

CP467410

Fusion 360 Case Study: Designing a Lab Supply

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Learning Objectives

- Discover the benefits of e-cooling simulation.
- Learn how to solve complex routing challenges using the sophisticated routing tools in Fusion 360.
- Discover key components of the mechanical design and design the enclosures around them.
- Learn how to solve your own challenges related to your electronic products.

Description

In this class, we'll use the case study of designing a lab power supply. This is a common tool in electronics and by working through its design, we'll highlight how the electronics workspace, e-cooling simulation, and mechanical modeling can all be capitalized on to simplify and expedite the production of this product. As we go through the case study, we'll clearly demonstrate the benefits of having all of these tools in one application—namely, faster product development with fewer rework cycles.

Speaker(s)



Jorge Garcia

I'm a Community Manager for Fusion 360 Electronics/EAGLE, here at Autodesk. I have been working with EAGLE for 10 years. I earned my Bachelor's Degree in Electrical Engineering from Florida International University in 2008. I love electronics and building projects focusing on power and control applications. I have an affinity for embedded systems and programming.



Heath Houghton

Heath Houghton is the product manager for fluids simulation products. As Product Manager, Heath guides the development efforts and roadmap decisions for flow and thermal simulation projects. Heath joined Autodesk with the acquisition of Blue Ridge Numerics CFdesign. He was in a technical role with Blue Ridge Numerics for several years and he continued in that role with Autodesk before transitioning to Product Manager. Heath has over 20 years of experience with both fluids and structural simulation tools. In his spare time, Heath enjoys archery and training his bird dogs.

Design Overview and Theory of Operation

A lab power supply is a vital piece of equipment in the electronic engineer's toolbox. It provides a variable voltage that can be used to power a wide range of prototype circuits. Commercial units include additional features like current limiting, power monitoring, multiple voltages etc. In this class we will be looking at a very basic design, that illustrates how Fusion 360 can be used to speed up the design cycle.

PLEASE NOTE: THE DATASET MAY HAVE ERRORS AND IS NOT COMPLETE IT WAS PREPARED FOR THE PURPOSES OF ILLUSTRATING FUSION 360 IN AN ELECTRONICS AND E-COOLING CONTEXT. WE HAVE PROVIDED A LOT OF DETAIL IN THE DATASET BUT AUTODESK WILL NOT BE HELD RESPONSIBLE FOR MISUSE OF THE DATASET, IT IS PROVIDED AS IS WITH NO WARRANTY OF ANY KIND.

WARNING: LINE VOLTAGES FROM AN OUTLET ARE DANGEROUS. NEVER TROUBLESHOOT THIS CIRCUIT LIVE, ESPECIALLY ON THE PRIMARY SIDE OF THE TRANSFORMER. YOU ASSUME ALL RISK IF YOU CHOOSE TO BUILD THIS CIRCUIT AND AUTODESK WILL NOT BE HELD LIABLE.

KEY ELECTRONICS RESOURCES

- <https://www.electronics-tutorials.ws/> -This site has an excellent series of beginner tutorials in Electronics. If electronics is completely new to you this is where I recommend starting.
- <https://help.autodesk.com/view/fusion360/ENU/?guid=ECD-OVERVIEW> – This is the documentation for Fusion Electronics.
- *Switchmode Power Supply Handbook* by Keith Billings and Taylor Morey 3rd Edition pgs 2.197-2.205
- *Building Power Supplies* by David Lines 2nd Edition Chp 2 and 3
- <https://www.onsemi.com/pub/Collateral/LM317-D.PDF>

How to run the thermal experiment

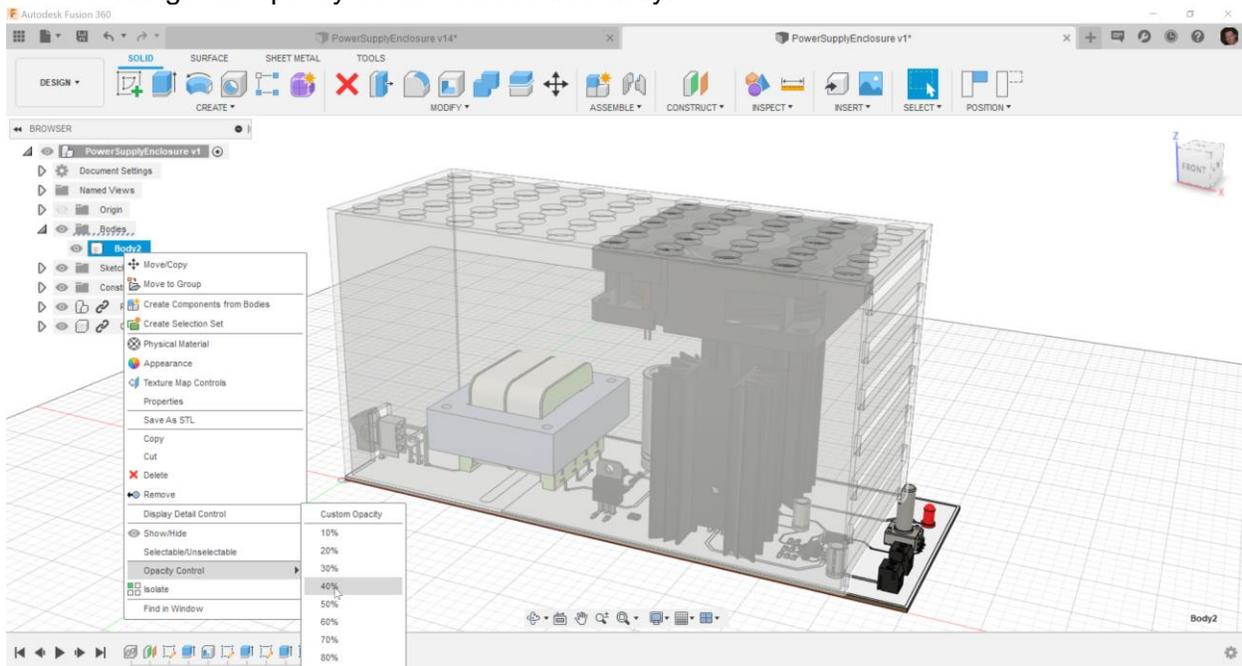
In the lecture video we showed results from 5 different configurations for the electronics assembly. This portion of the handout will take you through step by step how to run the experiment and view the results in Fusion 360.

Setup the experiment

There are quite a few automations built into the integration between the 3D PCB and electronics cooling, but there are a few steps to ensure an accurate representation of the thermal model. On a high level, we need to make sure we have the proper materials assigned, assign the thermal dissipation of the major components, let the model know which way points to the ground, and if we have a fan, assign the flow. There are some optional steps we will also cover. We will walk through all the steps here.

Open the design model

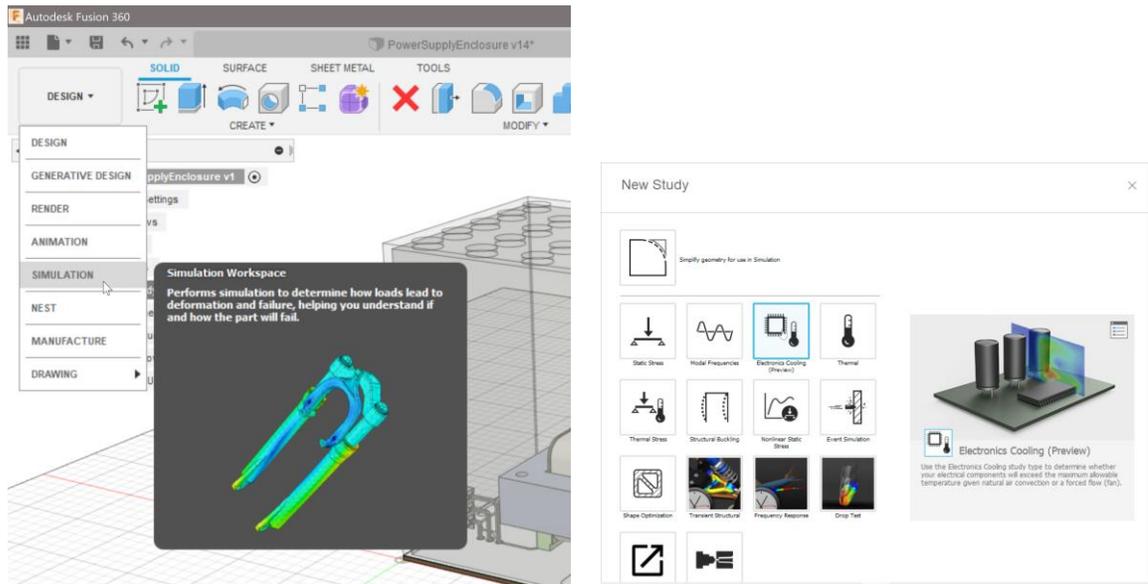
Open PowerSupplyEnclosure using Fusion 360. It will open up in the 3d design environment. You will see the Power Supply 3D PCB and the Fan linked in. There is also a body for the enclosure. It might make things easier to RMB on "Body2" and change the opacity so we can see internally.



Design Model.

Enter the electronics cooling space

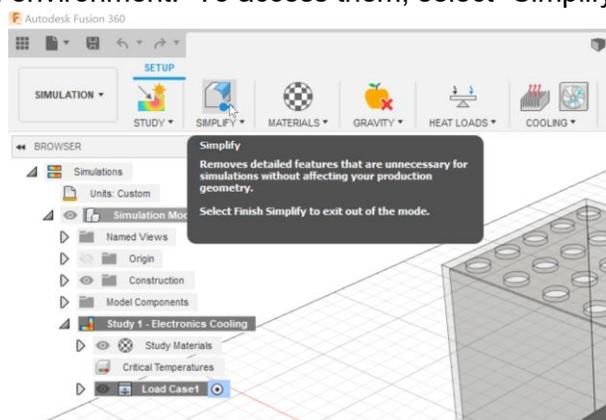
Select Simulation, then select Electronics Cooling as the study type.



Entering the Electronics Cooling Space

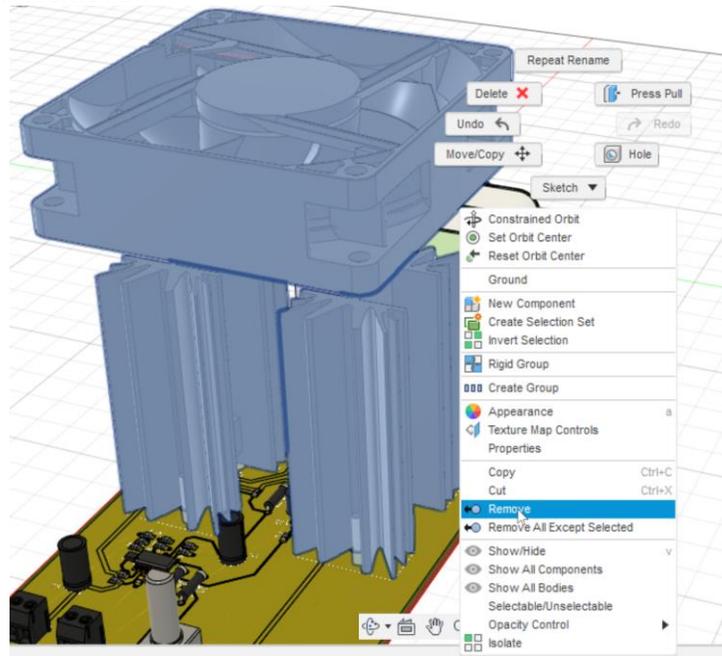
Alter the geometry

Electronics cooling uses a derived model from the 3D design model. The design model has all of the components in it, including both heat sinks, the fan, and the final circuit layout. As we experiment, we likely would like to try alternate configurations, edit geometry, move things around, etc. without altering the official design geometry. To accomplish this, we use a derived model of the design model. This means if you make changes in the simulation environment, it does not affect the design model, but if you make changes in the design model, you are notified that the simulation model is out of date and the changes can be accepted in the simulation model. This associative downhill flow of data allows for early experimentation and then easy updates to our simulations as the official design changes. For clarity, let's rename the "Simulation Model" to Baseline. There are specific tools to remove components, remove detail, etc. within the simulation environment. To access them, select "Simplify" in the toolbar.



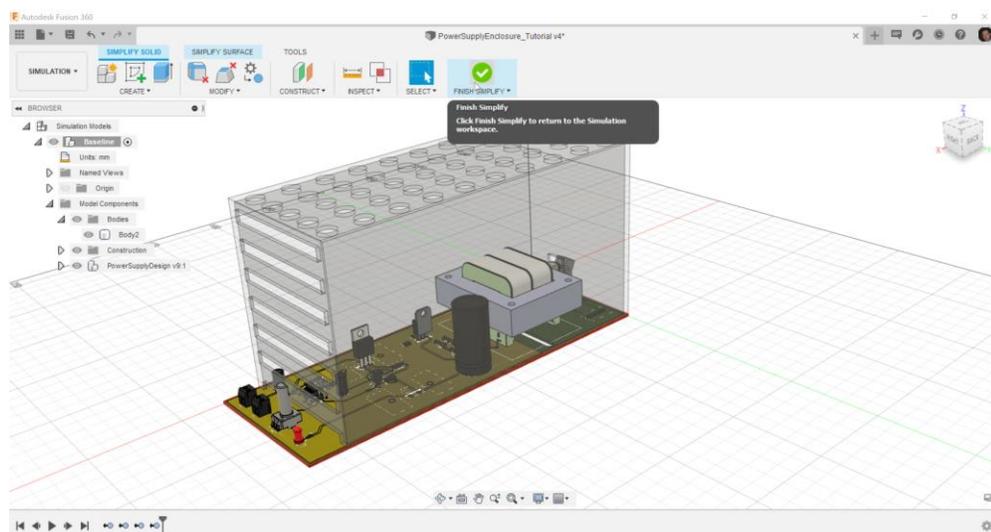
Simplify Command

We want to see what happens if we use the simplest design first. We need to remove the heat sinks, the fan and the second regulator. Select each component, and then RMB and select “Remove”.



Removing the components

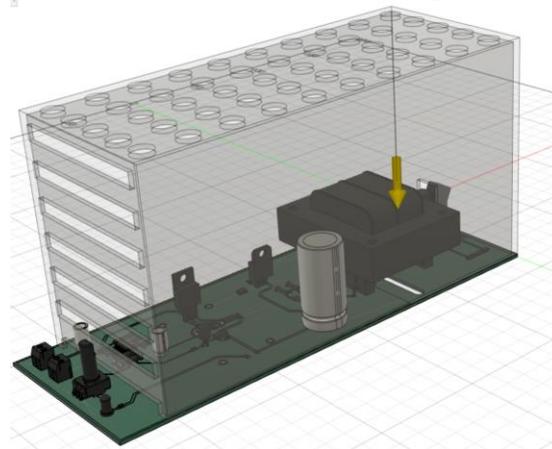
Time History is on for this workspace, so we can easily edit or undo any of our actions. Time History looks like a little play list on the bottom toolbar. This will be useful as run additional experiments where we want these components. Below is what our model should now look like. To go back to defining the materials, power dissipation, etc., we will select “Finish Simplify”.



Baseline Model

Materials

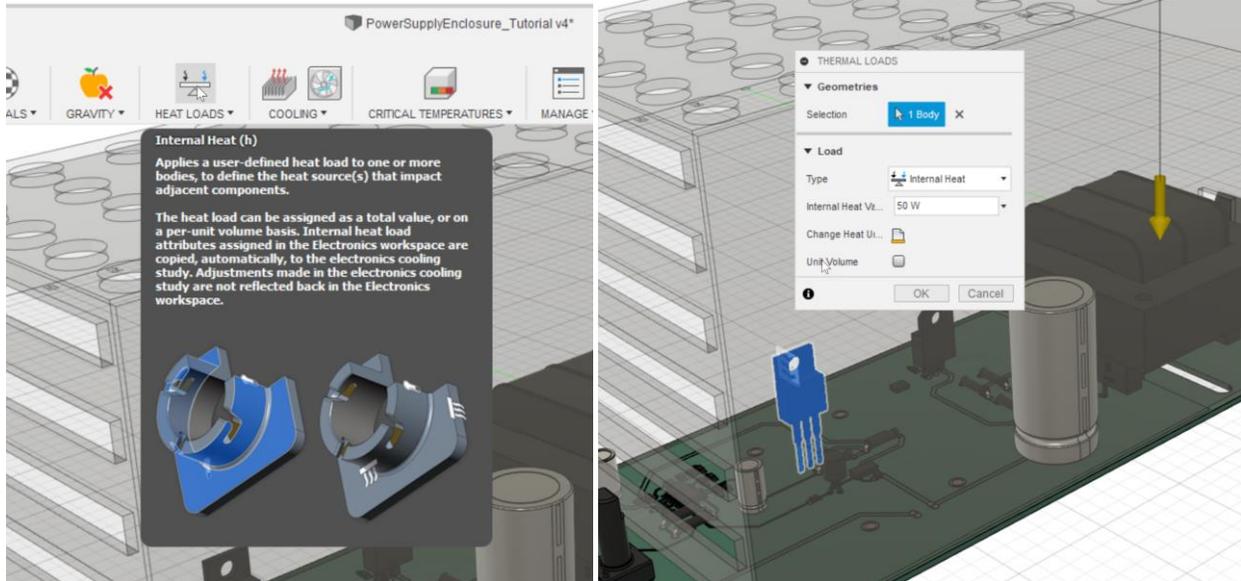
If you expand the drop down for Materials from the toolbar, you notice there is the ability to toggle “Display Study Material Colors”. Default should be checked. Notice that many of the materials are the same color. By default, the parts receive the material that is assigned from the design model. This can be overridden in the simulation study to enable investigation of the effect of material choices. By default, the PCB that comes from PCB will be FR4 and the copper traces and vias will be copper as the 3D PCB is created. All of the components mounted to the PCB will be “Discrete Material” “Discrete Material” is a decent average thermal conductivity for the electronics components on a PCB. If the PCB has heat sinks and the material was not altered in the design model, then we would need to override the material to something like Aluminum. We will do that in one of our alternate studies. For now, the materials are all what they need to be thanks to the automation built into electronics cooling.



Model Display as Material Color

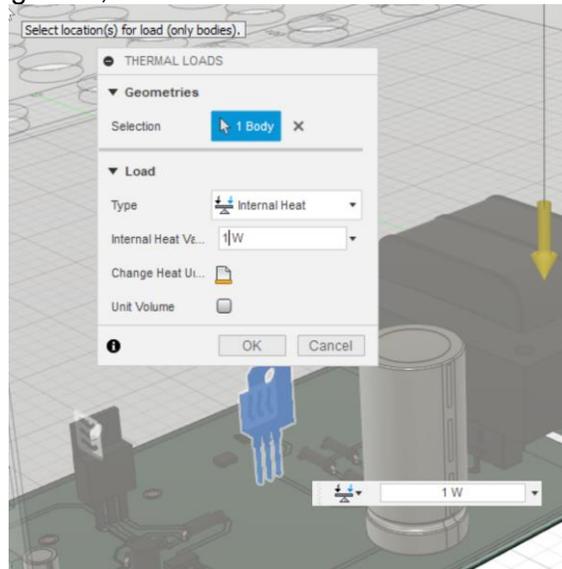
Power Dissipation (Heat Generation)

Expand “Load Case1” and then “Loads” in the Browser Menu. Notice the only assigned load is gravity. If we need to change the gravity direction (or even magnitude), then we would do it there. No changes needed for this model as gravity is pointing in the right direction. We do need to assign the power dissipation of the regulators. In the toolbar, select “Internal Heat”, then select the regulator in the image below and assign 50 watts.



Main Regulator Heat Load

Repeat for the other regulator, but this time make it 1 Watt.



Secondary Regulator Heat Load

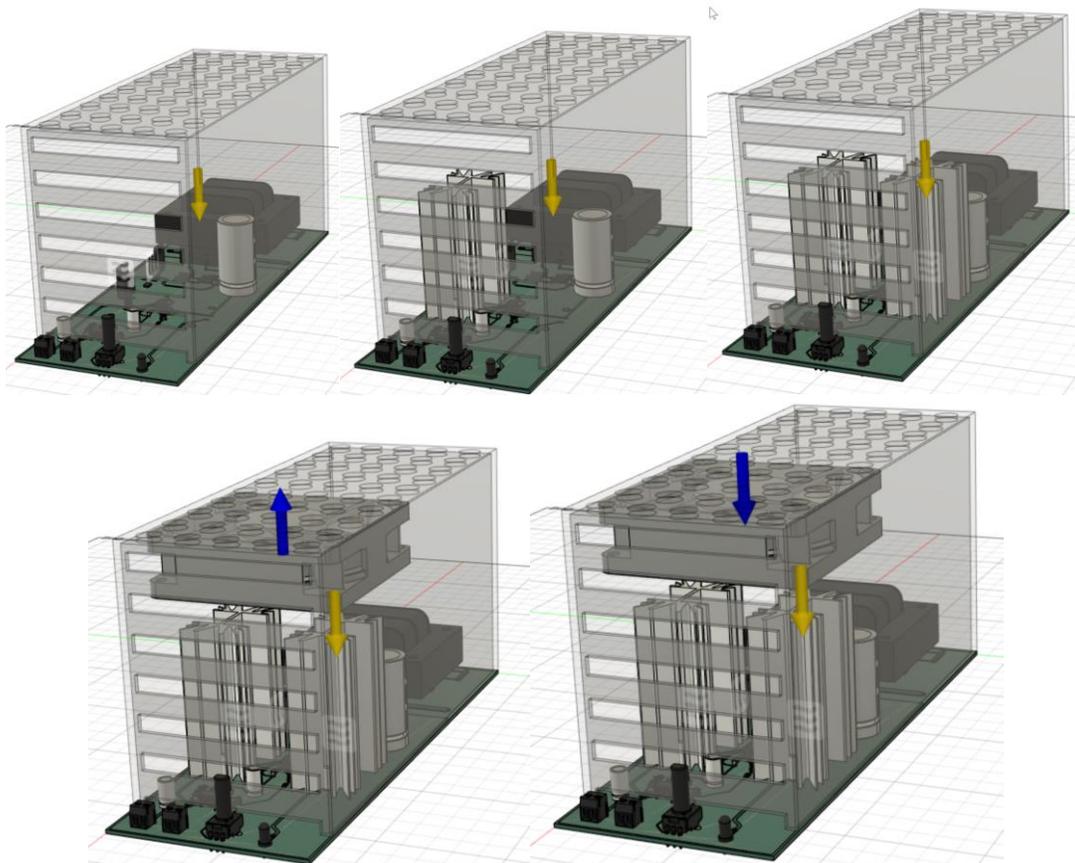
Critical Temperature Assignment (Optional)

Since there is no fan for this scenario, we have defined all that is needed for this experiment. An optional value that is helpful when it comes time to look at results is to assign critical temperature to our dissipating components. We can assign critical temperature to any component, but it makes most sense for the ones generating heat. Select the two regulators, then click "Critical Temperatures" in the toolbar. A dialog pops up that has the two components highlighted and lists all the bodies in the model. Type

“130” into the Critical Temperature field. This is an optional step, so we won’t go into much detail here but feel free to investigate this dialog. At this point, we can click solve if this is the only option we want to investigate. It is easy to set up additional experiments, however, so we will do so now.

Additional Experiments

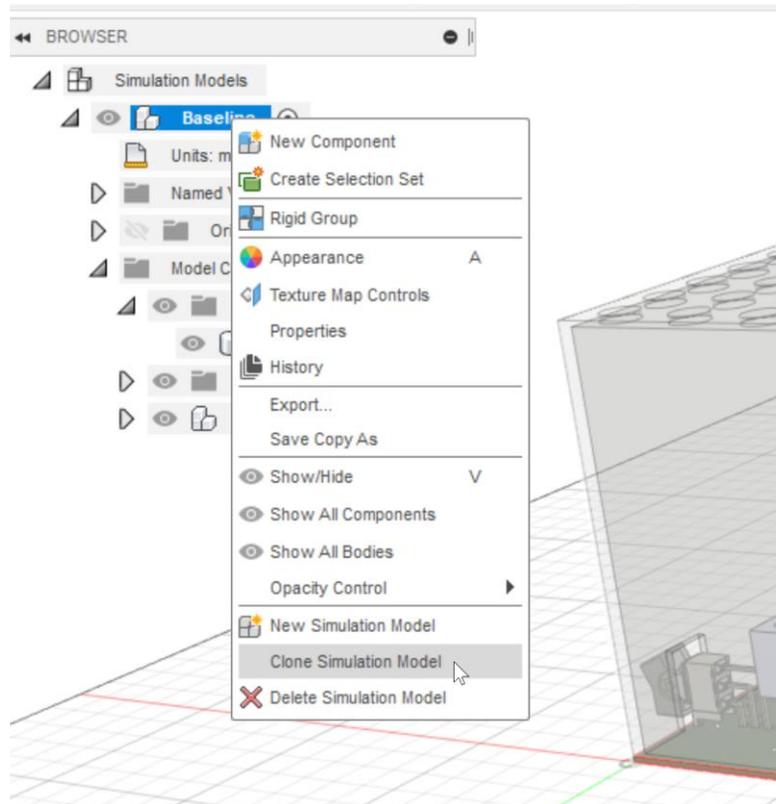
We want to also look at the effects of adding a heatsink to the main regulator, then also a resistor circuit with its own heatsink, then finally add a fan to the enclosure and investigate if fan flow direction has an impact on temperatures.



The Different Design Options

Clone the Design

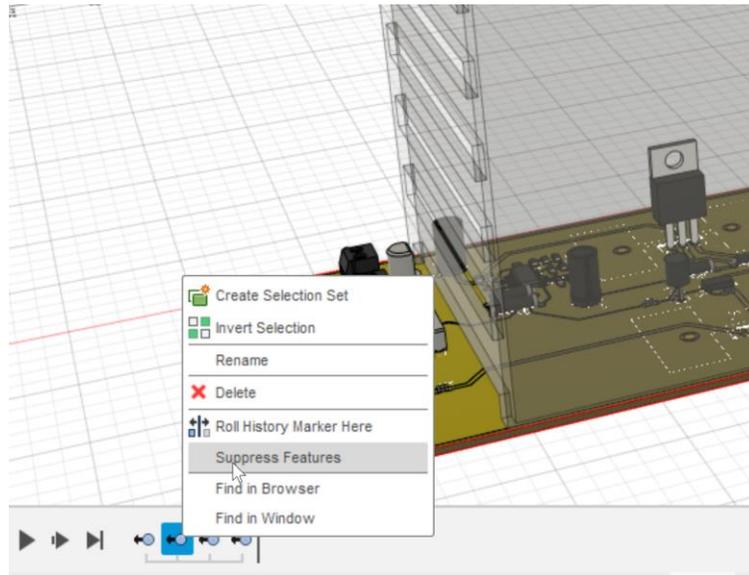
Select “Simplify” in the toolbar. Then RMB on “Baseline” in the browser tree. In the context menu that pops up, select “clone simulation model”. Select “OK” on the dialog that pops up. The new simulation model is an exact copy of “Baseline”. It will be named “Simulation Model2” or “Simulation Model3”, etc. Let’s rename it to “One HeatSink”



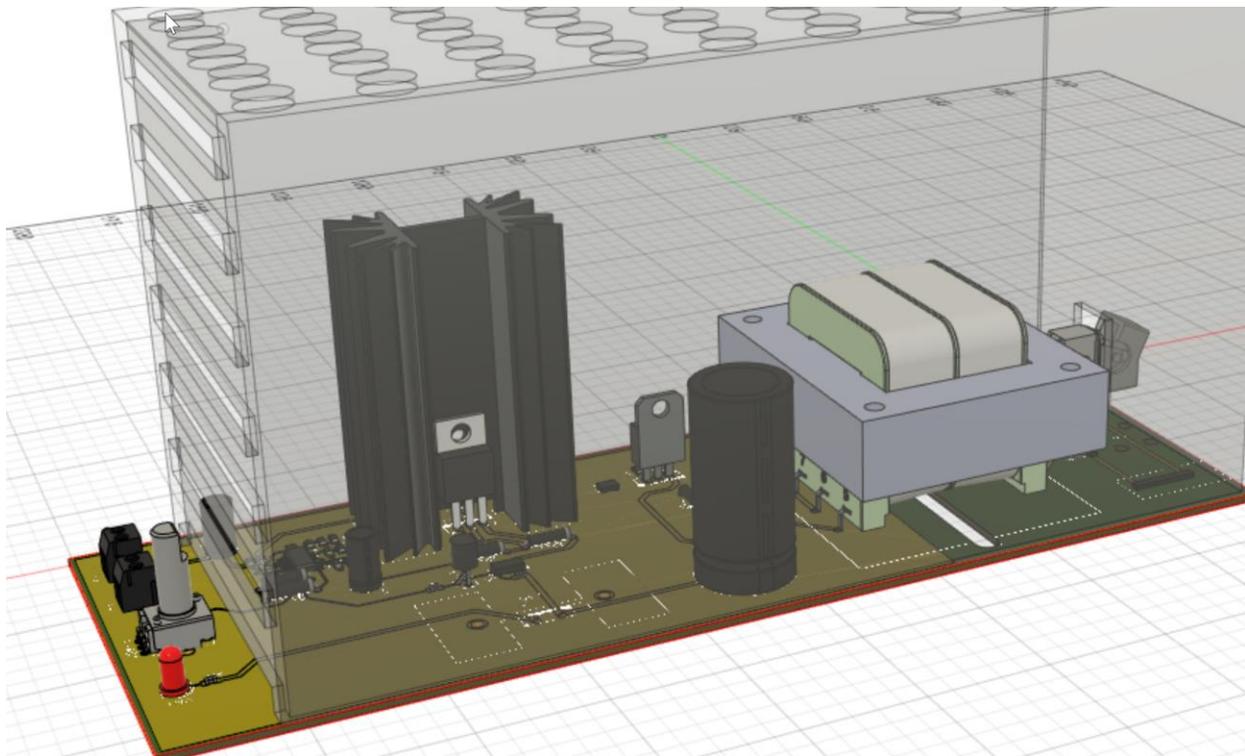
Clone the Simulation Model

Modify the Geometry

On the timeline at the bottom of the screen, RMB on one of the “RemoveInstance” features and select “suppress features”. If the heat sink that is attached to the main regulator appears, awesome...you’ve guess the correct feature we needed to suppress. If something else shows up, then RMB on it again and unsuppress. Then suppress a different one. Repeat until you’ve found the right feature to suppress and your model now has added only the heat sink that is attached to the main regulator.



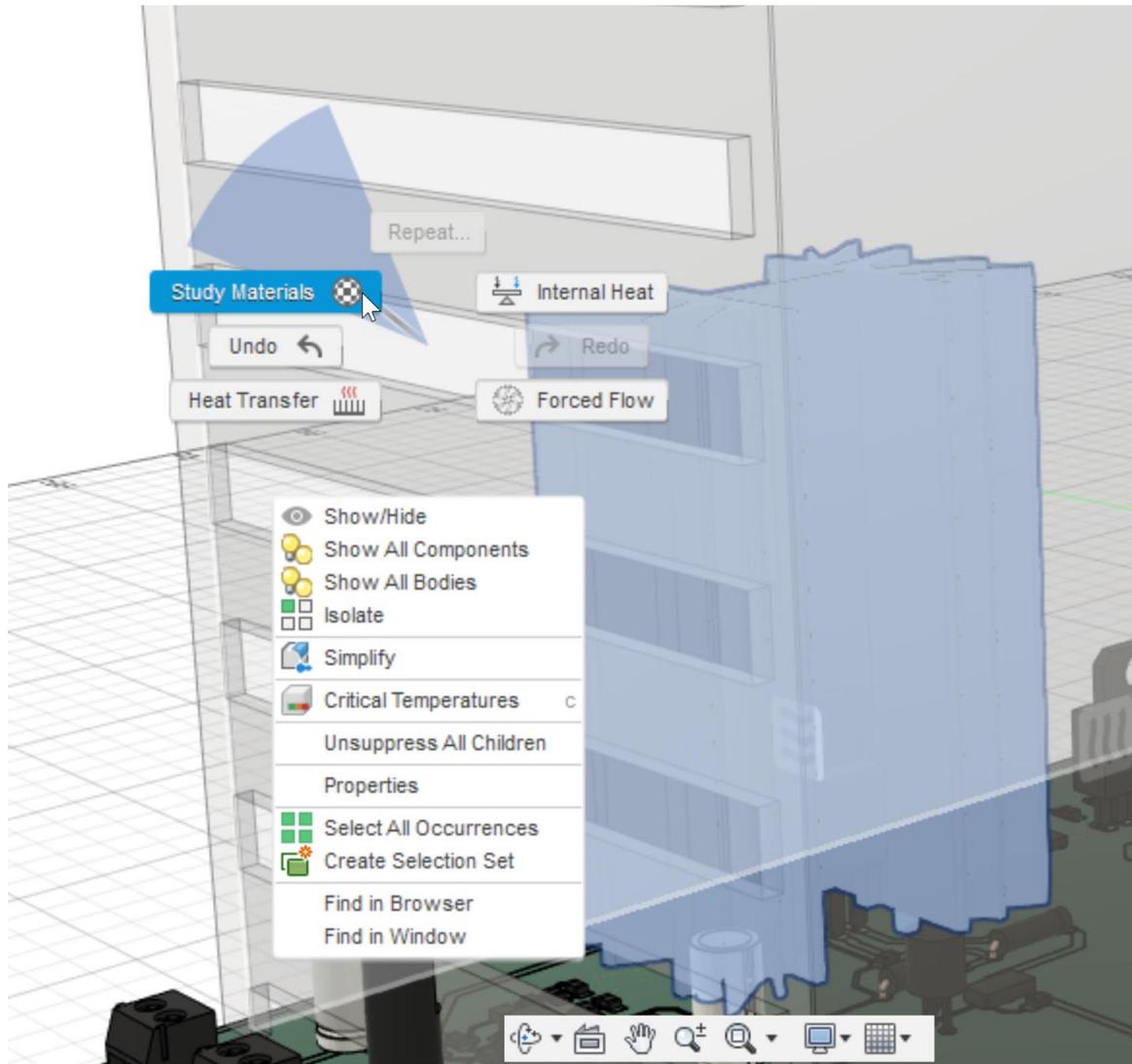
Suppress the feature



The correct heatsink we want

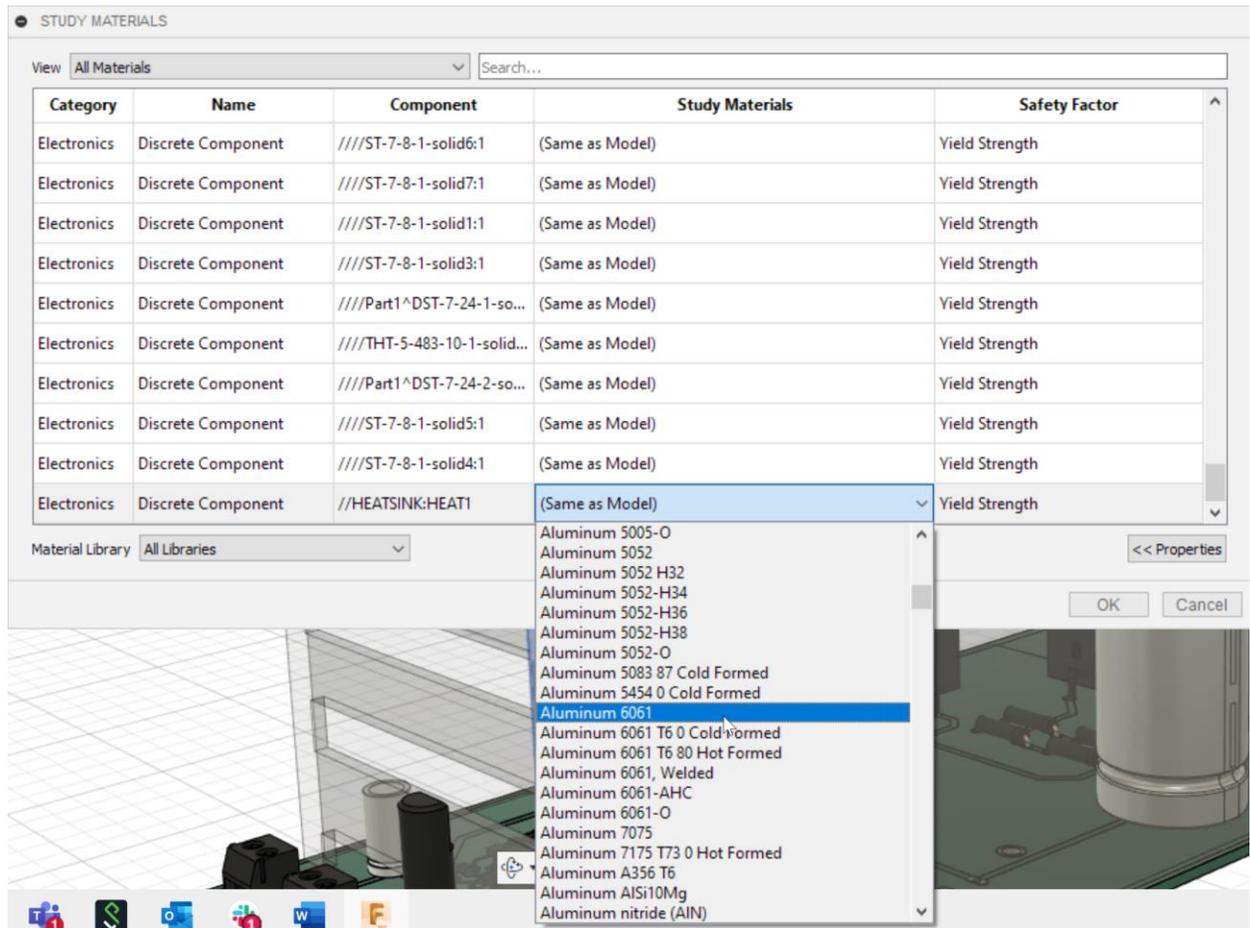
Materials

We need to adjust the material for the heatsink. Select “Finish Simplify”. Select the Heatsink body (double LMB on it is the easiest way). Then RMB and select “Study Materials” from the marking menu.



Edit the Heatsink Material

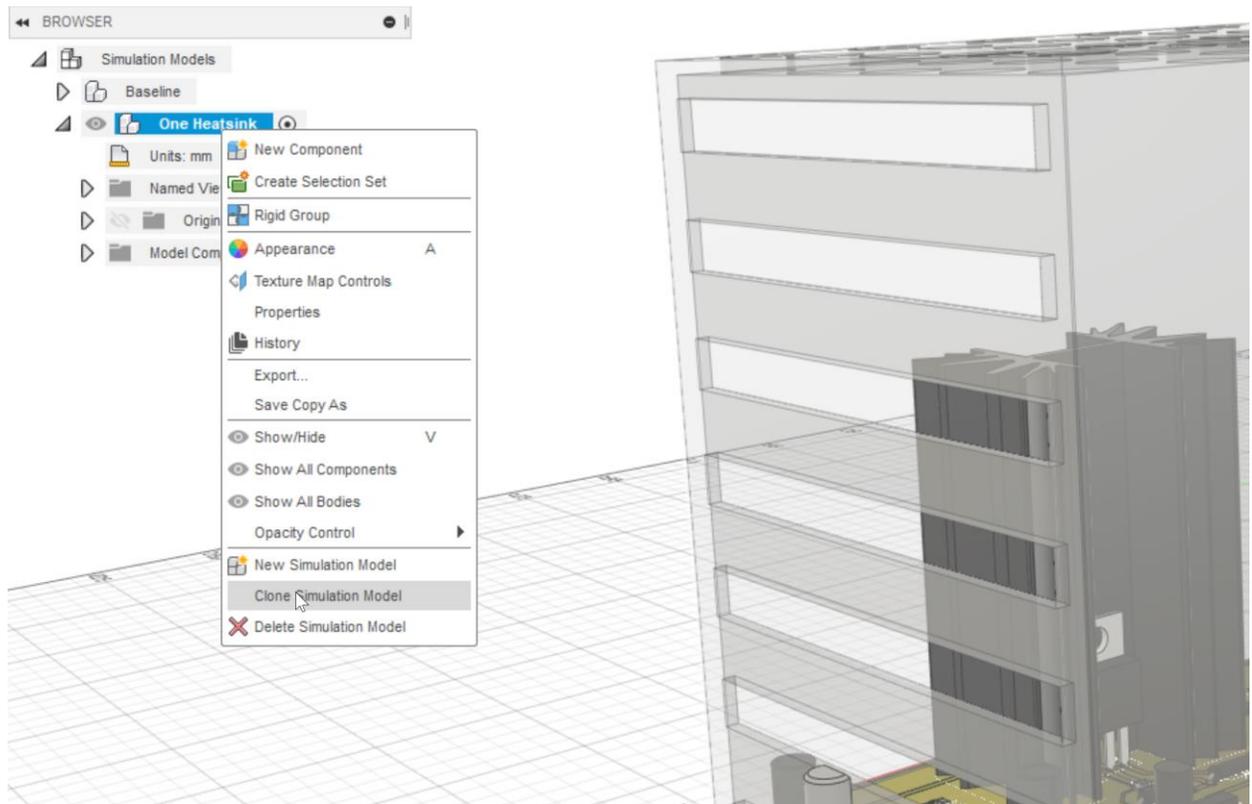
Change “(Same as Model)” for the prehighlighted component (the heatsink) to “Aluminum 6061” and select OK.



Changing the heatsink material

Clone the Design (again)

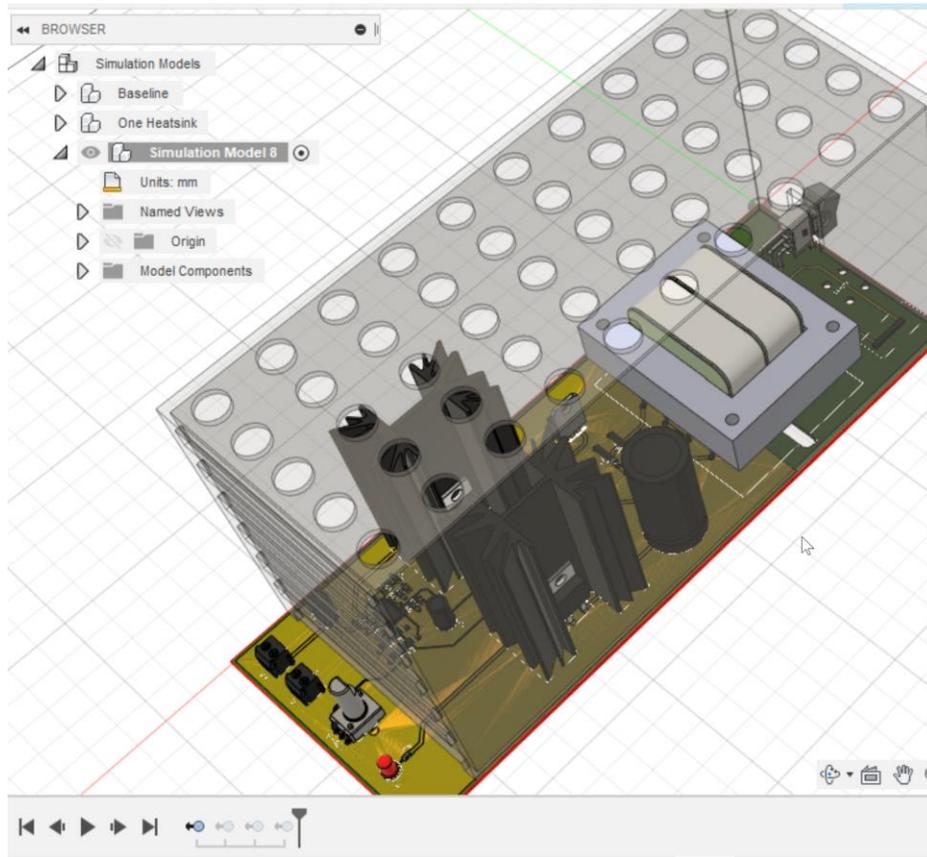
The second design is all ready to solve, but we will set up our third, fourth, and fifth experiments in the following steps. Select “Simplify” in the toolbar. Then RMB on “One Heatsink” in the browser tree. In the context menu that pops up, select “clone simulation model”. Select “OK” on the dialog that pops up. The new simulation model is an exact copy of “One Heatsink”. It will be named “Simulation Model3” or “Simulation Model4”, etc. Let’s rename it to “Two HeatSinks”



Clone “One Heatsink”

Modify the Geometry

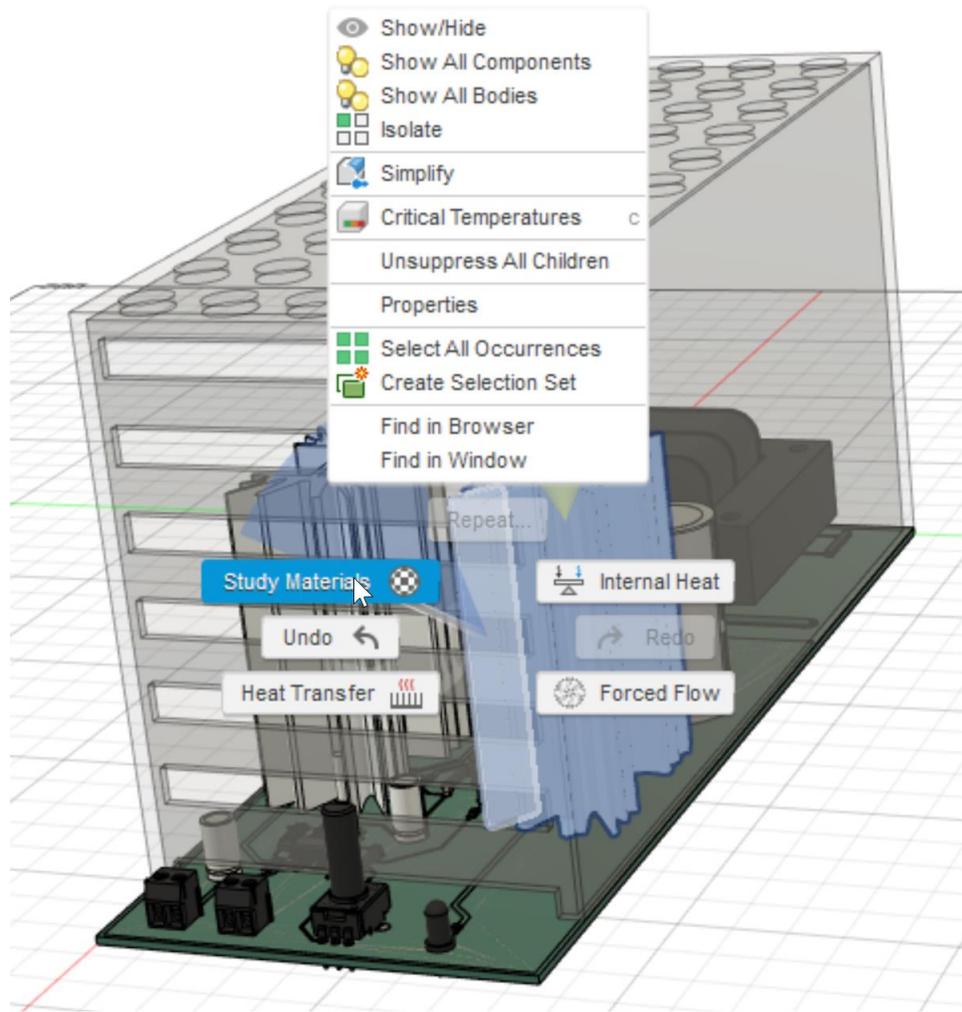
On the timeline at the bottom of the screen, RMB each of the remaining “RemoveInstance” features and “suppress features”. Do this for all except the one that removed the fan. We want to keep the fan “removed” for now. The model should look like this when you are done.



Two Heatsink Model

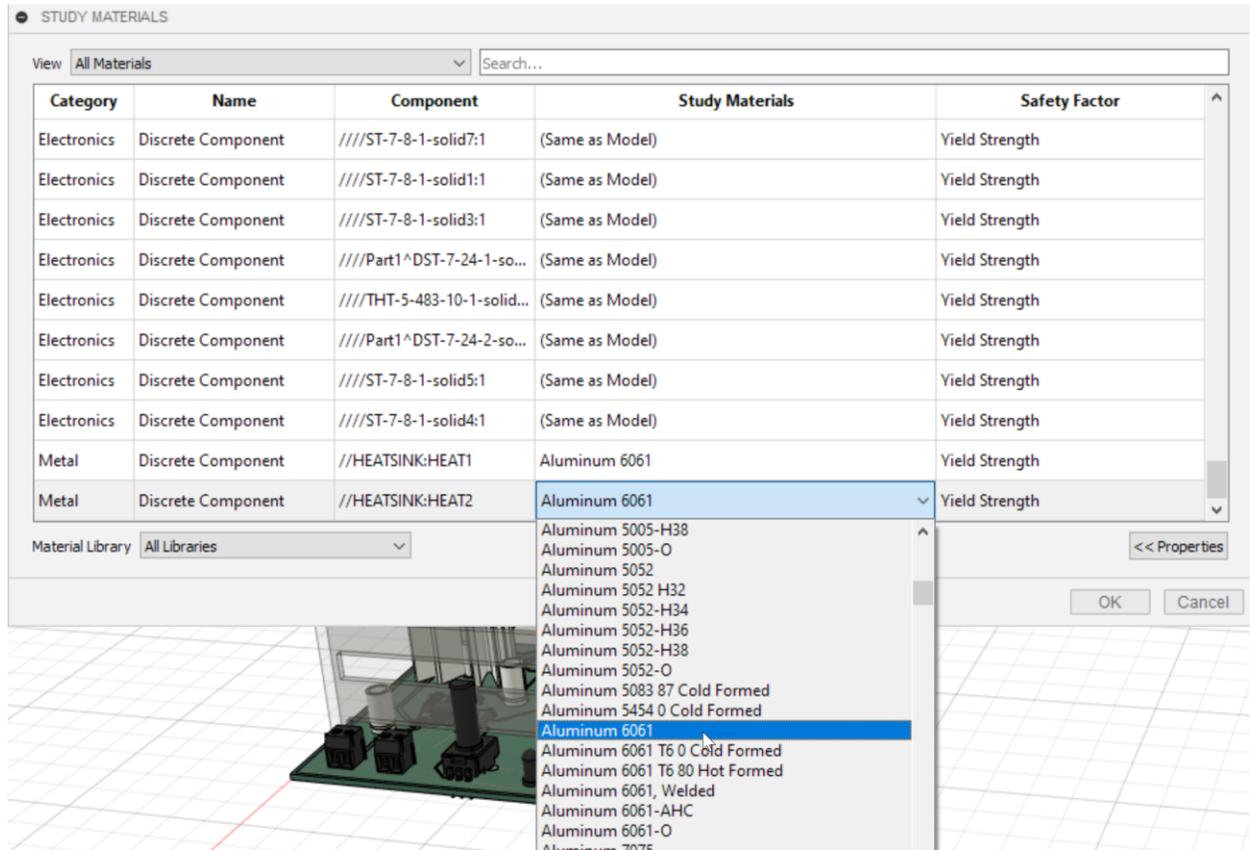
Materials

We need to adjust the material for the second heatsink. Select “Finish Simplify”. Select the Heatsink body (double LMB on it is the easiest way). Then RMB and select “Study Materials” from the marking menu.



Edit the Heatsink Material

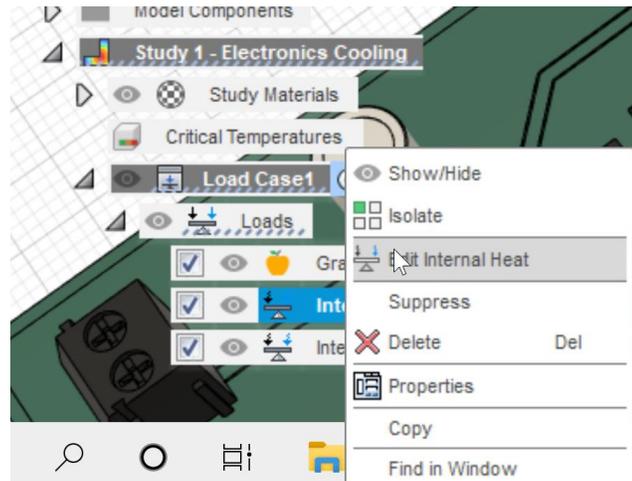
Change “(Same as Model)” for the prehighlighted component (the heatsink) to “Aluminum 6061” and select OK.



Changing the heatsink material

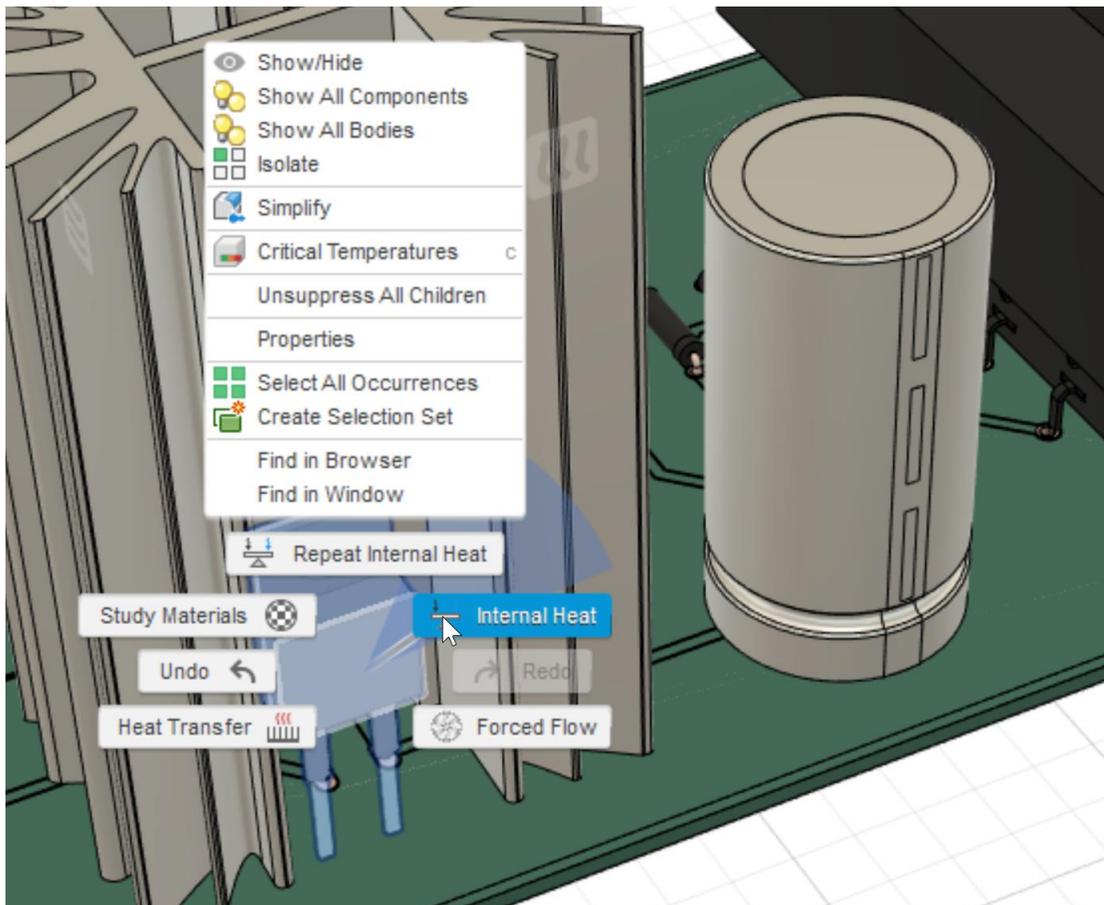
Power Dissipation (Heat Generation)

The design of this board with the resistor circuit added is to have 50 watts dissipation spread between the regulator and resistor. For this model, the regulator will dissipate 27 watts and the resistor will dissipate 23 watts. Find "Internal Heat1" in the browser, then RMB to select edit and change the assignment from 50 watts to 27 watts.



Changing the regulator heat dissipation

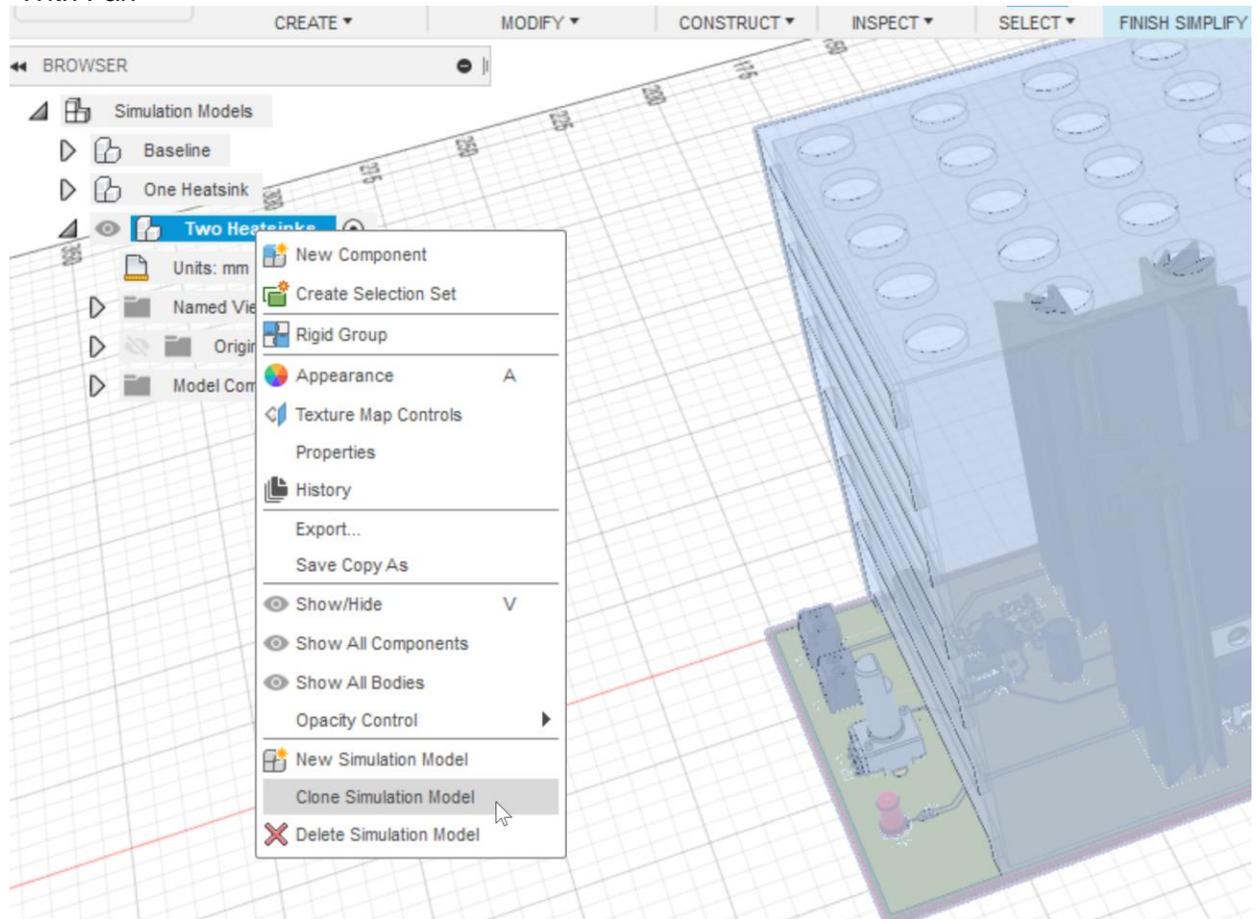
Now, select the body for the resistor and RMB to bring up the marking menu and select “internal heat”. Assign 23 watts to the resistor.



Applying the heat dissipation to the resistor

Clone the Design (again)

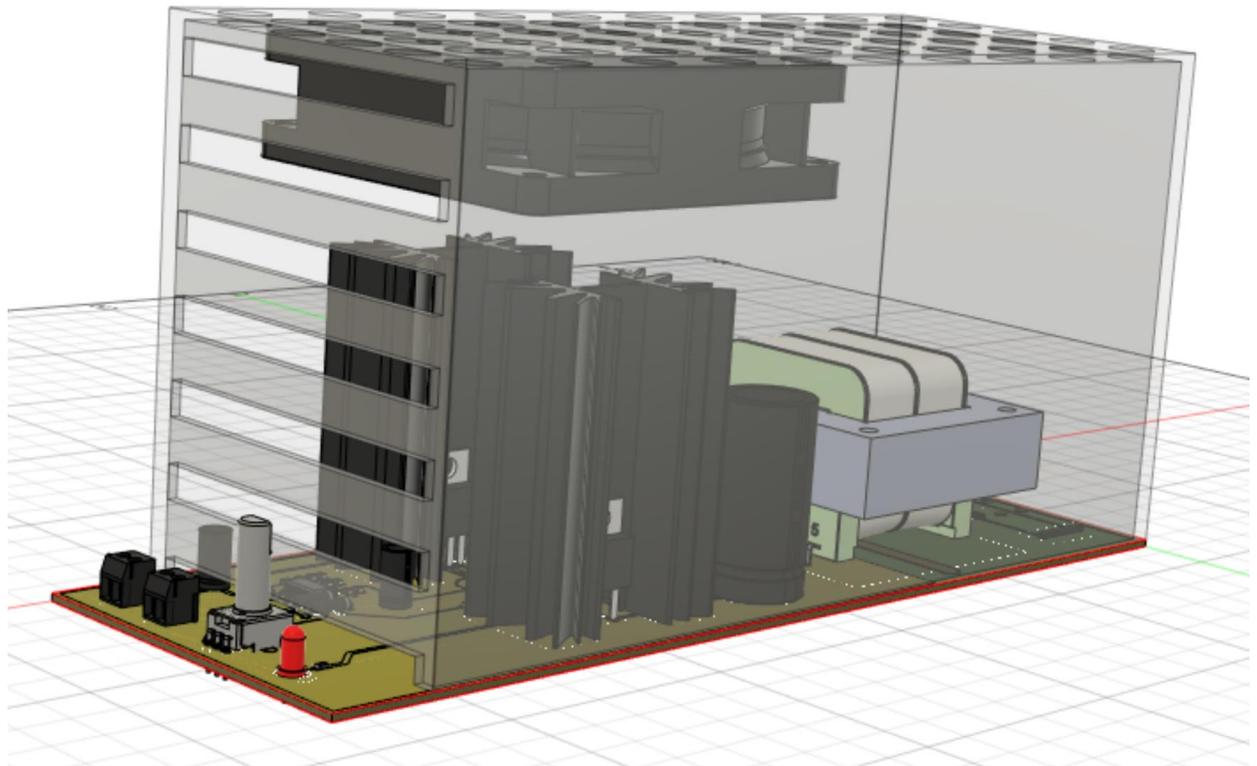
The third design is all ready to solve, but we will set up our, fourth, and fifth experiments now. Select “Simplify” in the toolbar. Then RMB on “Two Heatsinks” in the browser tree. In the context menu that pops up, select “clone simulation model”. Select “OK” on the dialog that pops up. The new simulation model is an exact copy of “One Heatsink”. It will be named “Simulation Model4” or “Simulation Model5”, etc. Let’s rename it to “With Fan”



Clone “One Heatsink”

Modify the Geometry

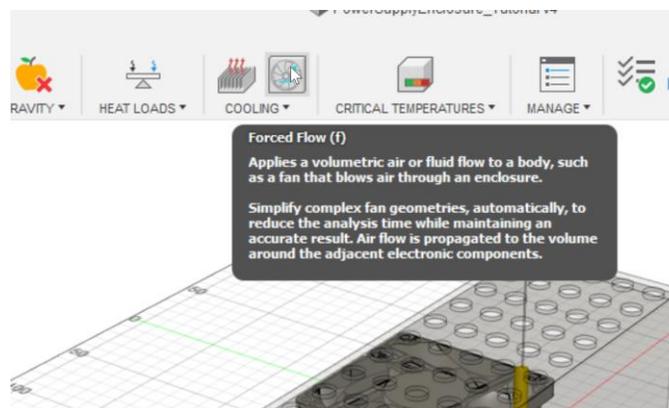
On the timeline at the bottom of the screen, RMB each of the last remaining “RemoveInstance” features and “suppress feature”. The model should look like this when you are done.



With Fan Model

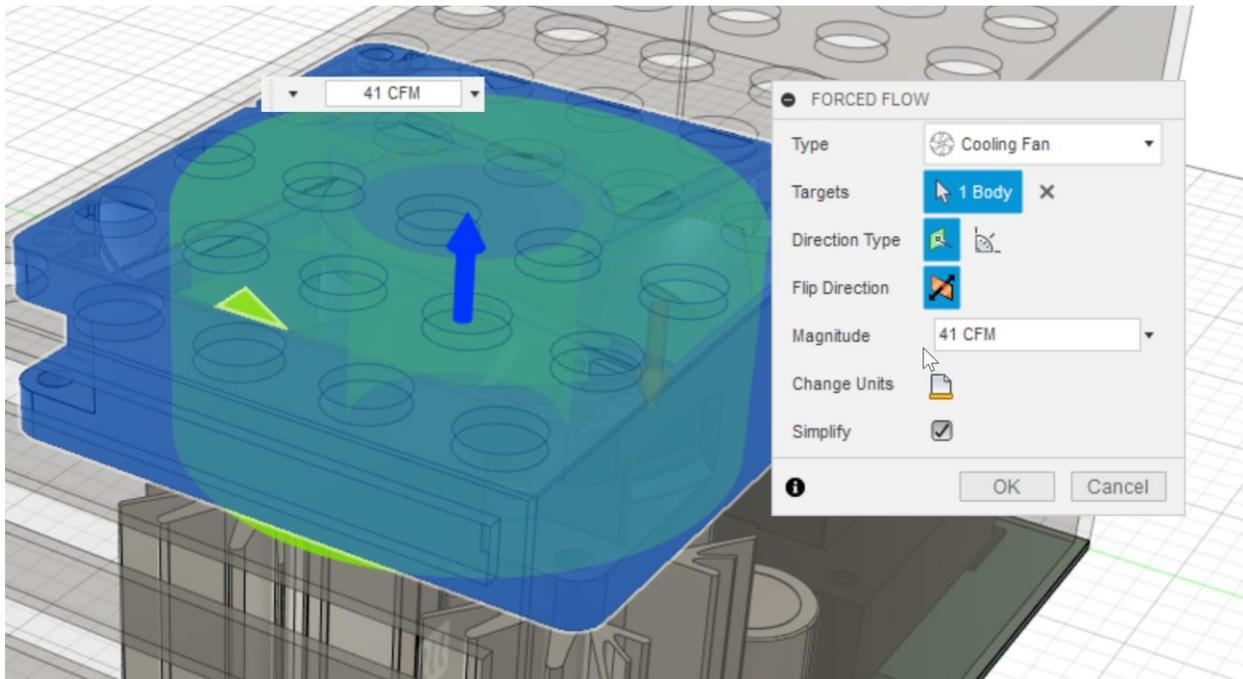
Fan (Forced Flow)

The experiment we will be running has a fan. We will run one experiment with the fan pulling air out of the enclosure, and another with the fan pushing air in. Select “Finish Simplify”. Select “Forced Flow” in the toolbar, then select the fan geometry.



Assigning the Fan Flow Rate

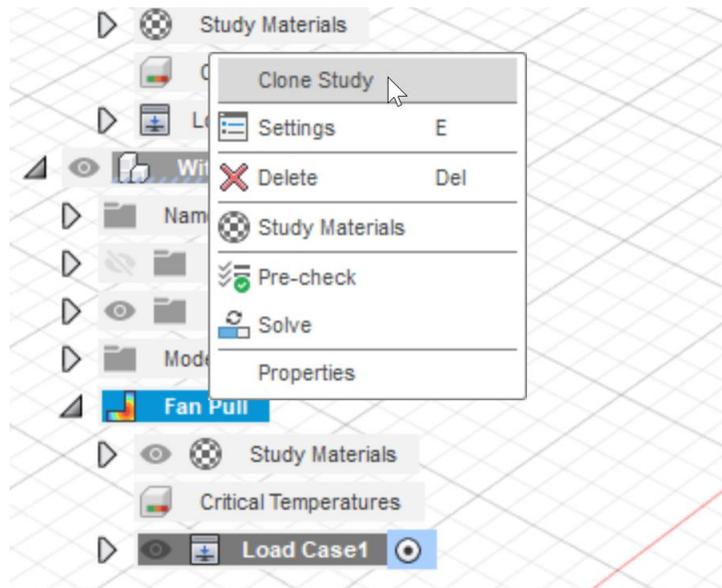
In the dialog that appears, flip the direction so the fan is flowing out (arrow indicator showing up out of the enclosure) and assign 41 cfm as the magnitude.



Assigning the Fan Flow Rate

Clone Scenario

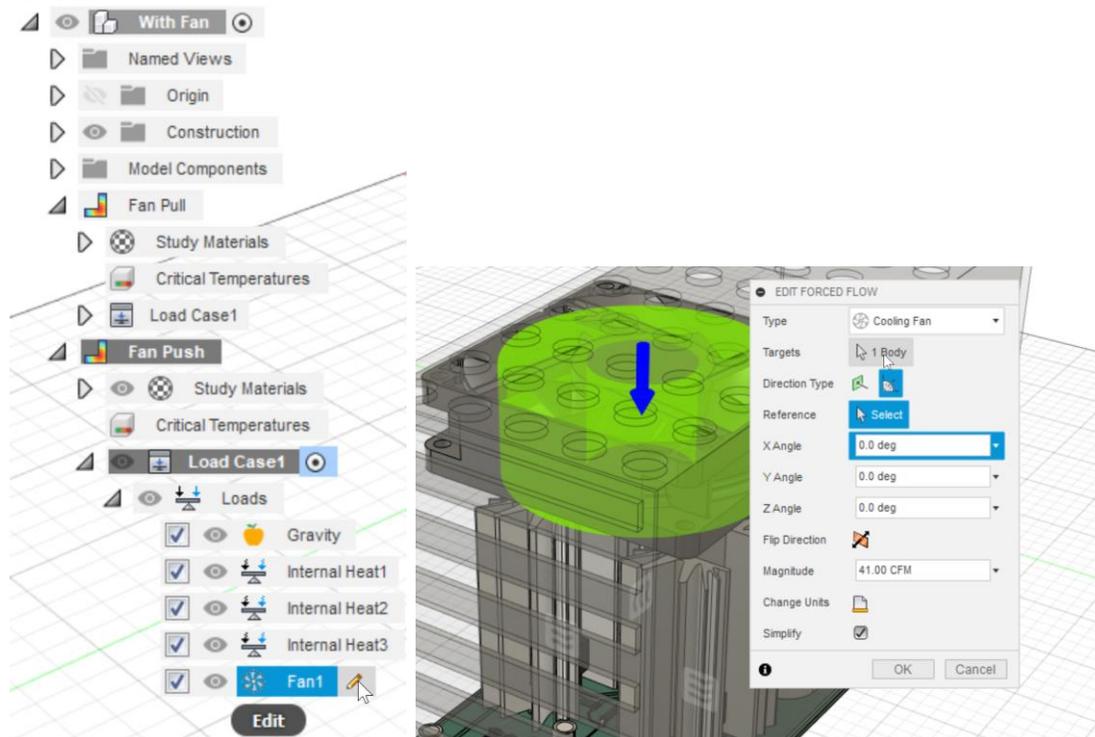
This experiment is ready to go. For our last experiment we will make a clone of the study and flip the fan direction, then we will have all of the models solve on the cloud and look at results when they are done. First let's rename the study. It is called "Study1-Electronics Cooling". Double click the name to rename it to "Fan Pull". Then RMB on it and select "Clone Study". Rename the new study to "Fan Push"



Assigning the Fan Flow Rate

Fan (Forced Flow)

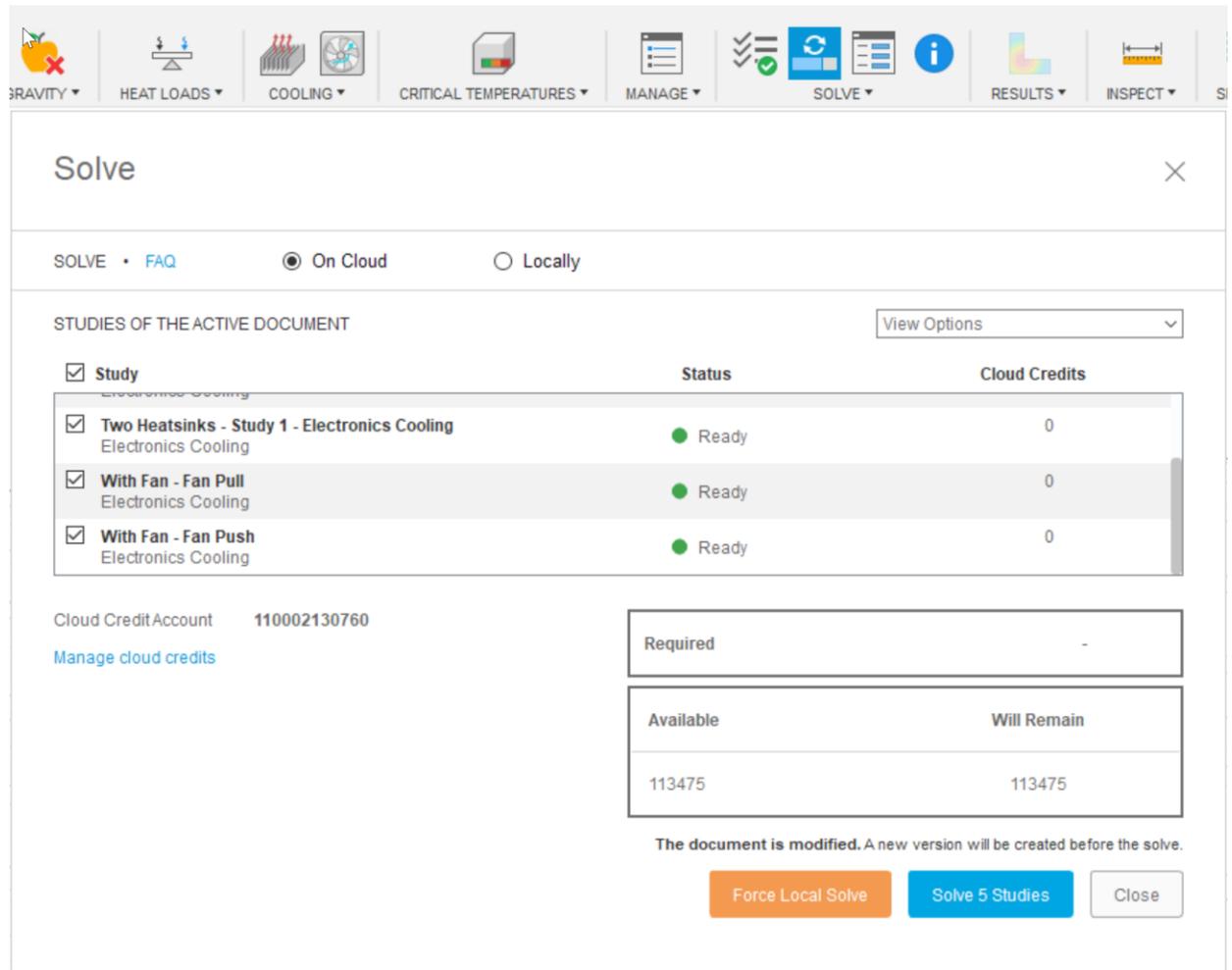
Expand the browser to show the Load Case1 and click to edit Fan1. In the dialog, flip the direction so the flow direction indicator is pointing down into the enclosure and select "OK".



Changing the Fan Direction

Solve all Scenarios

All the experiments are ready to go. Let's click "Solve" in the toolbar. Select all the studies. There should be 5 of them if you followed the entire handout. Select "Solve 5 Studies".



The screenshot shows the 'Solve' dialog box with the following details:

- Toolbar:** Includes icons for GRAVITY, HEAT LOADS, COOLING, CRITICAL TEMPERATURES, MANAGE, SOLVE, RESULTS, and INSPECT.
- Location:** SOLVE • FAQ, On Cloud, Locally
- STUDIES OF THE ACTIVE DOCUMENT:**

Study	Status	Cloud Credits
<input checked="" type="checkbox"/> Two Heatsinks - Study 1 - Electronics Cooling Electronics Cooling	● Ready	0
<input checked="" type="checkbox"/> With Fan - Fan Pull Electronics Cooling	● Ready	0
<input checked="" type="checkbox"/> With Fan - Fan Push Electronics Cooling	● Ready	0
- Cloud Credit Account:** 110002130760
[Manage cloud credits](#)
- Required:** -
- Available:** 113475
- Will Remain:** 113475
- Message:** The document is modified. A new version will be created before the solve.
- Buttons:** Force Local Solve, **Solve 5 Studies**, Close

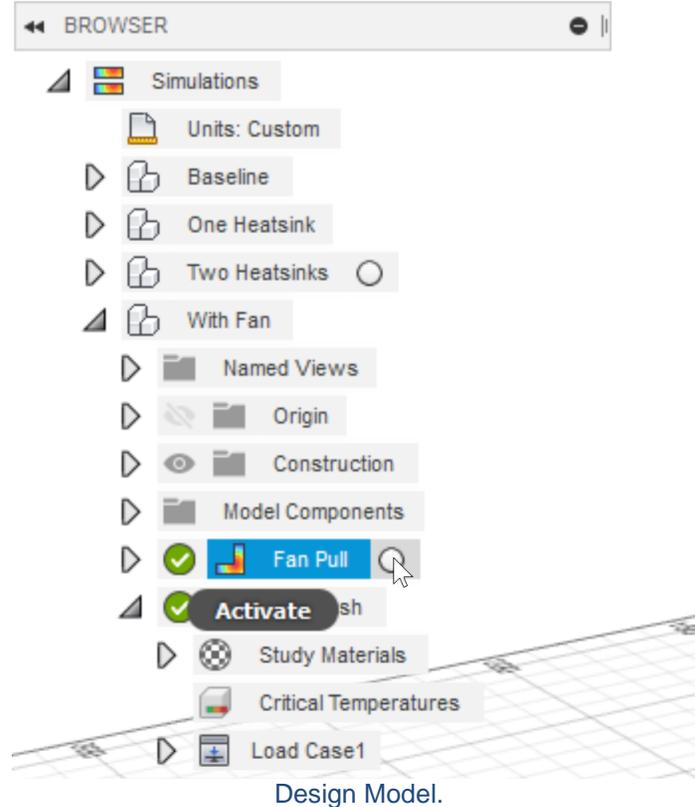
Solving the Studies

Look at the flow and thermal results

In the lecture video we showed results from 5 different configurations for the electronics assembly. This portion of the handout will look at the flow and thermal results for the "Fan Pull" study. After walking through the results for one study, use the same techniques to look at results for the other studies.

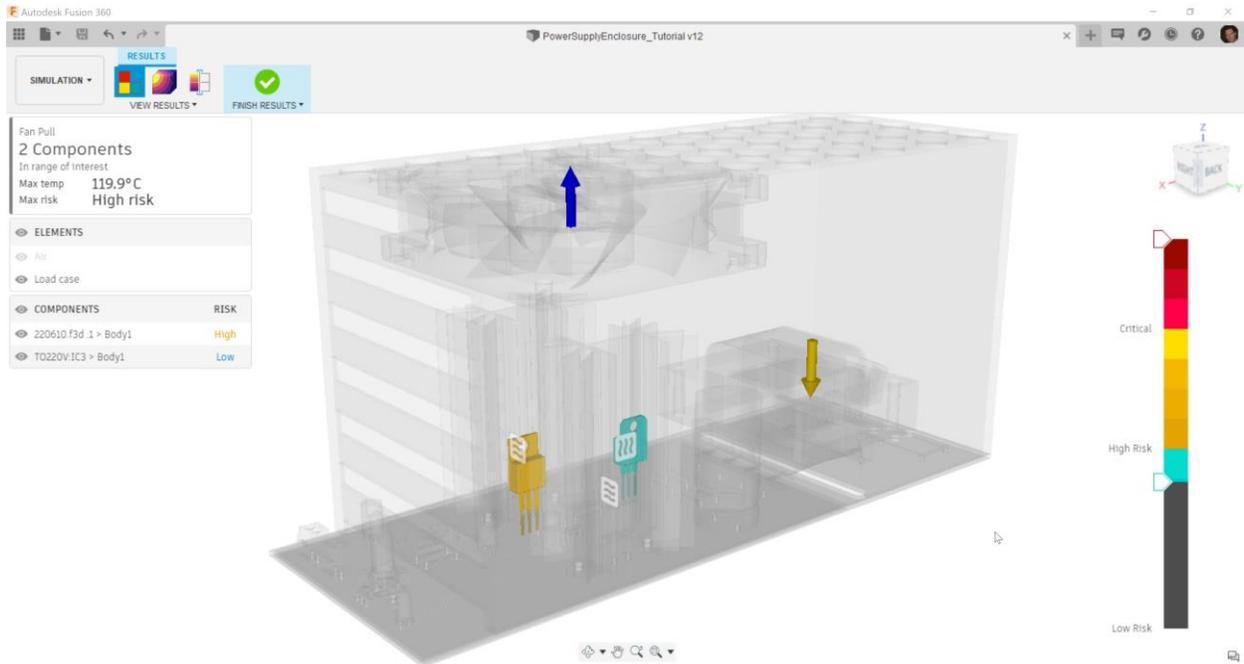
Activate the “Fan Pull” study

In the Browser Tree, expand “With Fan” and activate “Fan Pull”. Next, on the toolbar, select “Results”. The results were solved and stored on the cloud. The model will take initially approximately 30 seconds to load the results into the cache and display fully onto the canvas



Initial Results View: Risk Scale

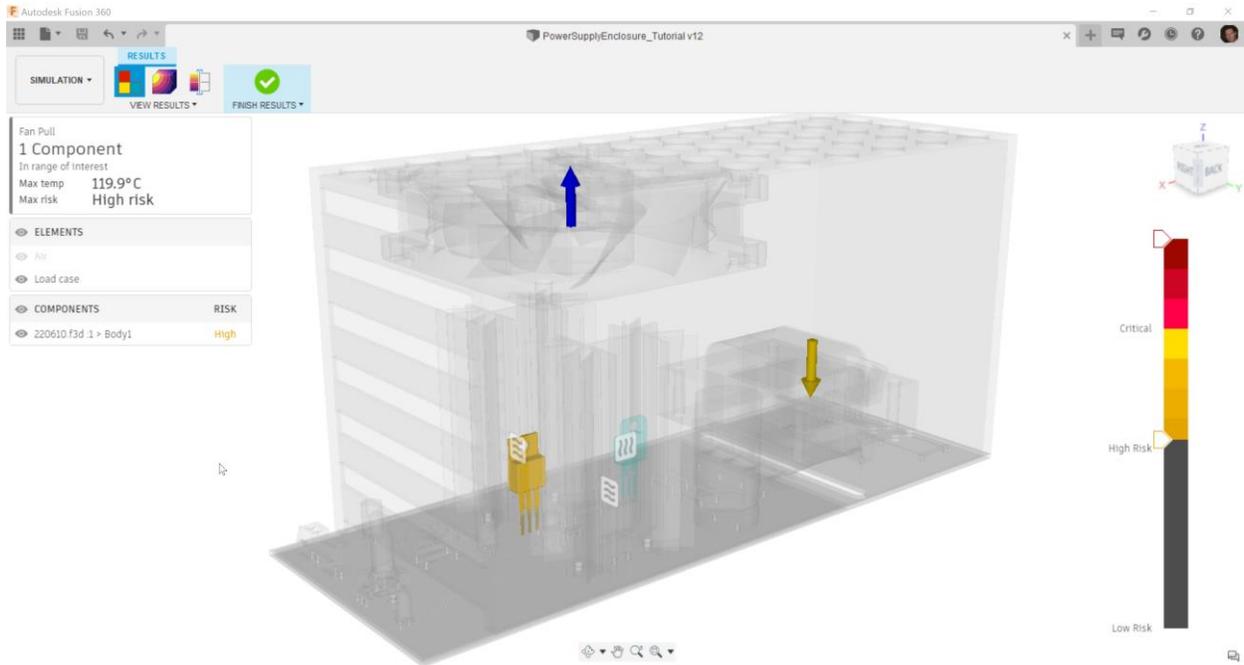
If you followed the steps to assign the regulators a critical temperature, the initial results view will be a view that shows whether the components specified with critical temperature are at risk of being too hot. The model is shown in transparent view with just the components assigned critical temperature fully opaque. Below is what that looks like.



Risk Scale (Critical Temperatures)

Adjust Range of Interest on Legend

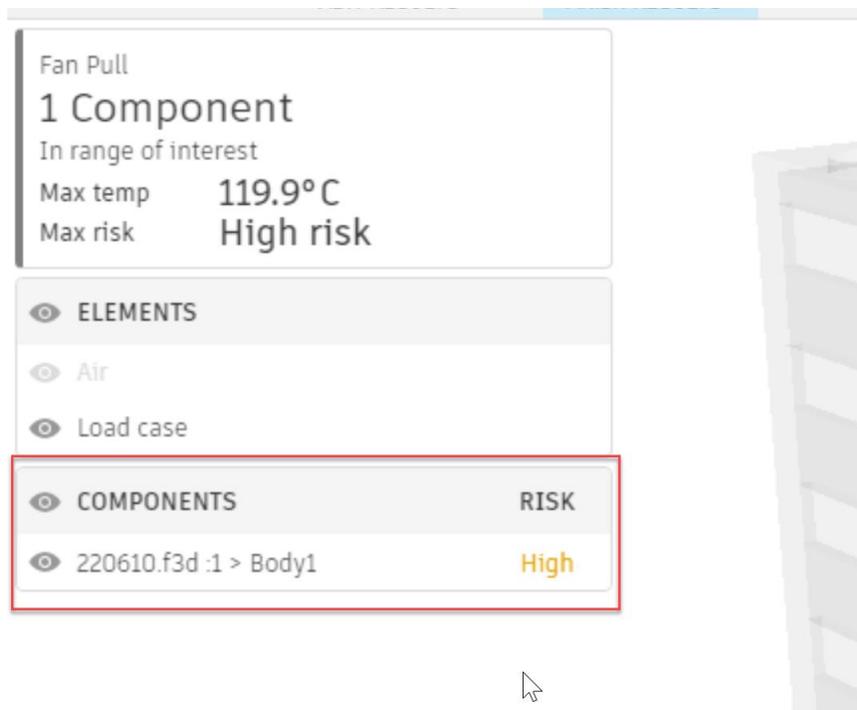
All of the color legends have the ability to be directly adjusted. In the case of Risk Scale, we can select the maximum range of interest indicator or the minimum range of interest indicator and adjust them up or down. This controls the visibility of objects painted by the results quantity. Grab the minimum range of interest indicator and slide it up past the “high risk” bar and you will notice the 1 watt regulator goes from opaque to transparent. Notice also the components list on the left now only shows the 27 watt regulator, since the 1 watt regulator is not within the range of interest. This is a control capability and also interaction between the display and components list we will commonly see for the results visuals in electronics cooling.



Effect of moving range of interest

Explore features of the components list

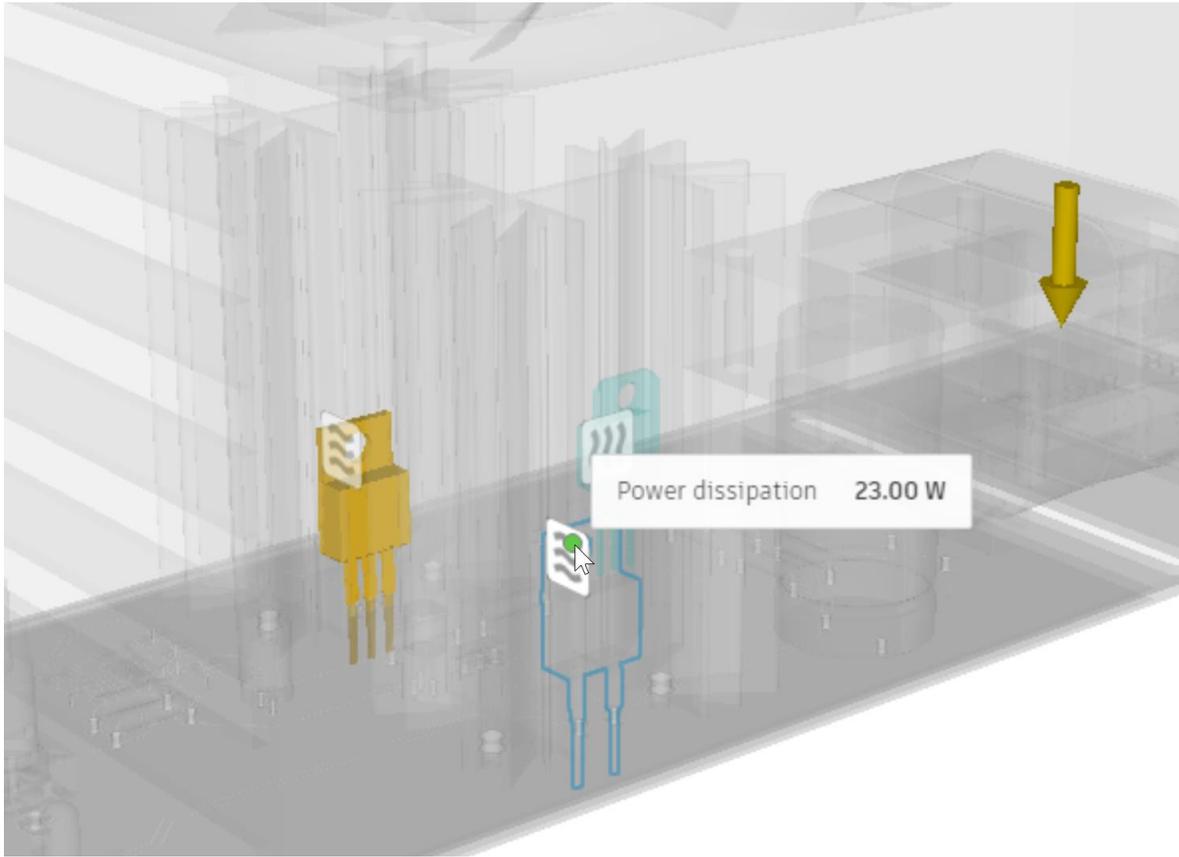
If you hover over the 220610.f3d:1>Body1 text in the components list, notice the components list will display the complete path of that component. Also notice that the component is highlighted in the canvas. If you select the component in the component list, the component will become highlighted until you unselect it either in the component list or in the canvas via LMB. If you select the eyeball icon next to “Components” all of the components in the model will be hidden. If you select it again, all the components will then become visible. The same action can be taken next to an individual component or components in the list.



Components List

Explore hover over items in canvas

If you hover over glyphs in the canvas, notice the load magnitude and the body it is assigned to is shown (if applicable). You control glyph visibility by toggling the eyeball next to “Load Case” in the “Elements” List.



Hovering over a glyph

Fan Pull

1 Component

In range of interest

Max temp **119.9°C**

Max risk **High risk**

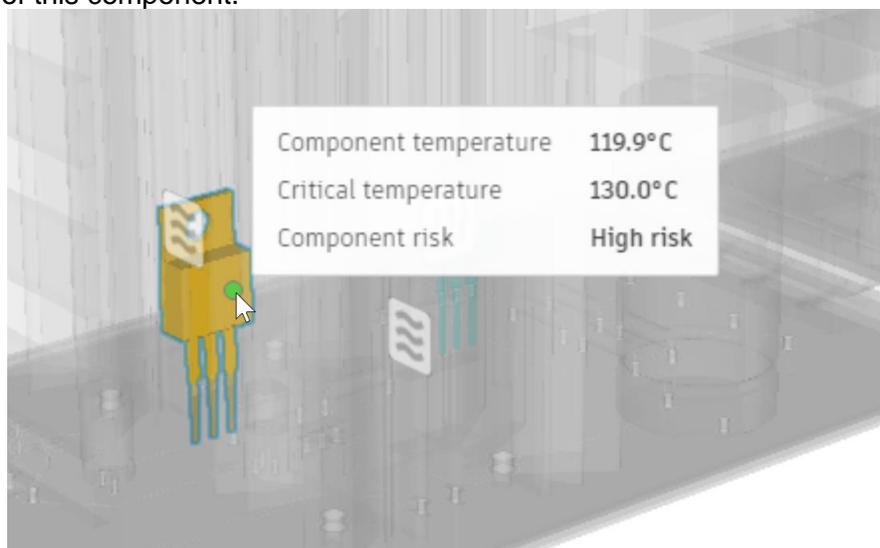
ELEMENTS

- Air
- Load case

COMPONENTS	RISK
<input checked="" type="radio"/> 220610.f3d :1 > Body1	High

Elements List

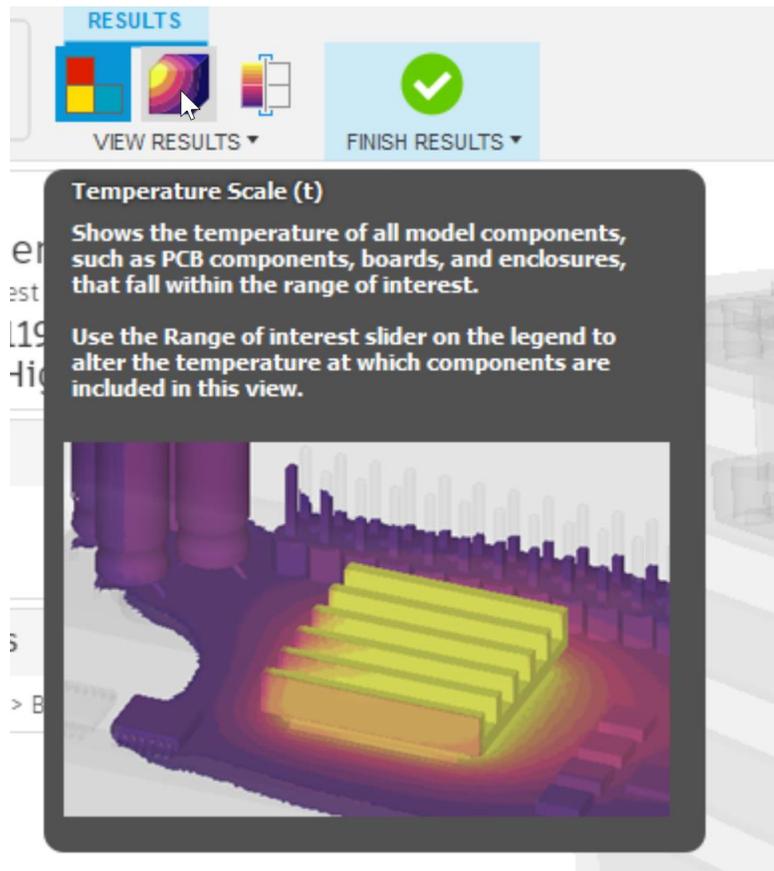
Hover over the 27 watt regulator and you will also see detailed information about the thermal performance of this component.



Hover for component information

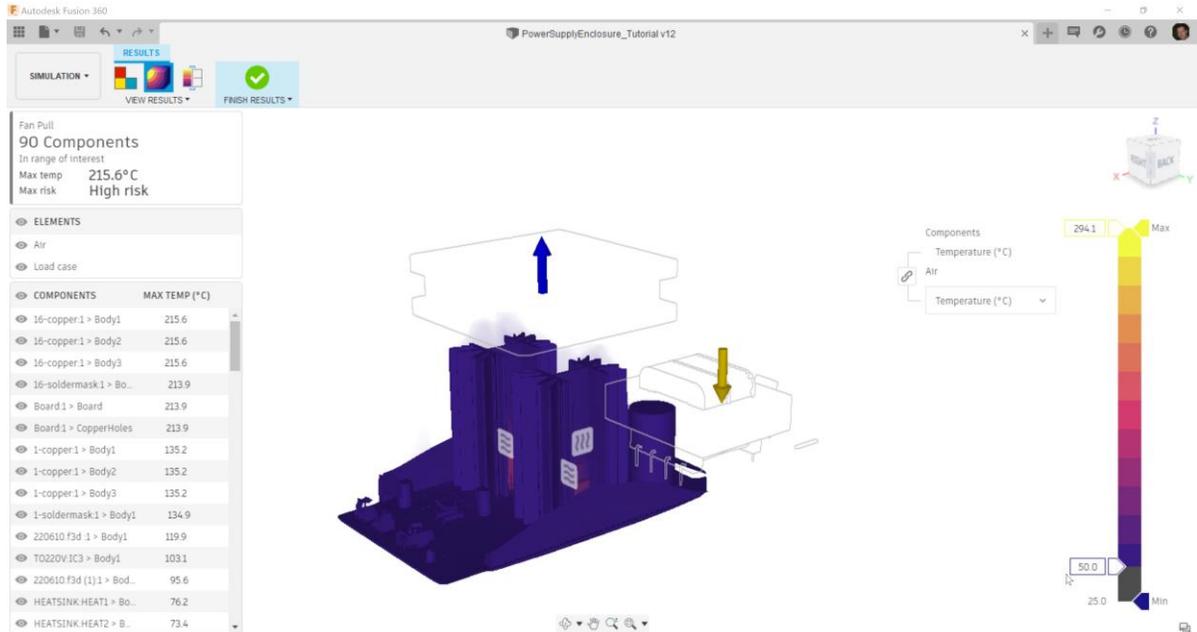
Results View: Temperature Scale

We will next explore the Temperature Scale results view. In the Toolbar up top, select the icon for Temperature Scale.



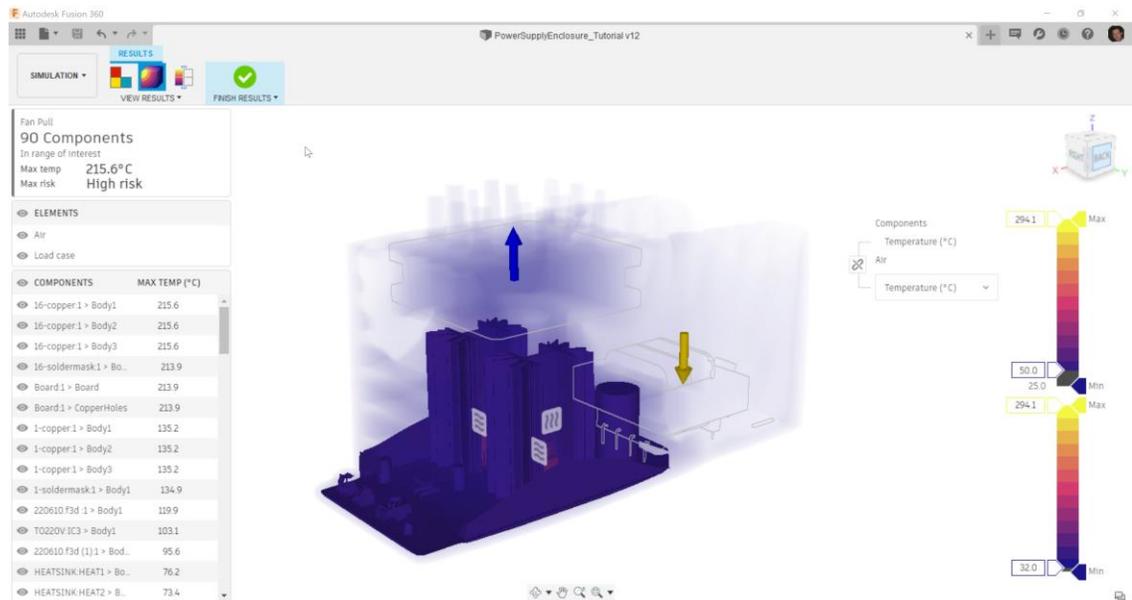
Temperature Scale

You should notice the Components list shows all the components in the model, now ranked from top to bottom by temperature. The elements list now displays both the air and load case available for visibility toggle. The canvas shows all the solids colored by temperature, and the air has a volume render also by temperature. The interaction between the canvas and the components list or the load case is the same as the Risk Scale results view. Let's explore the legend in this view first. Grab the slider for minimum range of interest and drag it to 50C or type in 50 in the number indicator for the minimum range of interest. You should notice that anything below 50C is no longer visible. There is an outline representation of many of the hidden components to help keep context of the overall assembly. The air volume render has also been adjusted to only show air above 50C. The view looks like below.



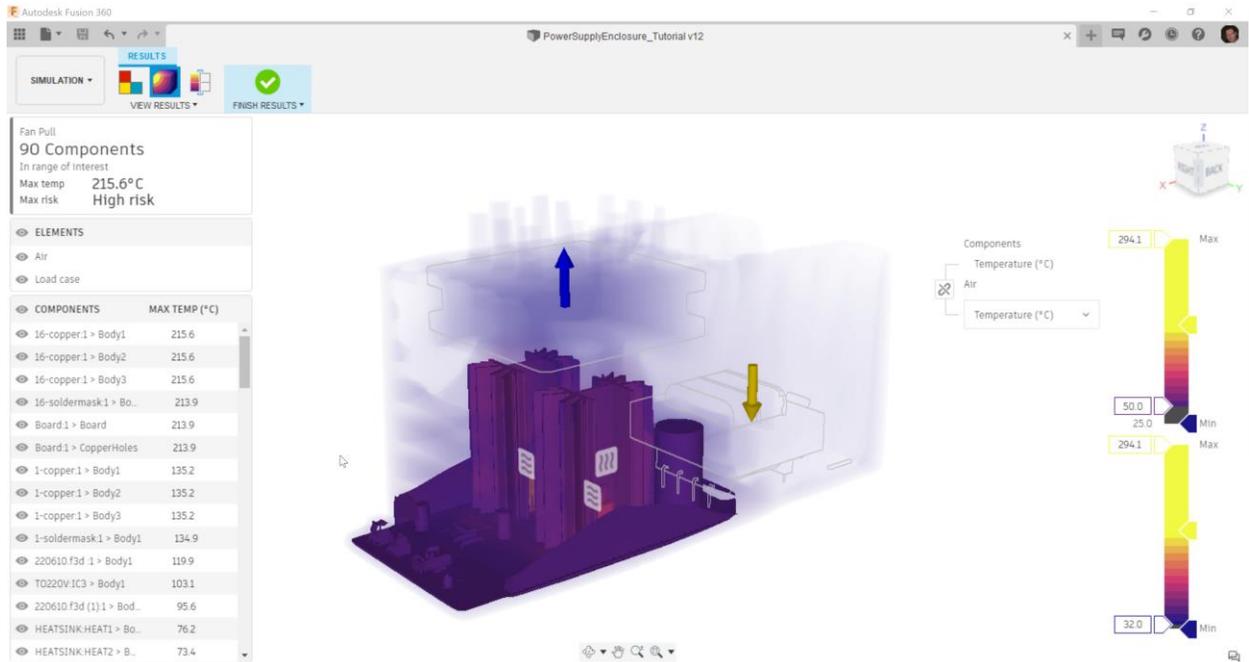
Range of Interest Adjustment

The type of result and legend value is linked for the air and components by default. Toggle the chain link icon next to “Air” to decouple the legend for air and components. The bottom legend is the one now associated to the air results independently. Move the bottom legend minimum range of interest slider down to 32C. You will see the volume render for the air now capture more range of interest and thus see more of the surrounding air volume render.



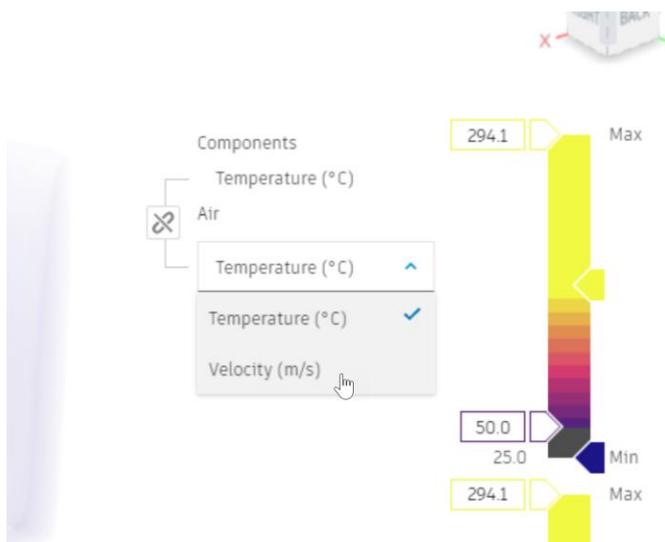
Decoupling the air and comonents legends

Next let's change the legend for the color thresholds. Grab the max threshold slider on either legend and move it down about halfway. More color differentiation on the lower end of the scale now shows and anything above the threshold will be bright yellow. Image below for reference.



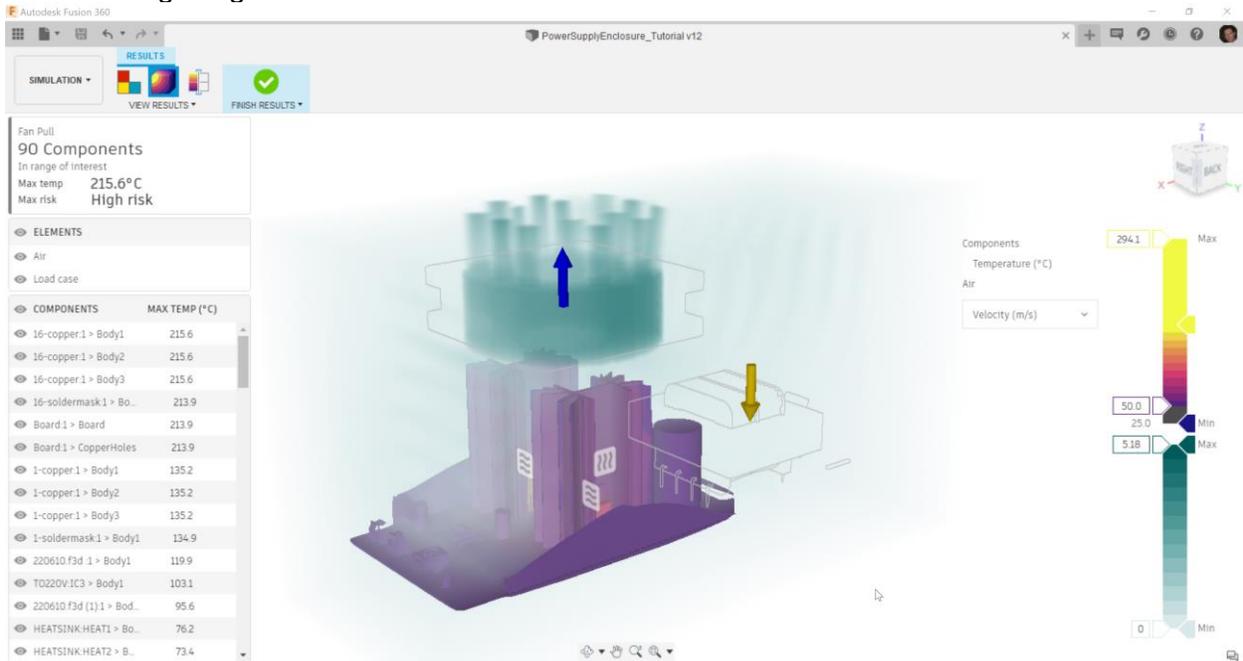
Effect of legend threshold adjustment

Next we will change the value shown for air from Temperature to Velocity by electing the dropdown under "Air" in the canvas and changing it to Velocity



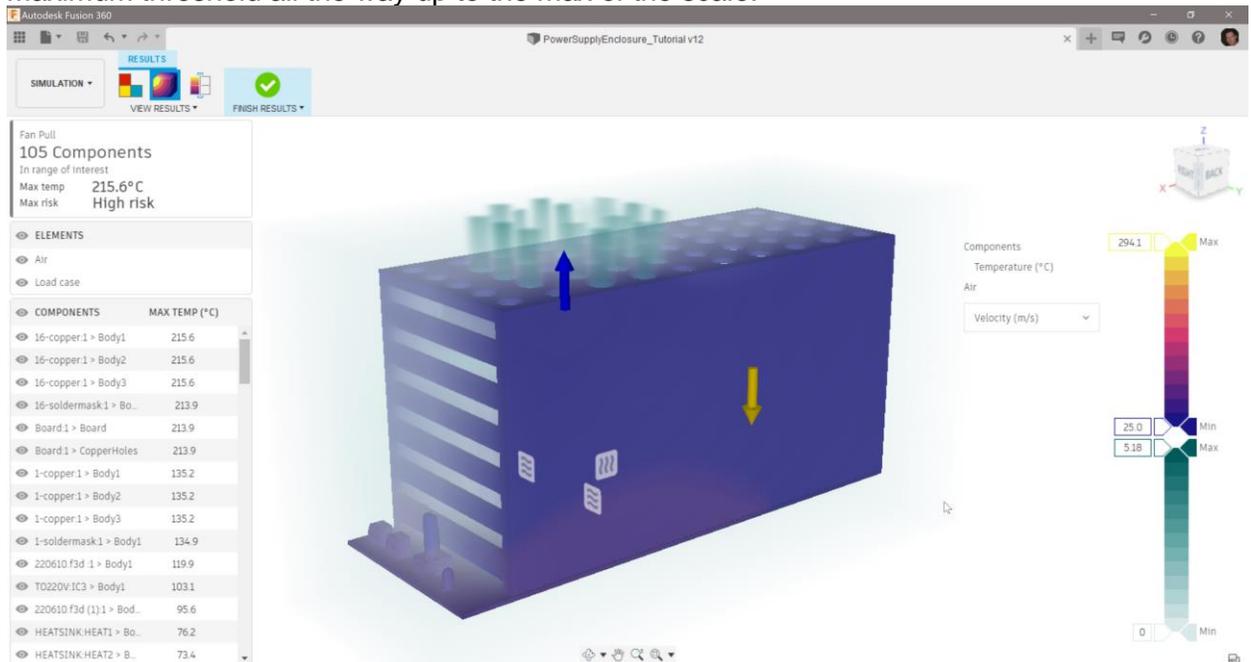
Changing the result display type for air

The resulting image will be as below



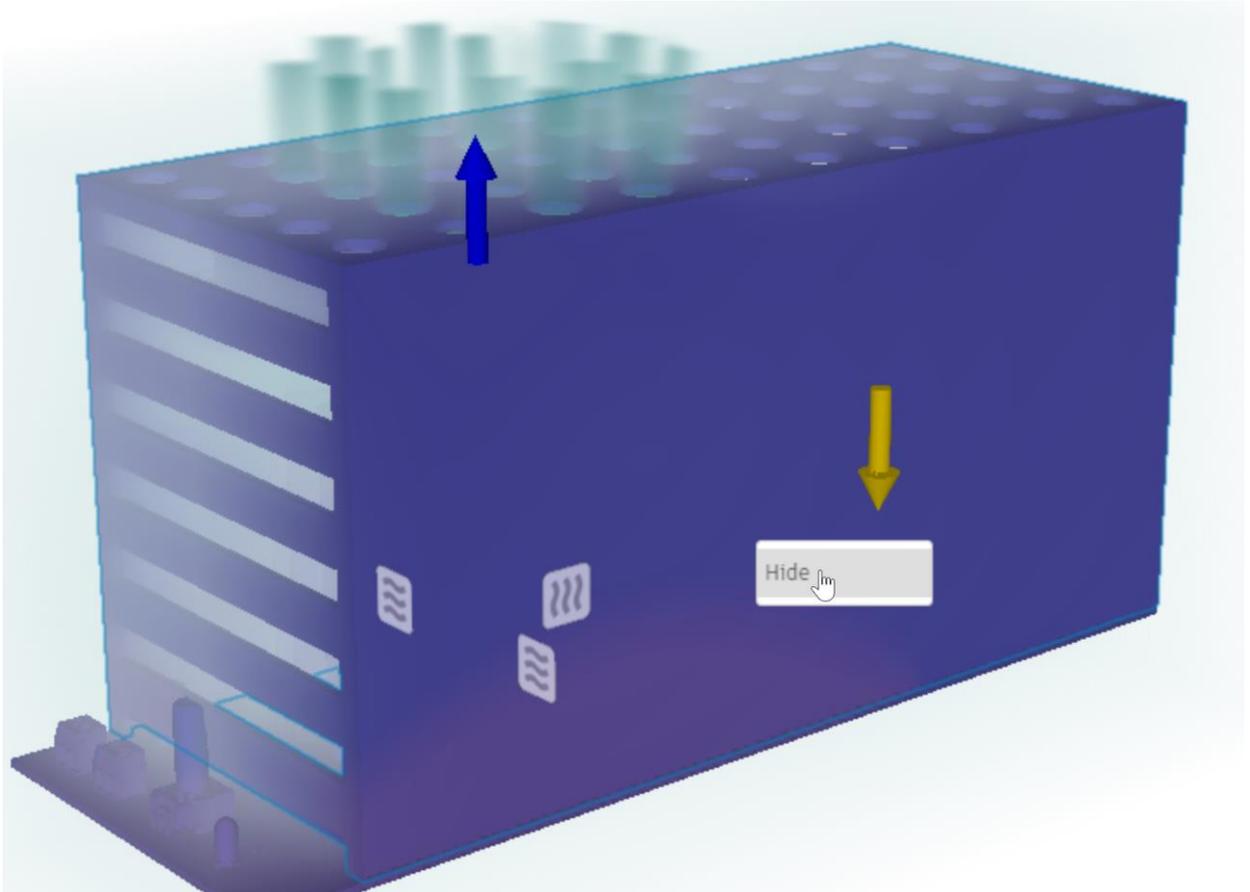
Air Velocity Volume Render, Components Temperature Display

Now let's put the Temperature Legend minimum range of interest to 25C and the Maximum threshold all the way up to the max of the scale.



Legends back to respective min/max values

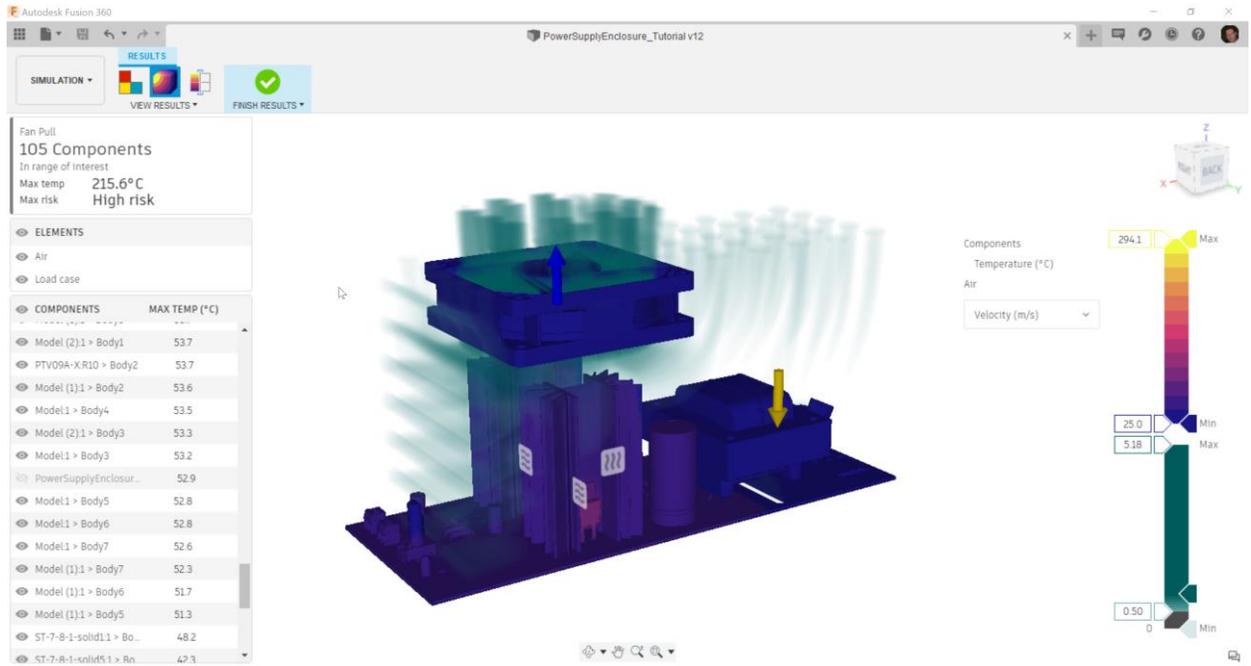
Now the enclosure hides too much of the action, so we will hide that component individually. We could do this in the component list, but rather than hunting for the correct component name there, simply RMB on the enclosure in the canvas and select “hide”.



Hiding an individual component

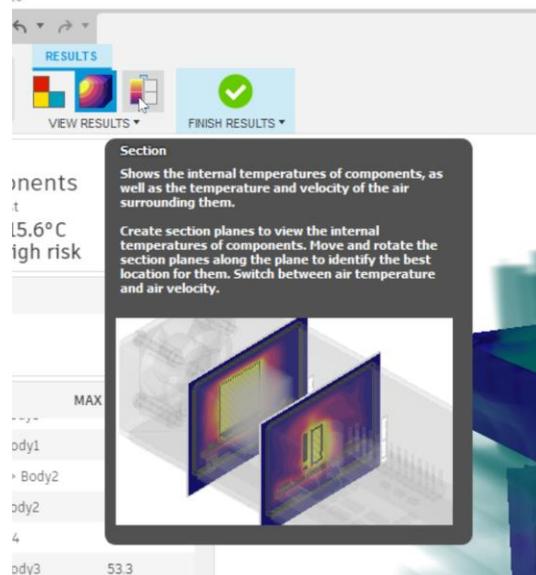
Notice the enclosure is still selected until you LMB somewhere else in the canvas. This is so if you want to unhide that component you just hid, you can easily scroll to it in the components list and it will be highlighted. You could (but don't) toggle the visibility easily in the components list.

Next we will adjust the volume render to accentuate the higher velocity regions. Grab the range of interest for velocity minimum and adjust it to 0.5 m/s. Also adjust the velocity maximum threshold down as far as you can using the slider on the upper right hand side of the velocity legend. The results will look like below.



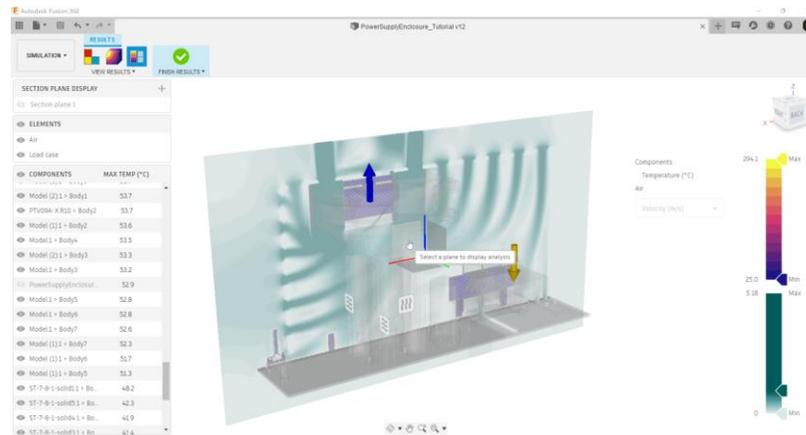
Section Results

Another option available for viewing results is to look at the results of a section on the model. Select “Section” in the toolbar.



Selecting Section Results

Since this is the first time to enter this result type on this model, there will be a triad of planes waiting for you to pick a plane for the first section to appear. Hover over the plane normal to y-axis (xz plane). Click on that plane.



Assigning the section view plane

A cross section of air colored by velocity and temperatures of the components along that plane show up. Notice that component visibility of the enclosure persists. It is hidden right now, even on the plane. Toggle the eyeball next to the “Components” in the components list to make all components visible. Now you should see the enclosure as part of the section view. You can adjust the threshold on the velocity legend as desired to better see the velocity in the air on this section. Notice there is a grab handle on the canvas for translation and for rotations. You can move the plan around as desired using these controls. Also, we can add more planes via clicking the “+” In the Section Plane Display Header. Toggling Air visibility in this mode also is honored. One thing to note is that while the plane is selected, LMB on any component will center the plane on that component. One other thing to note is that hovering over the plane will display the result value magnitude and other relevant data at the cursor location.



Cursor hovered over the regulator section view

Changing over to other results

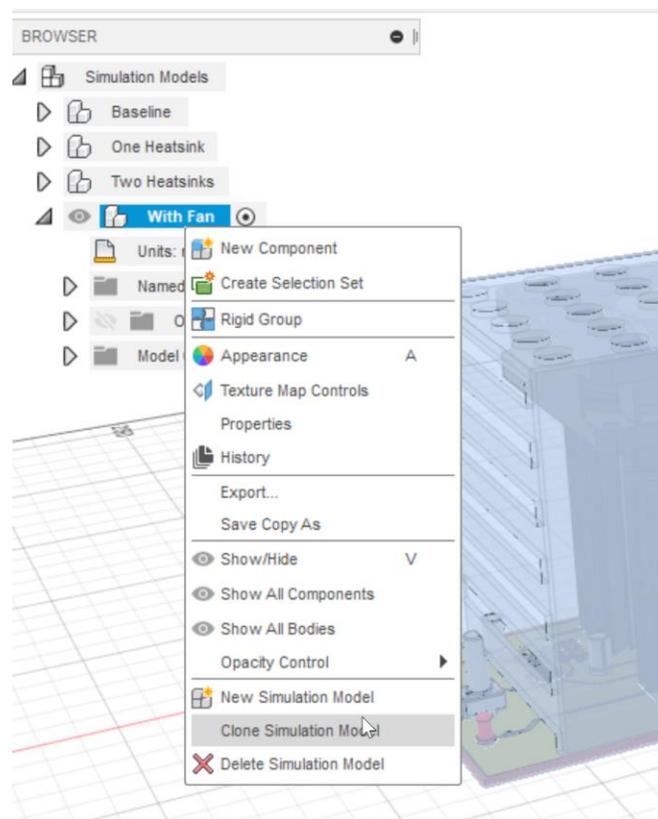
To look at the other experiments results, just select “Finish Results” in the toolbar and then activate one of the other simulation studies and select “Results” in the toolbar.

Bonus: Make higher fidelity regulator

When we looked at results we saw that the leads on the regulator were really hot due to the fact we modeled the regulator as one piece. This meant the legs were not modeled as copper and were modeled as generating heat. We will use the simplify environment to change the regulator and resistor to have separate copper legs and not have them generate heat.

Clone the Design (again)

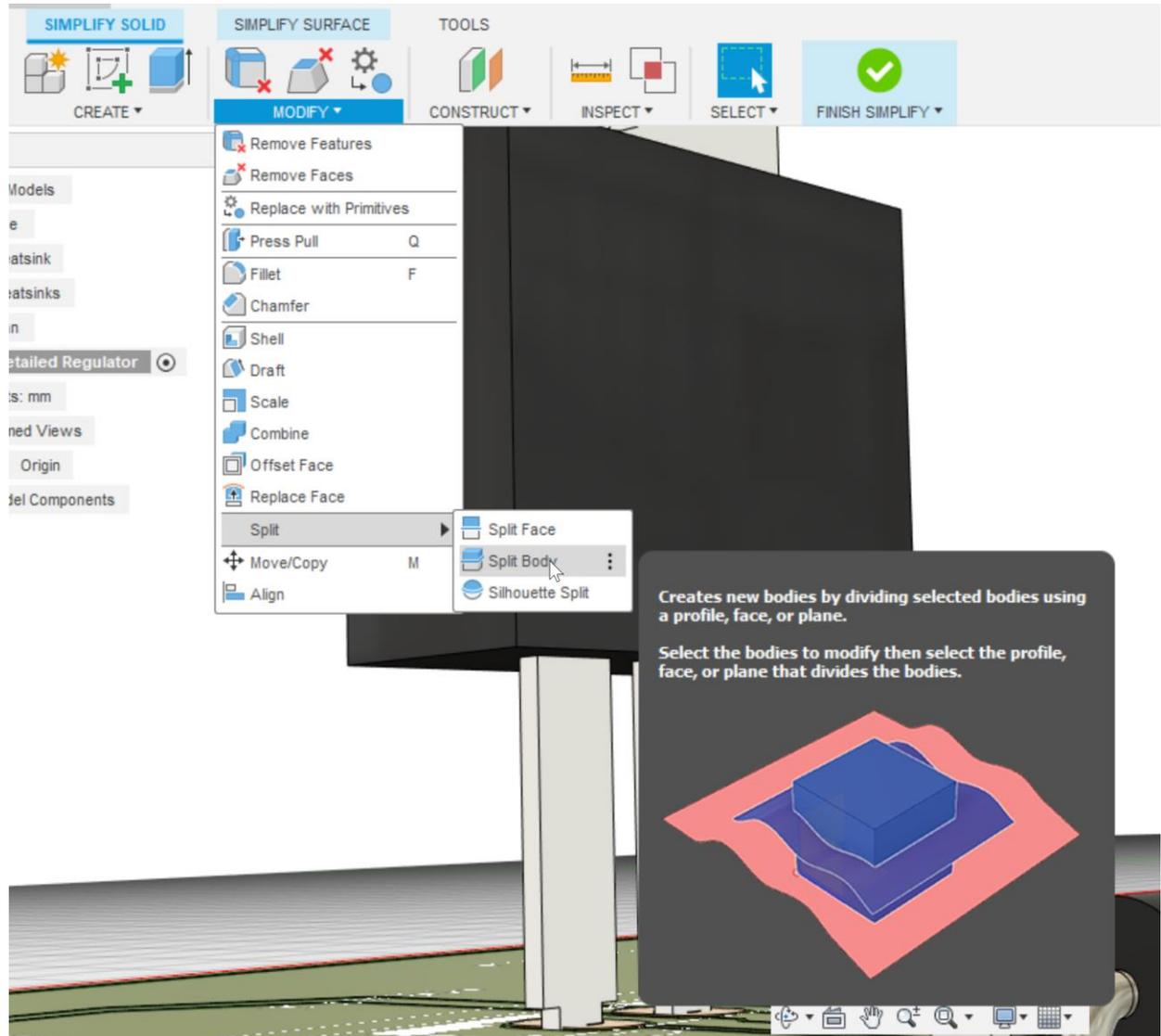
Select “Simplify” in the toolbar. Then RMB on “With Fan” in the browser tree. In the context menu that pops up, select “clone simulation model”. Select “OK” on the dialog that pops up. The new simulation model is an exact copy of “With Fan”. It will be named “Simulation Model5” or “Simulation Model6”, etc. Let’s rename it to “Detailed Regulator”



Clone “With Fan”

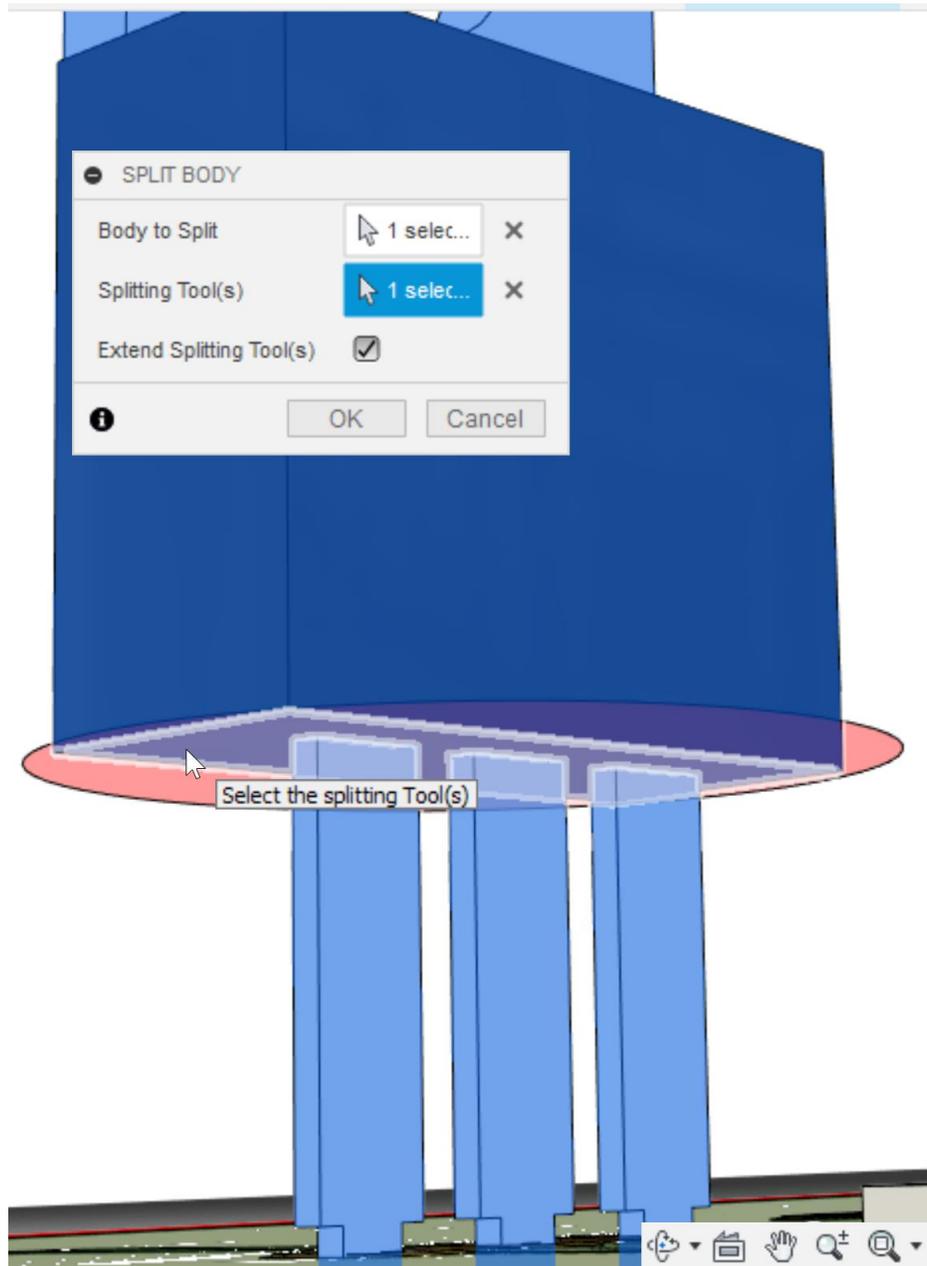
Modify the Geometry

Click on the dropdown “Modify” and select “Split Body”

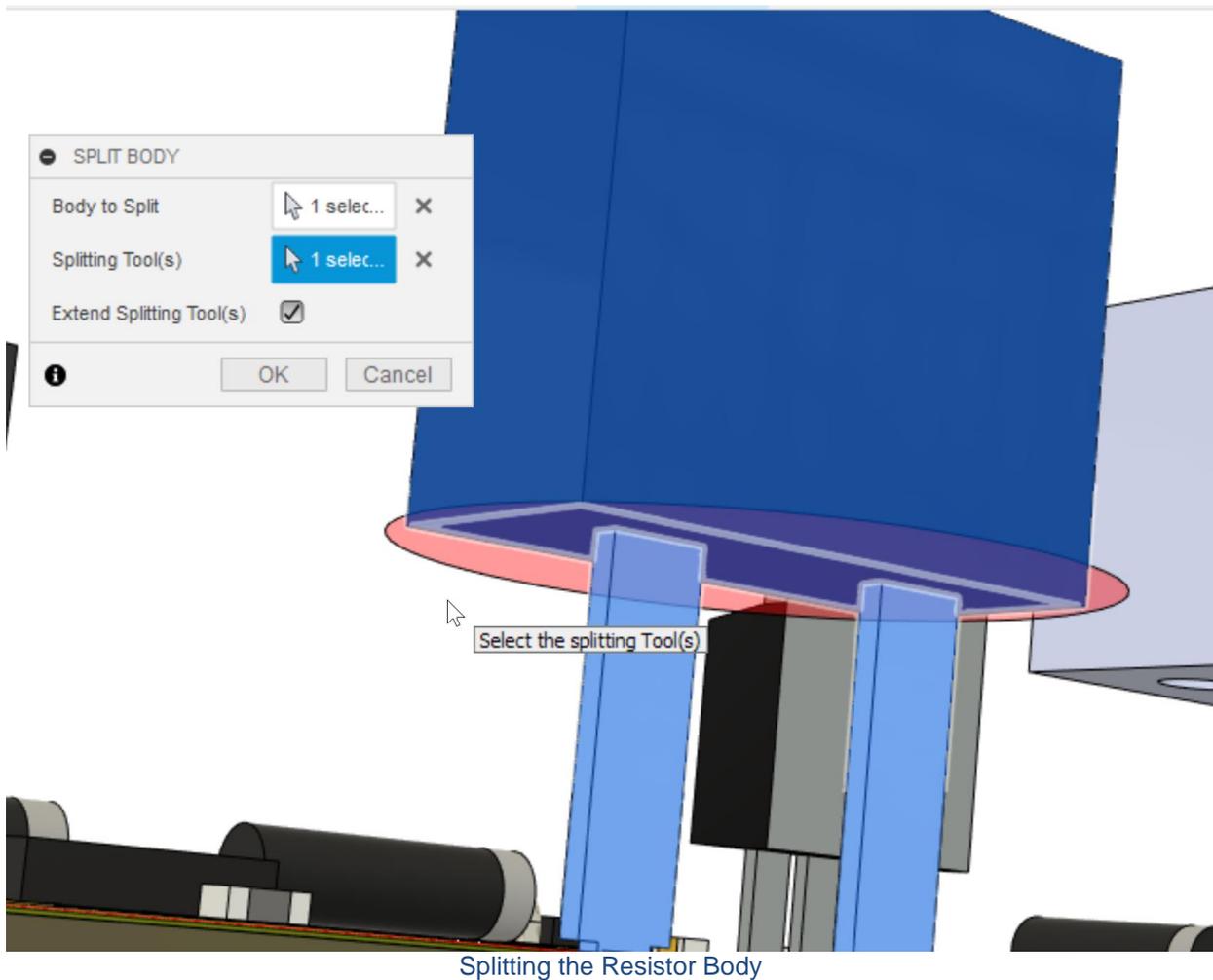


Splitting the Regulator Body

Select the regulator body as the body to split and the bottom planar face where the legs protrude as the splitting tool. When you select OK, the three legs will now be independent bodies. Repeat these steps to split the body for the resistor attached to the other large heat sink, then Select “Finish Simplify”.

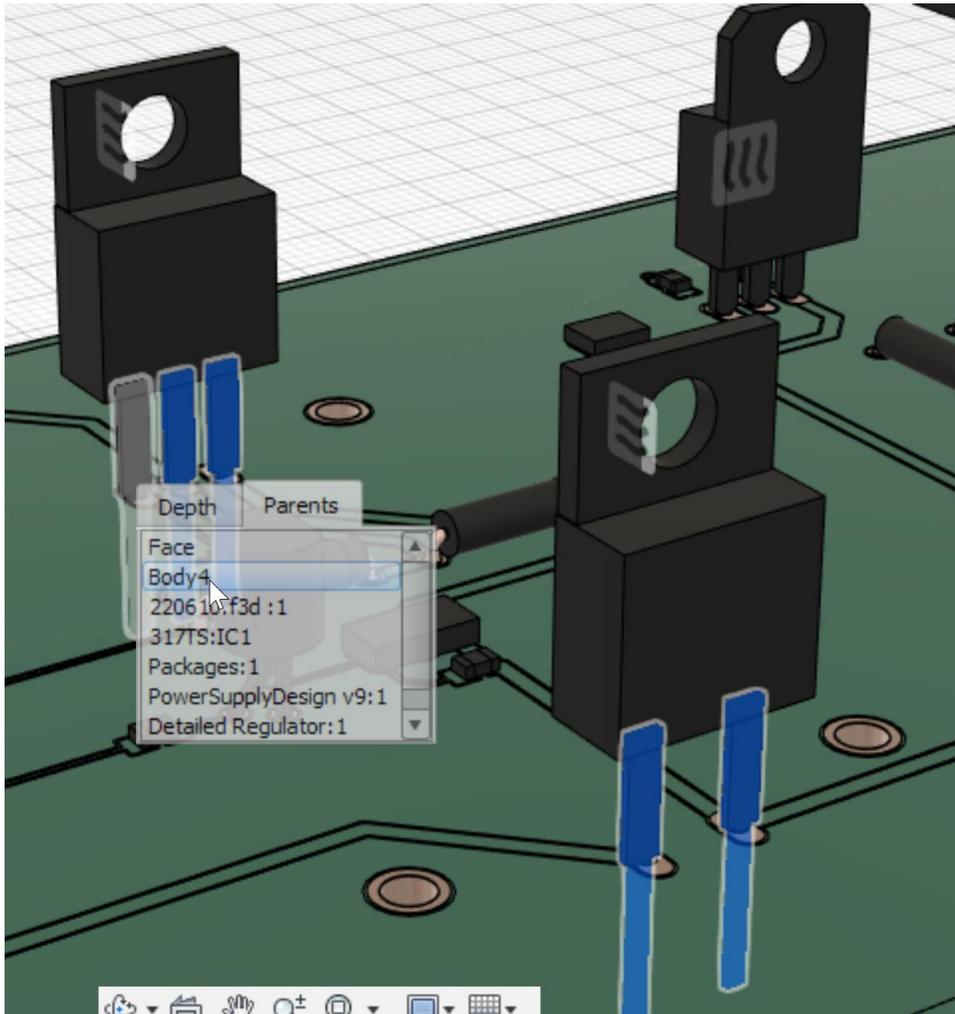


Splitting the Regulator Body



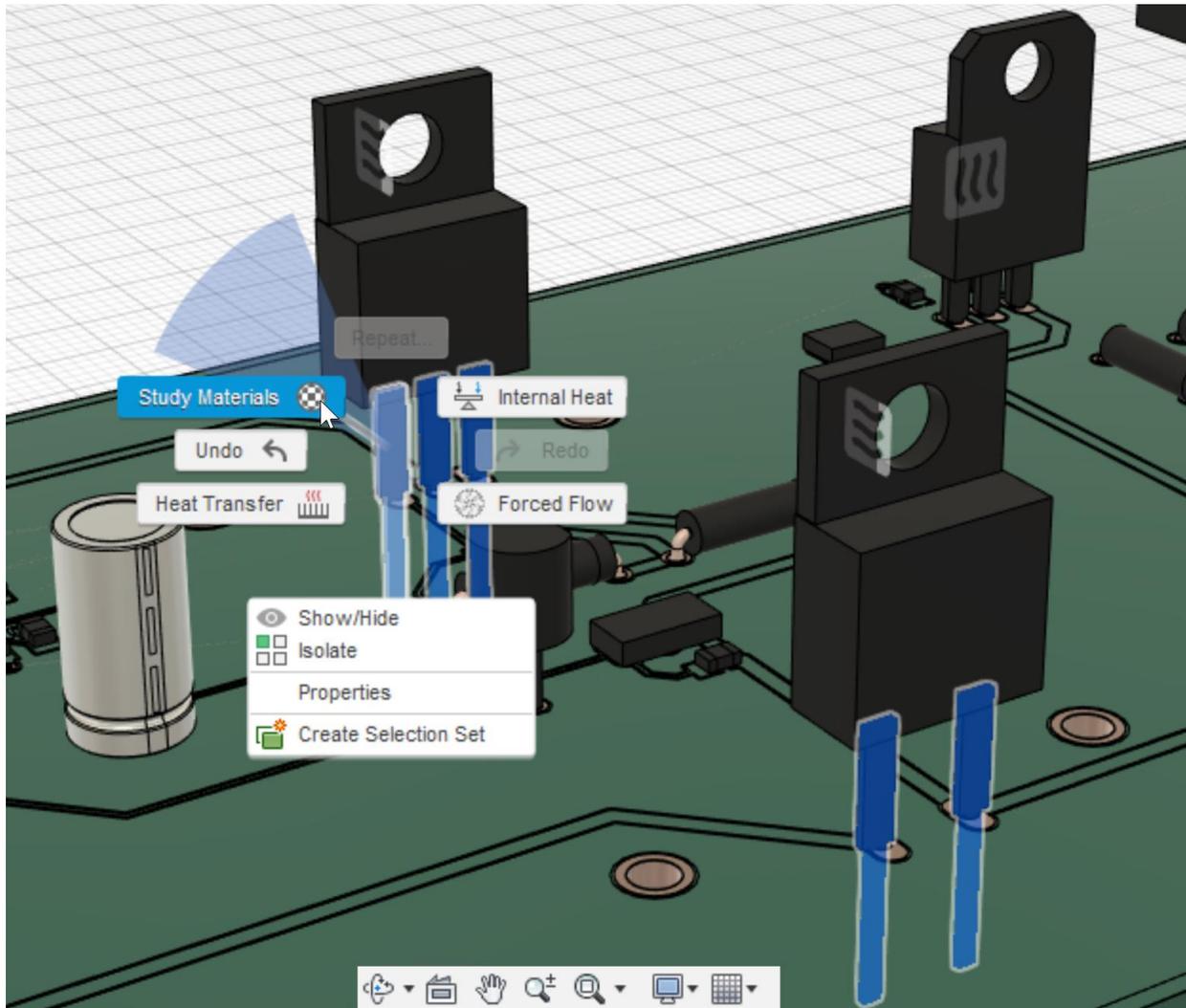
Materials

Since we've just added 5 new bodies (three legs on the regulator and 2 legs on the resistor), we need to verify the material assigned to the legs and the regulator and the resistor. Visually we can see they are the black material indicative of "discrete component", so that means we should change the material of the legs to copper. Select all 5 leg bodies. Tip: hold long left click on each leg, then select the parents tab to select the each leg body. If you hold CTRL while doing this, you will be adding to the selection so you can have all 5 bodies selected.



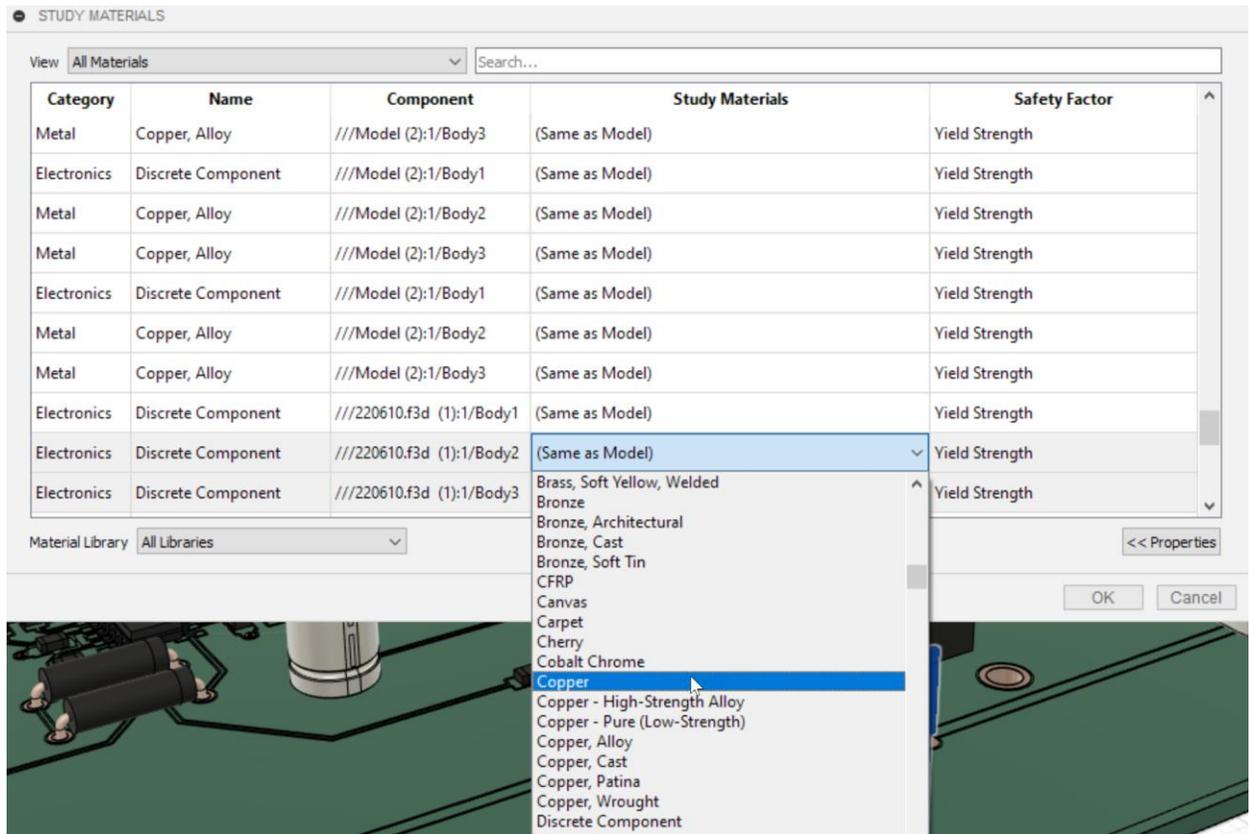
Select the leg bodies

RMB to bring up the marking menu and select "Study Materials"



Invoke the Study Materials Dialog

Change “(Same as Model)” for the prehighlighted components (the leg bodies) to “Copper” and select OK. Visually you should see the legs change to a copper color.



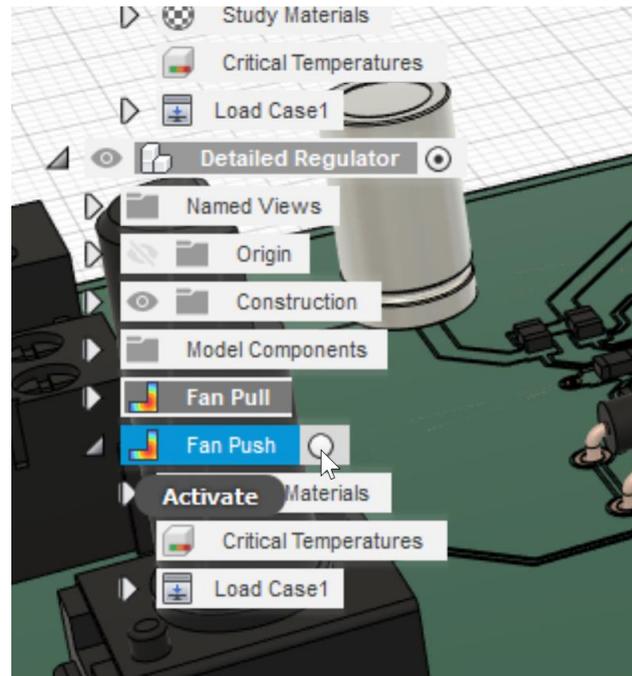
Changing the leg bodies material

Power Dissipation (Heat Generation)

The model has the heat generation assigned to the main bodies of the regulator and the resistor, so no update of the heat generation is necessary.

Repeat Material Update for “Fan Push” Study

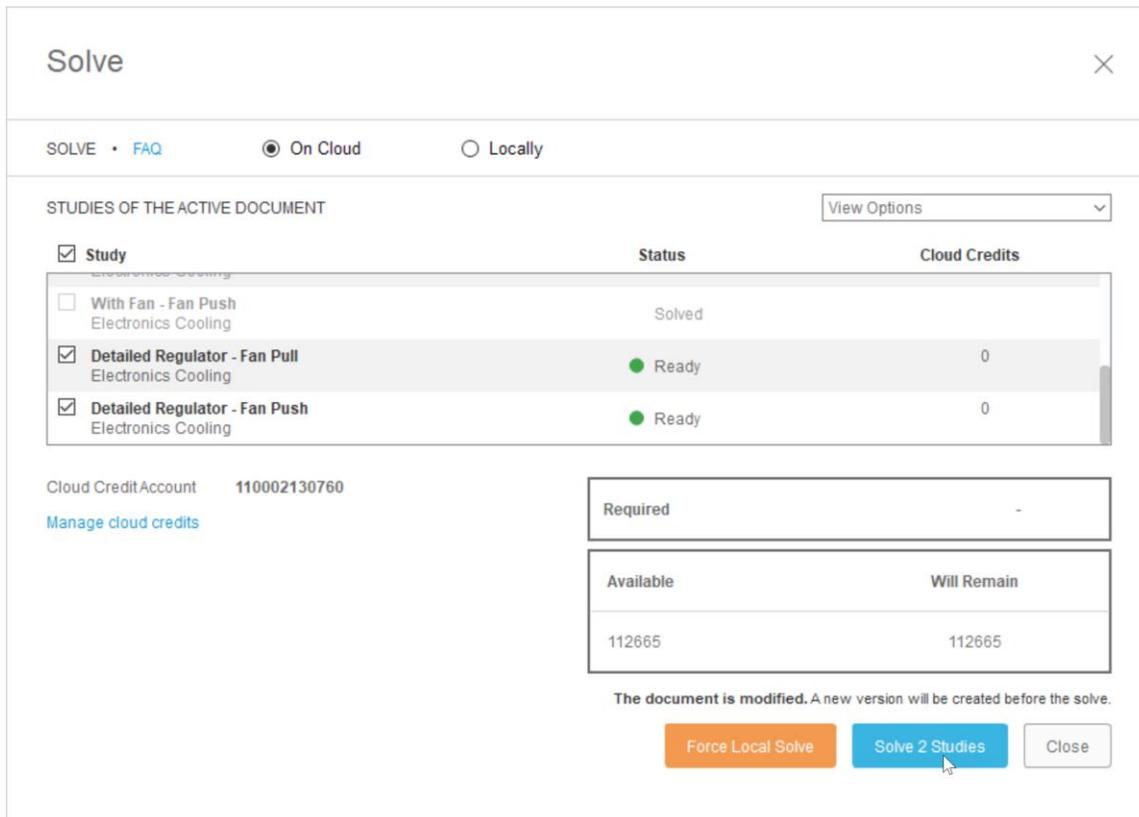
The Simulation Model we cloned had two studies (“Fan Pull” and “Fan Push”). We just edited the materials for “Fan Pull” in our new “Detailed Regulator” Simulation Model. Recall that each study can have independent material assignments which enables the investigation of material choices impact on the thermal study. Activate “Fan Push” and repeat the leg material reassignment for the “Fan Push” study.



Activating the “Fan Push” study

Solve the New Studies

The new experiments are ready to go. Let’s click “Solve” in the toolbar. Select two new studies. Select “Solve 2 Studies”.



Solving the 2 New Studies

Follow the results viewing guide from the previous study to see the new results when the solve is complete. You will notice much lower maximum temperatures that is more realistically located on the core of the regulator and resistor as opposed to the legs. The core temperature of the regulator and resistor won't be drastically different.

Fusion Electronics

Please refer to the class recording to see how the commands are used practically. In all honesty this design could be autorouted since there are no critical traces and the frequencies involved are so low. The main concerns for this design revolve around thermal management and making sure the traces are thick enough to carry that max 1.5A current.

Creating the Schematic

Almost all of the components come from EAGLE standard libraries. If there is any components you wish to glean and use for your own designs you can simply copy them to your personal libraries. Make sure you have the library you want to copy into open in Fusion 360 and then right click on the component in the schematic/board and select copy to library.

TIP: On the schematic editor, make sure to always stick to a 0.1”(2.54mm) grid. This is the default. If you deviate from this grid setting, it will become difficult to establish your

connections on the schematic. The same is true for the symbol editor in the library which is beyond the scope of this handout.

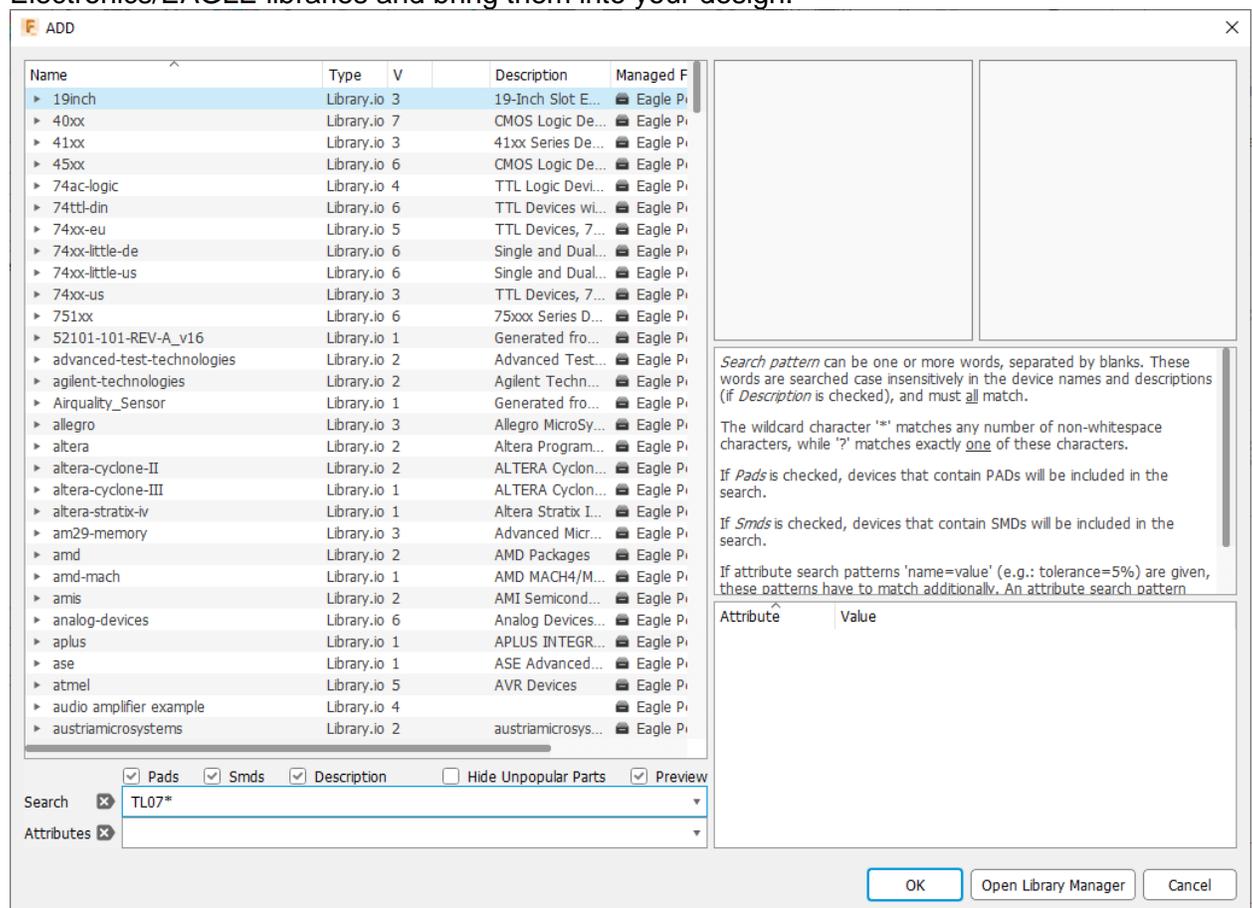
Commands used when making the schematic

Net

This is one of the most important command in the schematic editor, it is used to connect your components together. The command snaps to your component pins which simplifies making the connections.

Add

This command allows you to choose components from the large number of Electronics/EAGLE libraries and bring them into your design.



TIP: THE SEARCH IN THE ADD COMMAND USES THE * AND ? CHARACTERS AS WILDCARDS. USE THESE TO GENERALIZE YOUR SEARCH

Creating the PCB Layout

The layout is greatly simplified when you have a good schematic since all of the connections you defined there are automatically transferred to the layout. Some general rules of thumb for successful layout:

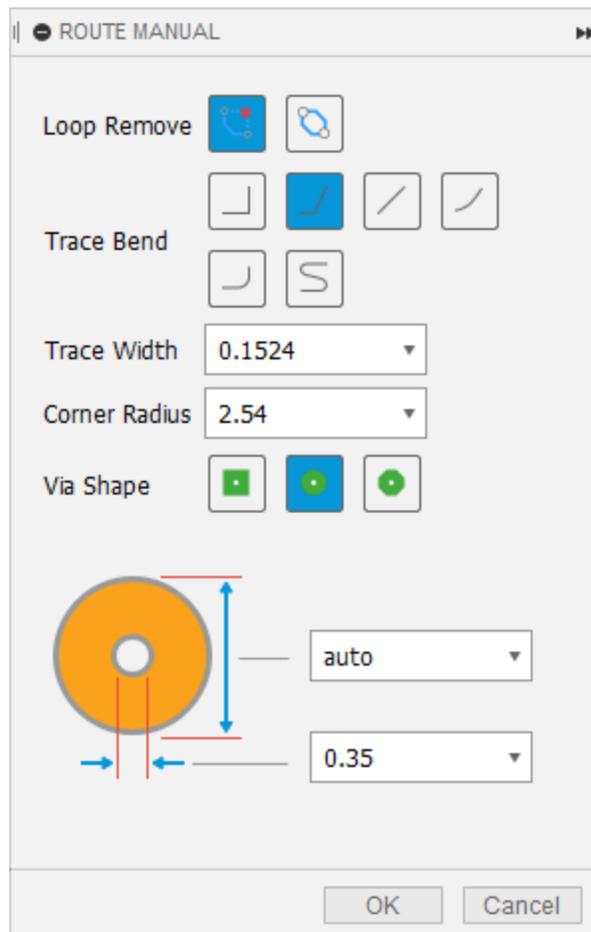
- Keep trace lengths short
- Try to route the circuit in functional groups this will allow you to make sure you can keep related components together which helps with the first point.
- Setup your design rules to match your board manufacturer early in the layout process to avoid unnecessary rework down the road.

Commands used in the layout

Ignore, Push, and Walkaround Violators

These environment modes are a powerful unique feature of Fusion 360 Electronics. In most ECAD tools these modes are tied to routing engine and are only available in that engine. However, in Fusion 360 Electronics these modes apply to the environment so ALL commands can leverage them. The demo in our class highlights the use of this feature.

Route Command



This is one of the key commands of the PCB editor. With this command active, clicking on an airwire allows you to convert it to copper. The loop remove function in the image above is useful for rework or exploring alternative options since it allows you to redo traces without having to rip them up.

The route command has many options that simplify the routing task, for example if you press the backspace key you can undo the last segment you routed, continuously pressing the backspace key can take you all the way back to the start of the trace. You can also use the space bar to transition between layers and much more.

You can also benefit from several sub-commands that are especially helpful in more complex cases.

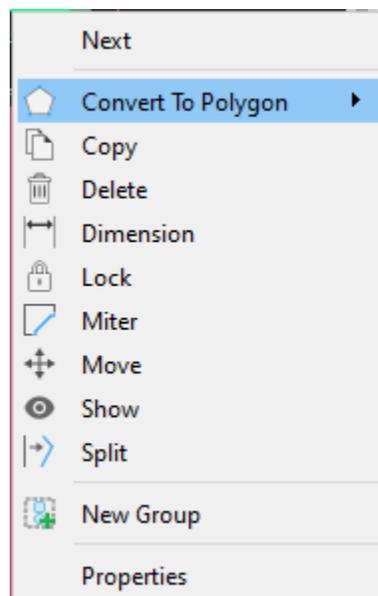
- Differential Pair – Signals are routed as differential pairs in order to improve their noise immunity along with other benefits. This mode allows you to route both members of the pair at the same time.
- Route Multiple – This mode as its name implies allows you to route multiple traces together at the same time. This is especially helpful when routing busses.

Fusion Documentation covers this command in detail.

Polygon

Polygons or copper pours are a vital part of printed circuit design. They provide many benefits including EMI shielding, low impedance return paths, heat dissipation and much more.

To create a polygon, simply click on the polygon icon and click wherever you want to place the vertices of the polygons. More often than not it's desirable to have a polygon's outline to follow the contour of the board. As long as the board outline has been correctly defined (closed contour, no gaps) you can create a polygon by right-clicking the outline. In the context menu you'll see the option convert to polygon.



TIP: THE CONVERT TO POLYGON OPTION LET'S YOU EITHER DUPLICATE THE OUTLINE ON ANOTHER LAYER OR REPLACE IT. FOR POLYGONS COPY IS THE BEST OPTION

CAM Export

Traditionally, generating manufacturing data for PCBs has been a complex task because every main feature of the board gets exported as it's own file. In an effort to simplify this process the CAM Export command analyzes your design and then selects most appropriate template to process your manufacturing data. In one click it will generate all the manufacturing files needed to manufacture your board. Besides the gerbers it generates drill files, a bill of materials, as well as a pick and place file.

Mechanical Assembly

Once you have created the 3D PCB by clicking the view 3D PCB option in the 2D PCB you need to insert it into another design in order to create an enclosure. In other words, the 3D PCB file should only contain the 3D PCB.

See the video for all of the details of how the 3D PCB is inserted into a new file. The following commands were used to create the mechanical assembly.

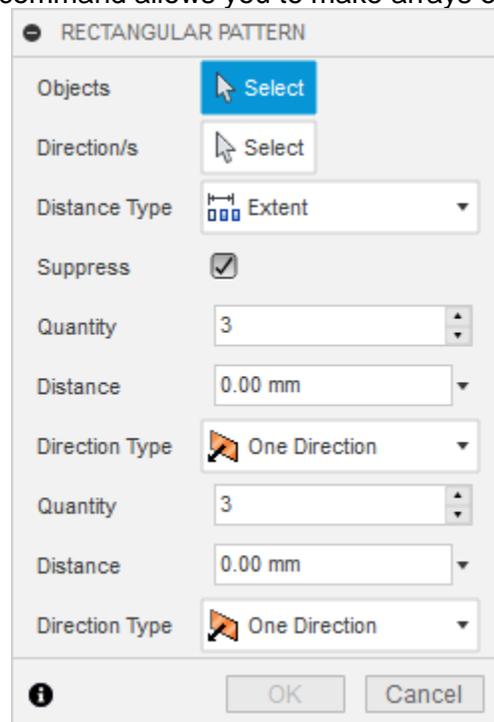
Sketch Environment commands

Project

The Project command allows you to take existing geometry and trace it into the sketch. By referencing existing geometry matching accuracy is guaranteed. See the full description here: <http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-6EE7B230-A280-45B7-8868-D96E4CE44B62>

Rectangular pattern

The rectangular pattern command allows you to make arrays of drawn objects.



This command was used to create the openings on the top and side of the enclosure. There is also a circular array command for making polar arrays. For more details here: <http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-24C29C45-D9BC-47BF-B409-36BDC7733989>

Line and Circle commands

These are fundamental drawing operations that are available in most MCAD programs but when coupled with dimensioned constraints they allow complex shapes to be built up very quickly and efficiently.

Line Documentation: <http://help.autodesk.com/view/fusion360/ENU/?guid=SKT-SKETCH-CREATE-LINES>

Circle Documentation: <http://help.autodesk.com/view/fusion360/ENU/?guid=SKT-SKETCH-CREATE-CIRCLES>

Dimension Command

The Dimension command makes it really easy to define precise lengths and dimensions for objects in your sketch. This is typically easier than having to draw the lines to length in the first place. Dimensions documentation:

<http://help.autodesk.com/view/fusion360/ENU/?guid=SKT-SKETCH-CREATE-DIMENSIONS>

Solid Environment Commands

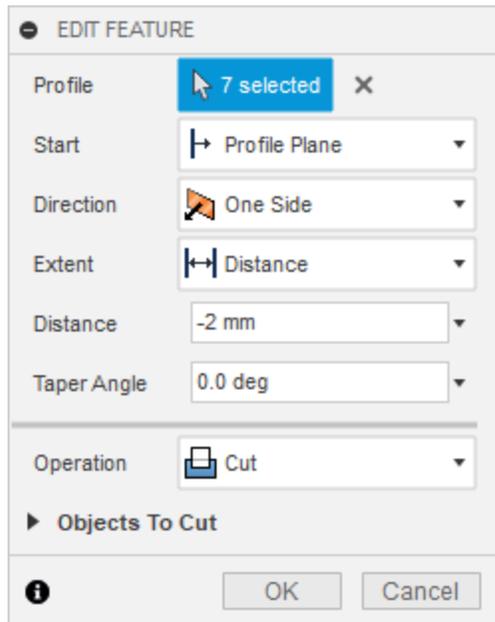
Offset Plane

The construct menu contains many commands that you can use as references to create new geometry. In this design, I used the Offset Plane to define the top of the enclosure and then extrude down. Offset Plane Documentation:

<http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-1C665B4D-7BF7-4FDF-98B0-AA7EE12B5AC2>

Extrude

Also known as Push-Pull this is the most basic command to convert a sketch into 3D geometry.



The EXTRUDE command is extremely powerful. It can be used to cut existing geometry like in the above example or it can create a new body. I used it in combination with the patterned sketches to make the openings in the enclosure body. One thing that can make this command even more powerful is if the extent is set to be a piece of geometry instead of a fixed distance. In that scenario if the reference geometry changes then the cut or the extended geometry will change. Extrude documentation:

<http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-A6BD887A-4F83-4EAE-AEF9-D777B94629EF>

Shell

The shell command is exceptionally useful for making enclosures, because it can hollow out a piece of solid geometry. Shell documentation:

<http://help.autodesk.com/view/fusion360/ENU/?guid=GUID-83A14252-0009-42AA-AE7E-435F595DE41B>

CONCLUSION

The goal of this document is to serve as “cheat sheet” of the most important commands we used in this to produce the power supply we demonstrated in the class. In the class, we used ECAD-MCAD workflows, e-cooling and much more. Fusion 360 is an exceptionally powerful tool and many documentation pages, tutorials and examples have been made about it. We hope that this class has given you a glimpse into what is possible with Fusion 360