, solid material should be changed to represent the actual real world conditions. For example the walls should be specified as concrete, the desks wood, etc.

Additionally natural convection does play a significant role in the phsyics of cooling this classroom example and gravity should be turned on an specified in the appropriate direction.



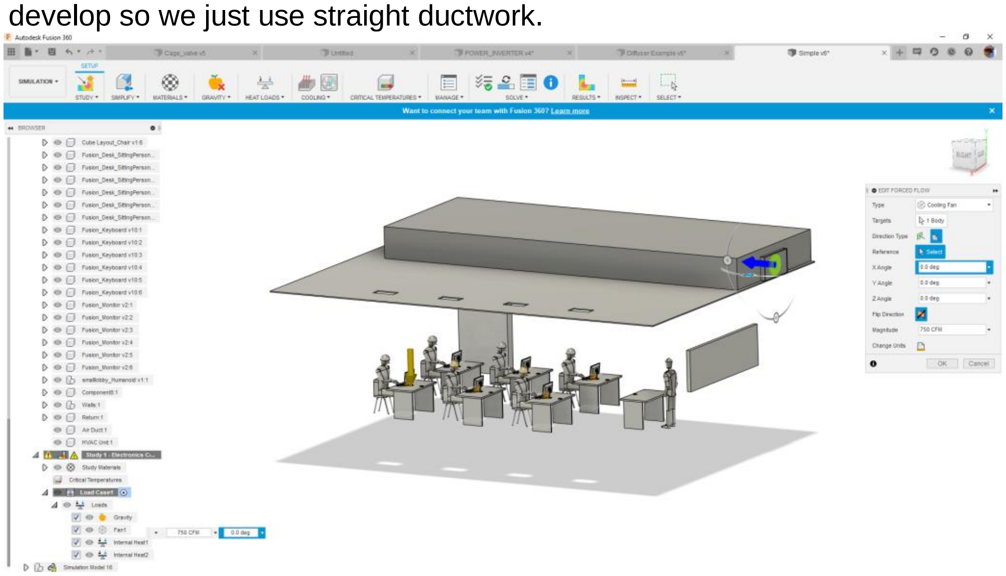
Model Setup

The heat loads are a little more critical and need to be assigned appropriately. In this case we are assuming each individual dissipates ~50W and the computer (keyboard) dissipates ~25W



Model Setup (Heat Loads)

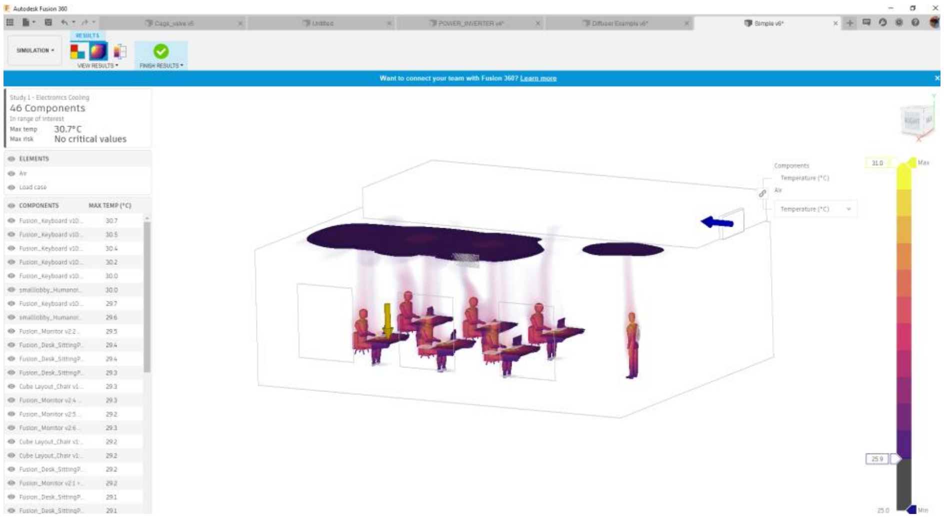
The flow source is a fan. Due to the model we could have had the HVAC system transform like previous examples but in this model there is a little more room for the flow to develop so we just use straight ductwork.



Model Setup Fan Assignment

Step 3: Interrogate the Results

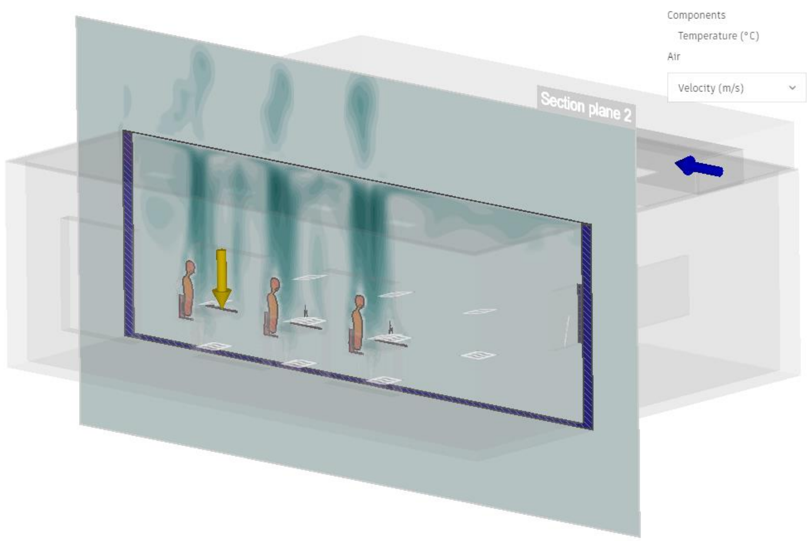
Using the temperature scale you can see some of the hotter areas in the room   This includes the people and laptops as expected.



Temperature Results

Using a Section View you can visualize the temperature or velocity on plane in different locations and orientation in the model.

Temperature Results Sectional View

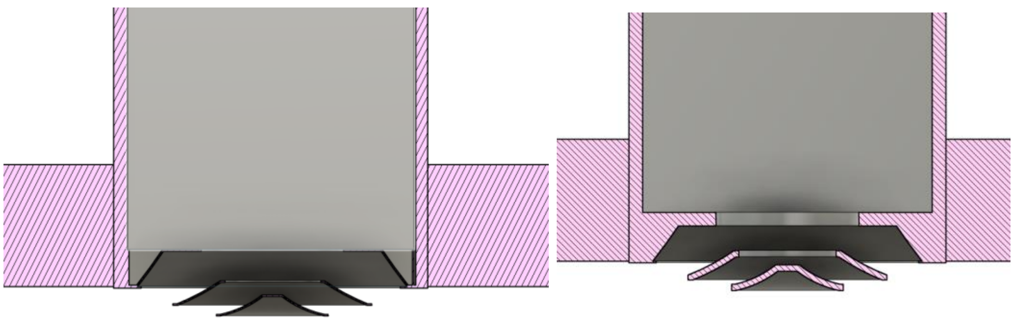


Sectional Results Air Velocity

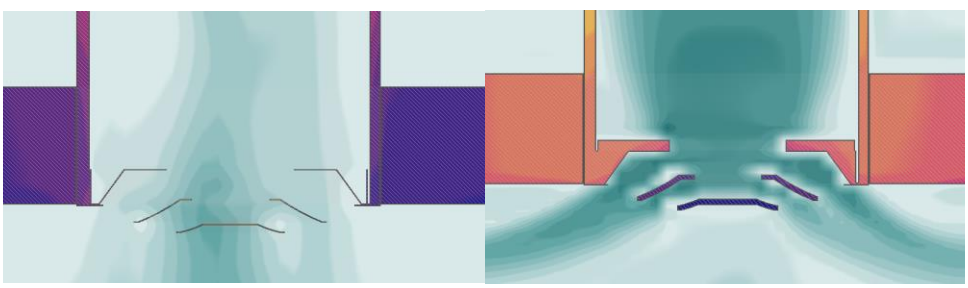
Good practices for setting up flow problems not related to Electronics Cooling

Remove Small Geometry:

The FEC technology basically use a grid to generate the simulation model. In the case of thin geometry the grids may not be dense enough to capture the solid wall. By making the walls thicker, the solver can account for this.



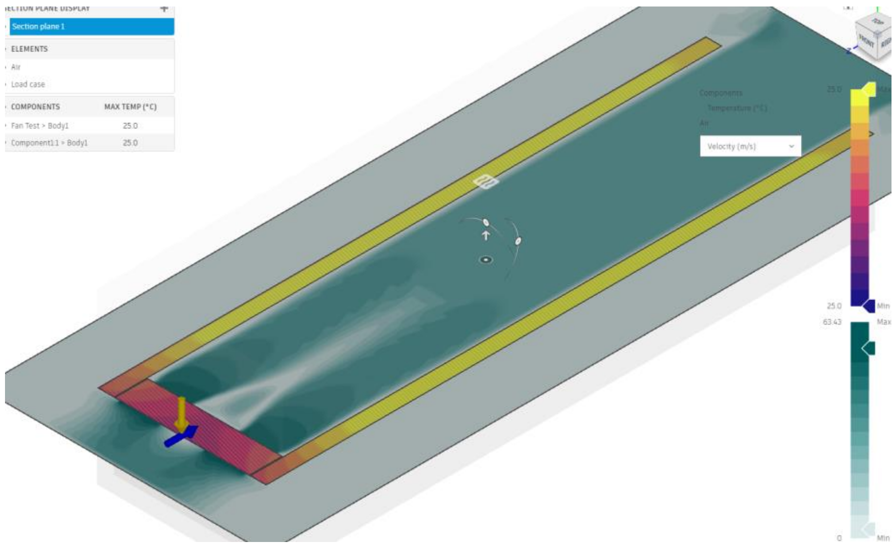
Diffuser Modifications



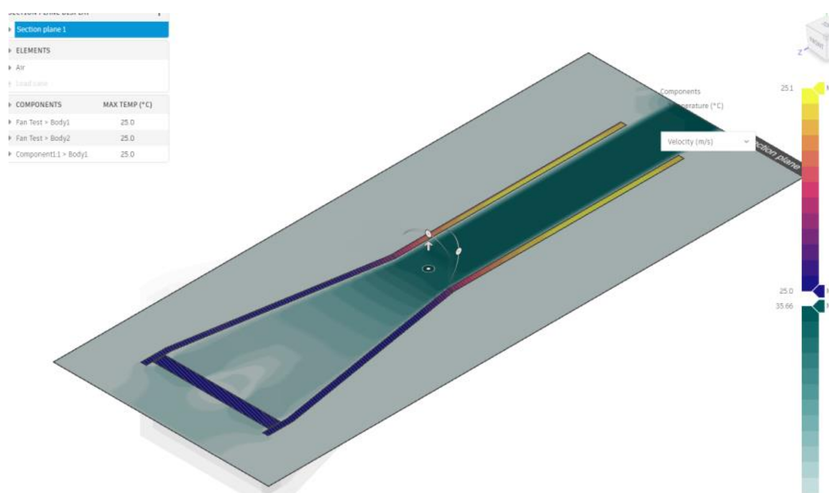
Thin vs Thick Velocity Results

Create Transitions for Fan to help flow develop:

The fan material in FEC is the only mechanism to generate forced flow. The drawback is the fan has a velocity profile that includes the hub of fan. This is more noticeable in large models. You see below the tranisition I've creatd helps the velocity normalized a little bit better.



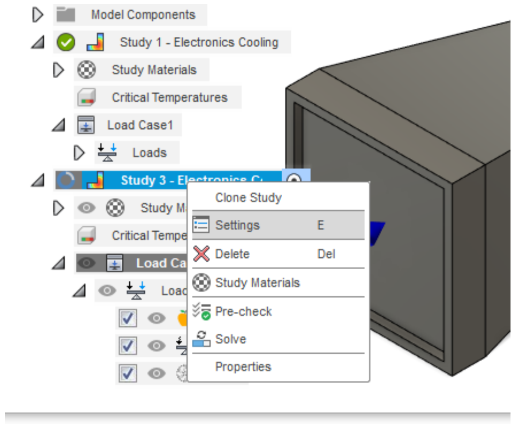
Velocity Distribution No Modifications

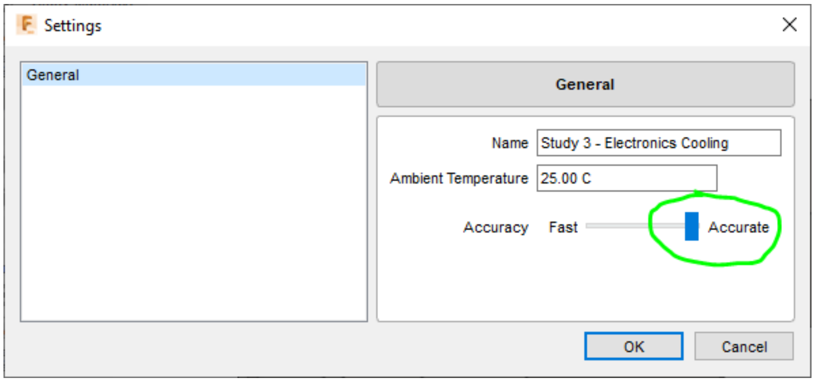


Velocity Results with Modification

Always use the most accurate settings

Due the nature of these non-electrical simulations, the most accurate results should help with some of the challenges. It may take longer to solve but is still recommended.





Simulation Settings

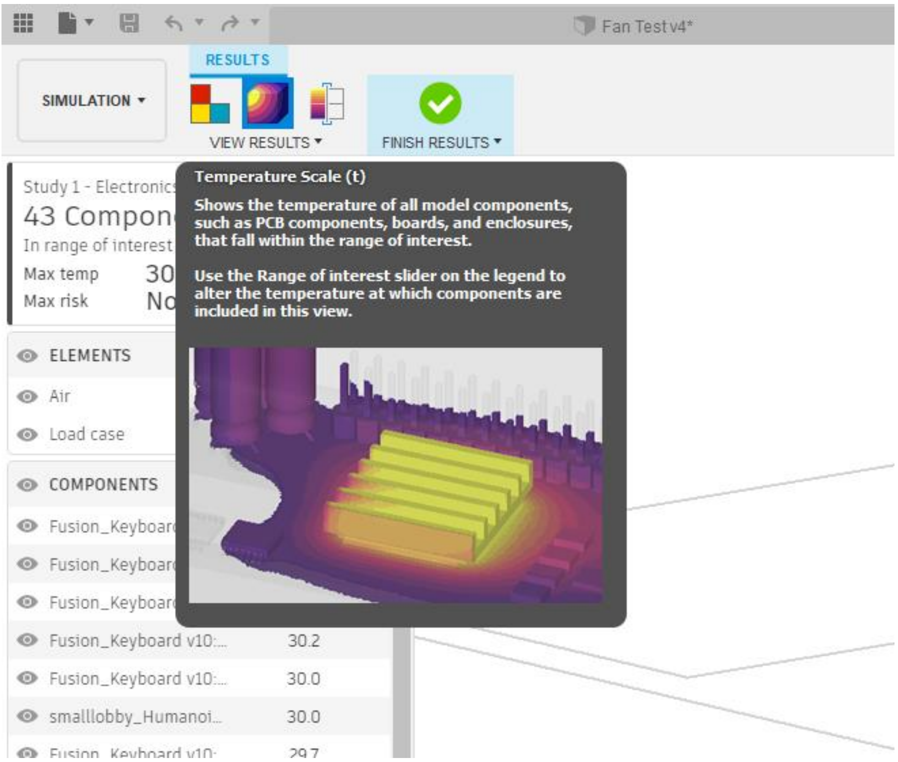
Discover different flow results within Fusion Electronics Cooling

Remember FEC was developed for understanding electronics cooling. Most of the visualization is centered around themal performance.   Today, the technology is able to visualize velocity magnitude, but does not have directional information such as vectors. FEC does not include other information such as pressure drop.

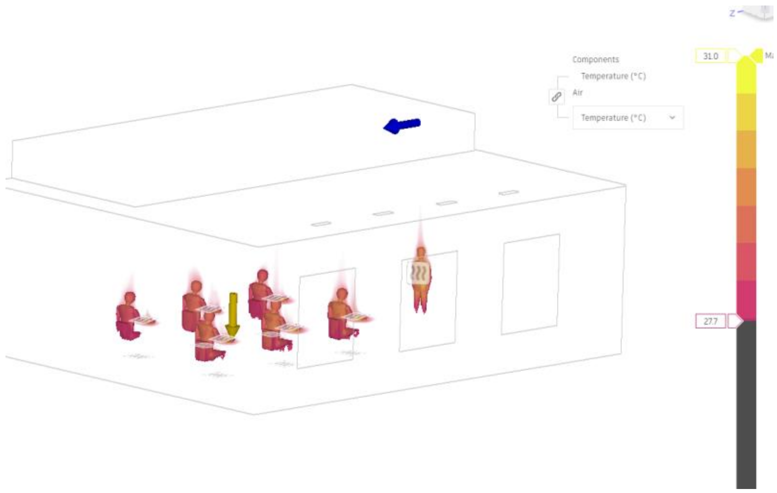
Temperature Scale Results

This mode will display different ranges of the results. In the first image, if temperature is chosen, the user can move the slider in the temperature scale to isolate areas within a given range. It is a great way to identify hot spots in a model or where temperature exceed certain values.

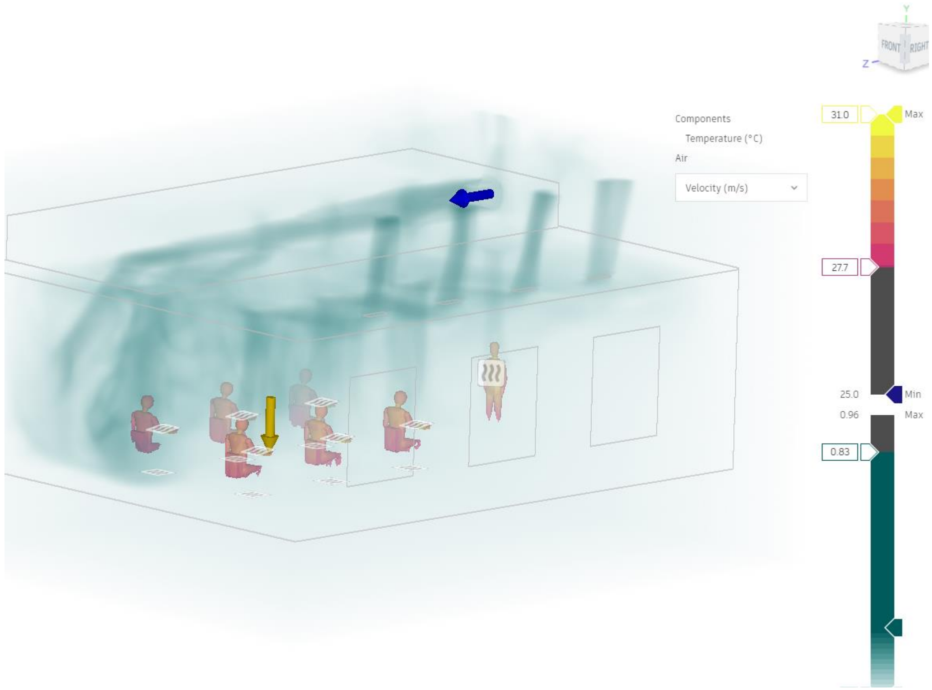
If the user changes to Velocity, a similar results appears however the technology is showing a velocity within a given range. It does not show direction but can be usefull to see if there are dead spots or high glow areas within a model.



Temperature Scale Option



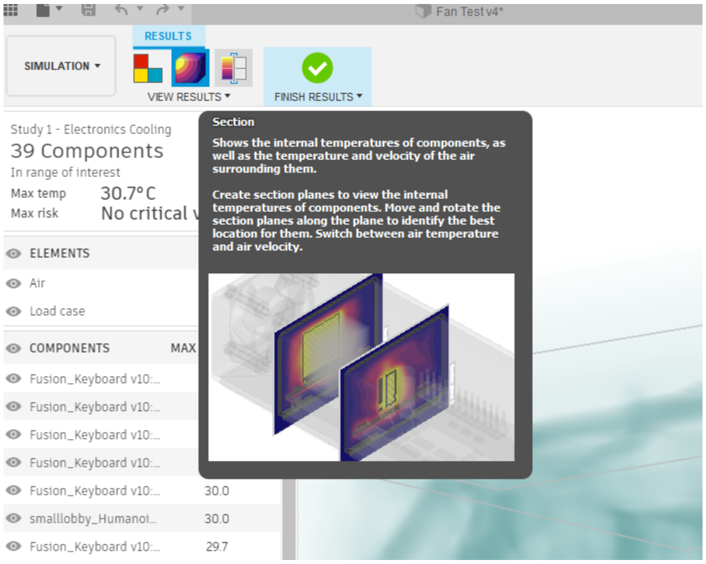
Temperature Visualization



Air Visualization

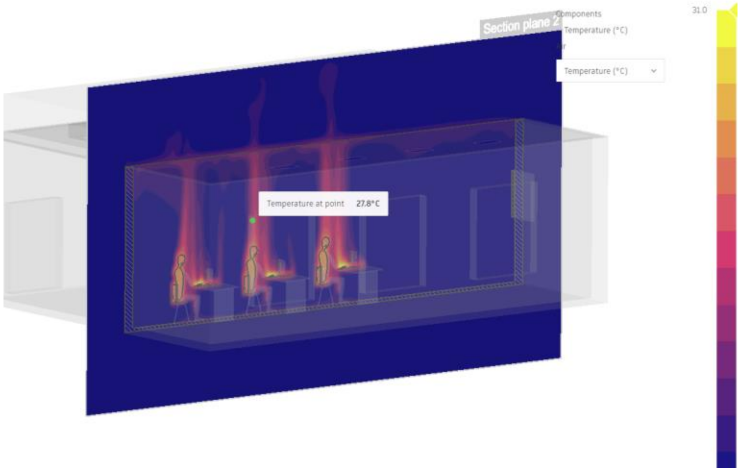
Sectional View

User can also switch to a Sectional View. This view takes the results and places them on a planer surface. The plan can be re-oriented and or slid throughout the model. There is the ability to look at velocity. If you place your mouse on the plane, you can also get numerical values. Much like the Temperature Scale results, the range and scale can be adjusted to focus on certain values.



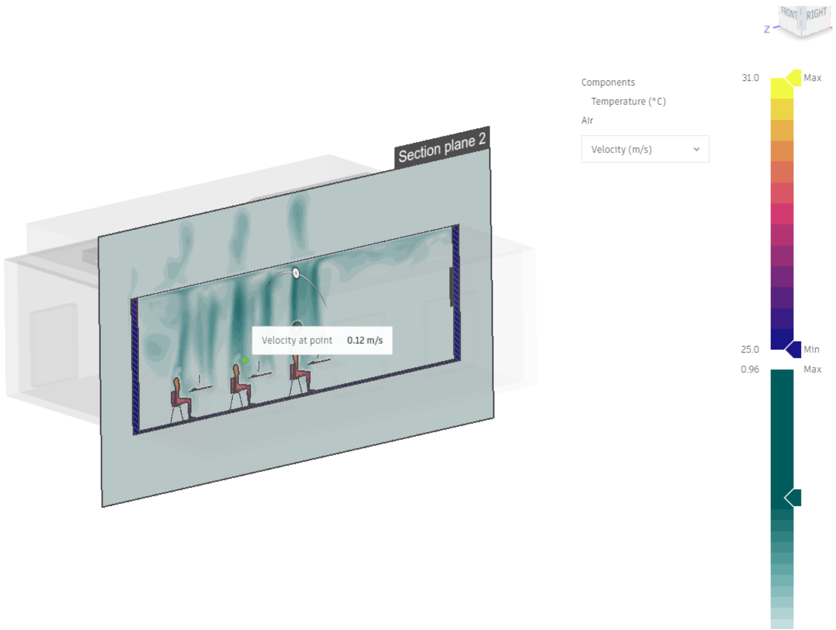
Section Views

By switching to temperature, users can visualize gradients on a plane. They can also place the mouse on the plane to get actual values.



Section View Temperature

There is the ability to look at velocity. If you place your mouse on the plane, you can also get numerical values. Much like the Temperature Scale results, the range and scale can be adjusted to focus on certain values.



Section View Velocity

Summary

While the Fusion Electronics is exciting technology to help customers designing electronics components it is possible to expand the use to other types of problems. Like any simulation tools certain assumptions need to be made and users need to understand this is not a replacement for Autodesk CFD but rather a compliment to give users design insight they otherweise might not have into the performance of their products or HVAC design.