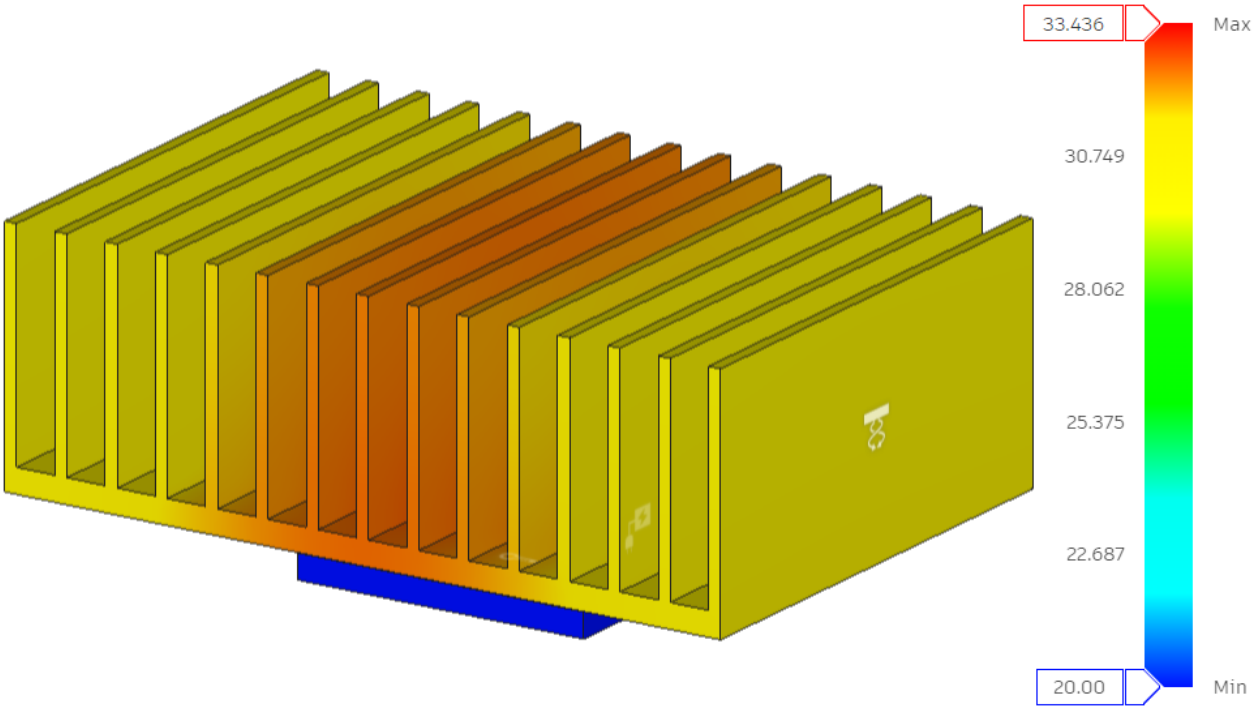
FEA Thermal Simulation for Heatsinks using Fusion

My pretty FEA results should be worth an art credit.



Today's lesson is sponsored by Diggerland in West Berlin, NJ

DIGGERLAND

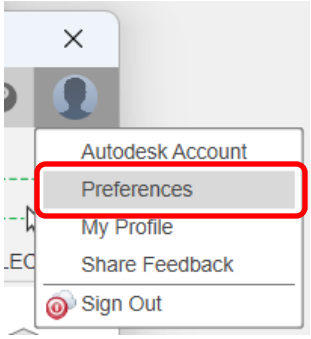
DRIVE RIDE SOAK SLIDE

CONSTRUCTION THEME & WATER PARK

(856) 393-5962
100 Pinedge Drive
West Berlin, NJ 08091

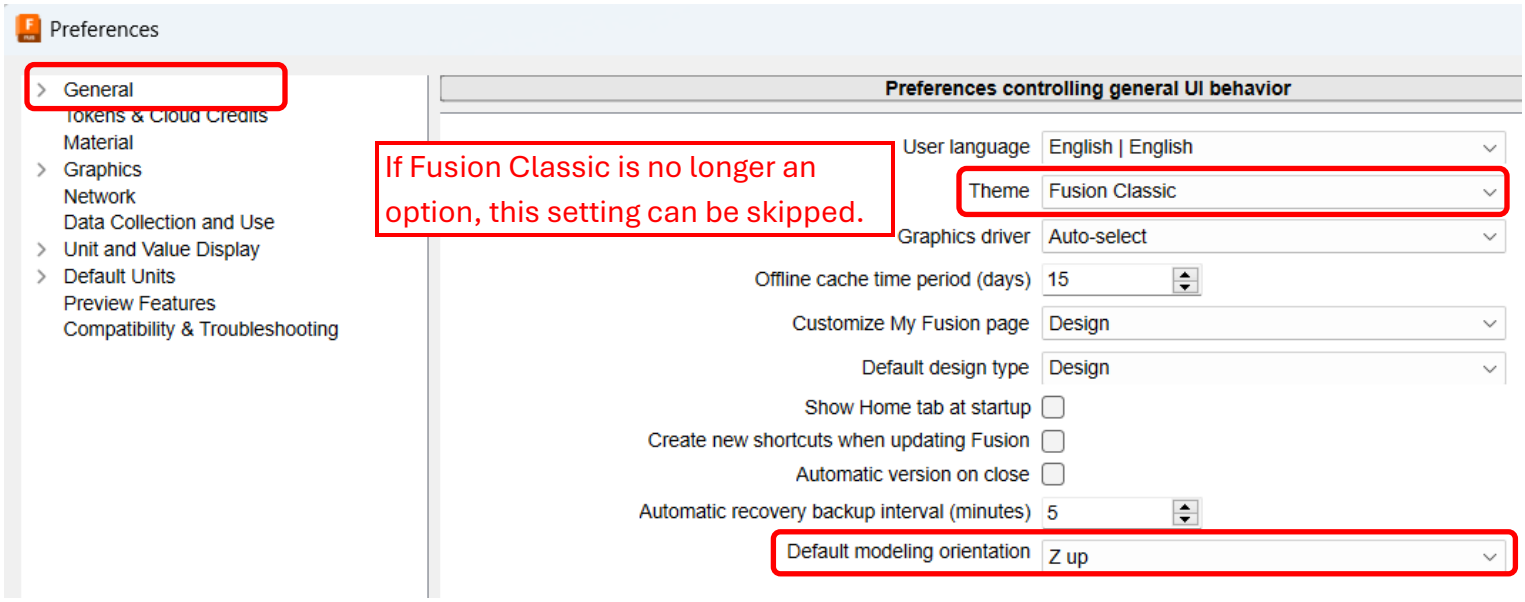
Contents

Setting Fusion Preferences

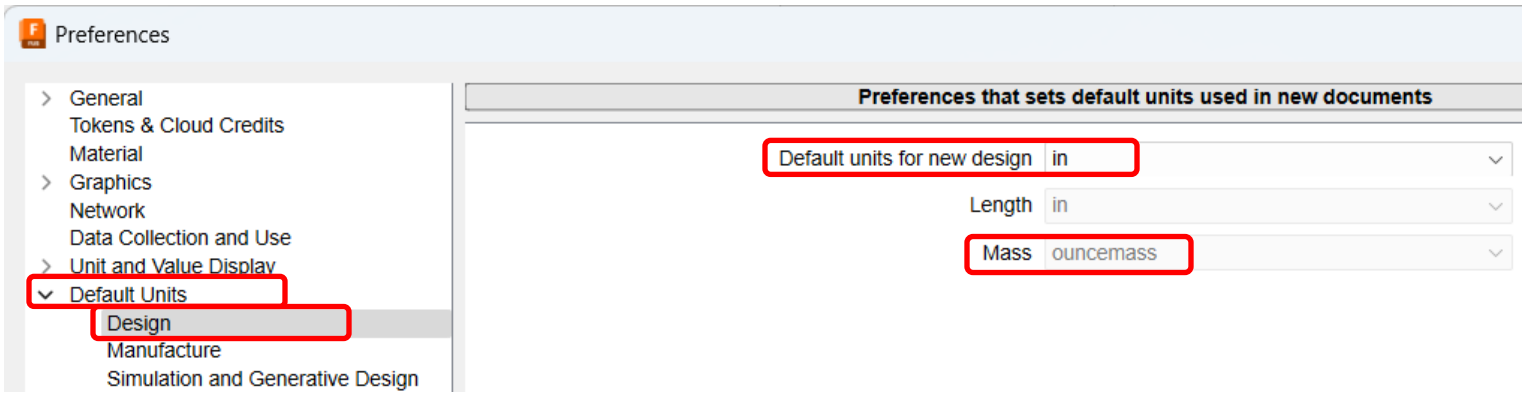


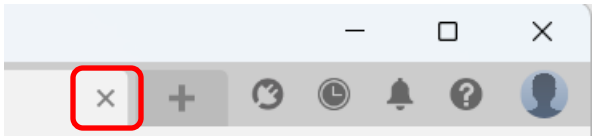
- if this is your first time using Fusion open Preferences by clicking on the top right icon and selecting Perferences

- in the **General** section select **Fusion Classic** for the **Theme** and **Z up** for **Default modelling orientation**



- in the **Default Units** section select **Design** and change the **Default units** to **in** and **Mass** to **ouncemass**
- click the bottom right **Apply** button and then the **OK** button



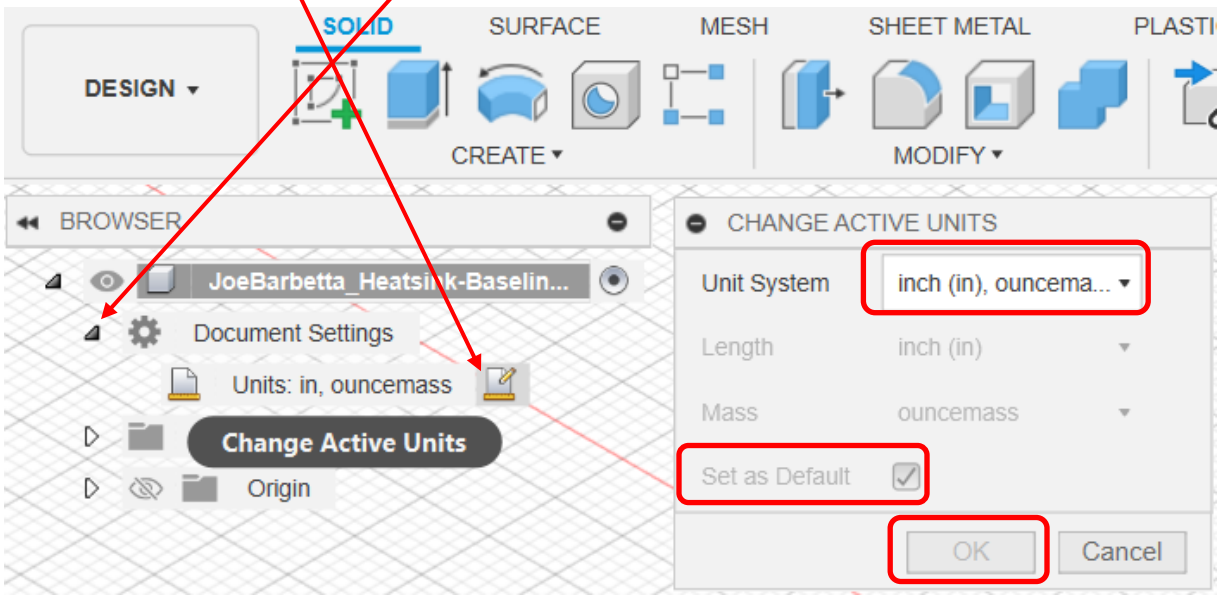


- click on the **x** to close the project, which will start a new project. This is done to allow new Preferences to be applied to the new project.

Starting a Design in Fusion

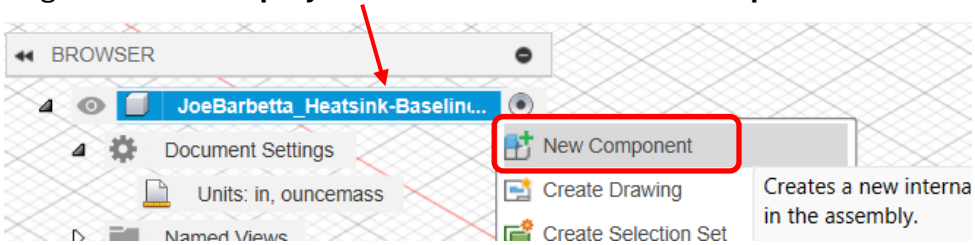
- open **Fusion**. If there is no icon on the Desktop, use the Windows search (magnifying glass icon) and type fusion
- from top **File** icon select **Save** and name the file.
  Use your name followed by **_Heatsink-Baseline** e.g. **JoeBarbetta_ Heatsink-Baseline** (note the use of the underscore)

- in the left **"BROWSER"** click the **arrow next to Document Settings**
- click on the **edit icon** that appears to the left when you hover over **Units**
- ensure **Active Units** are set to **Units: in, ouncemass** and click **OK**. You can also enable **Set as Default** if it is not grayed out.

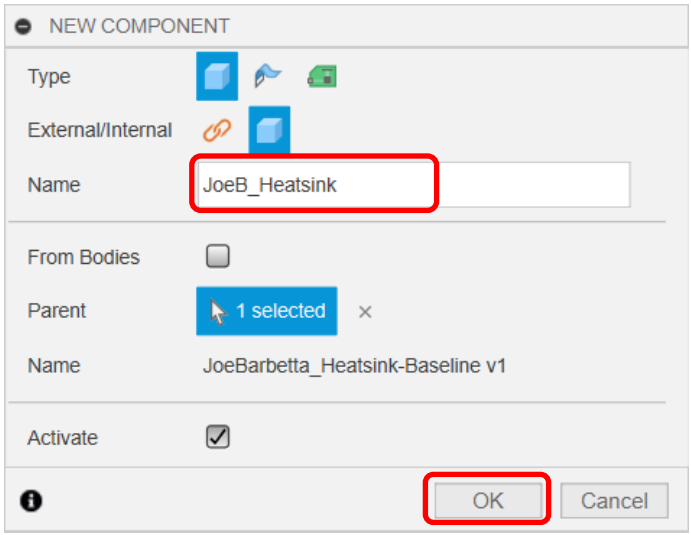


Creating a Component for the Heatsink

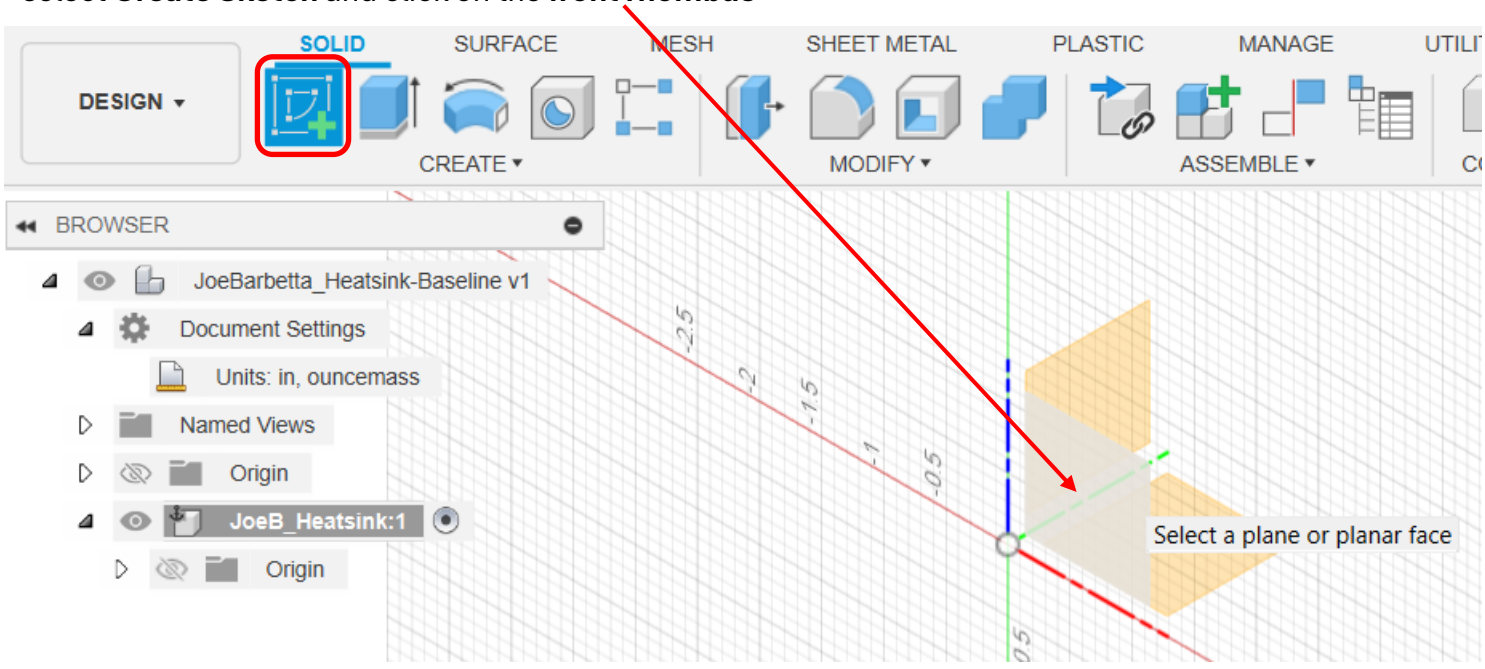
- right-click on the **project name** and select **New Component**



- for the Name enter your **first name and last name initial followed by "_Heatsink"**, e.g. "JoeB_Heatsink"
- click **OK**

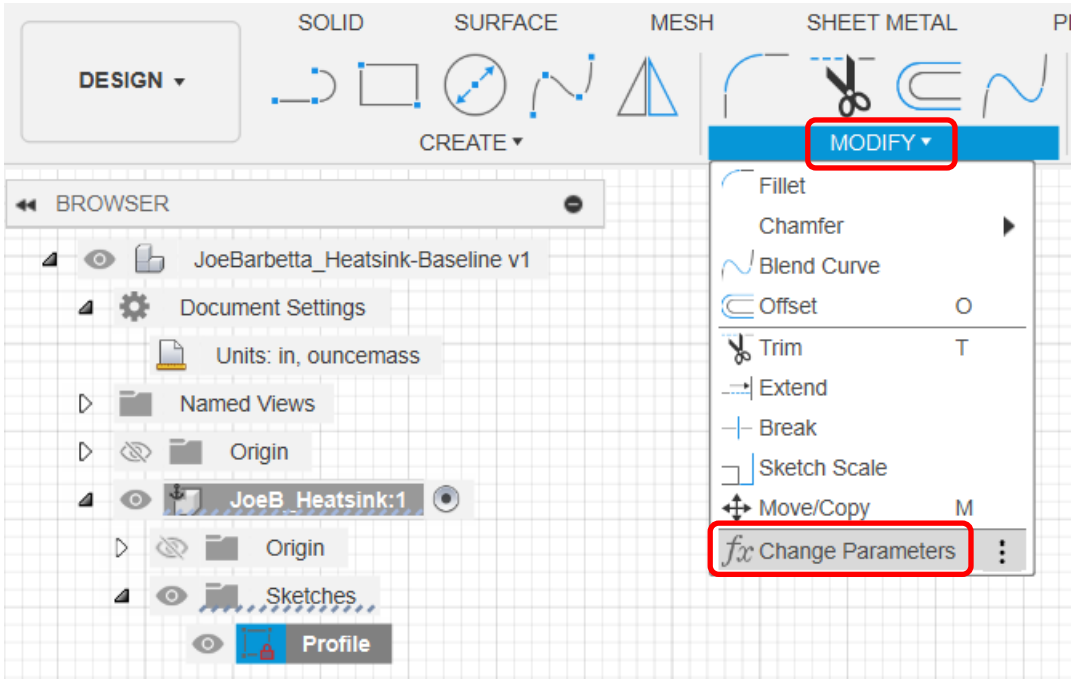


- select **Create Sketch** and click on the **front rhombus**

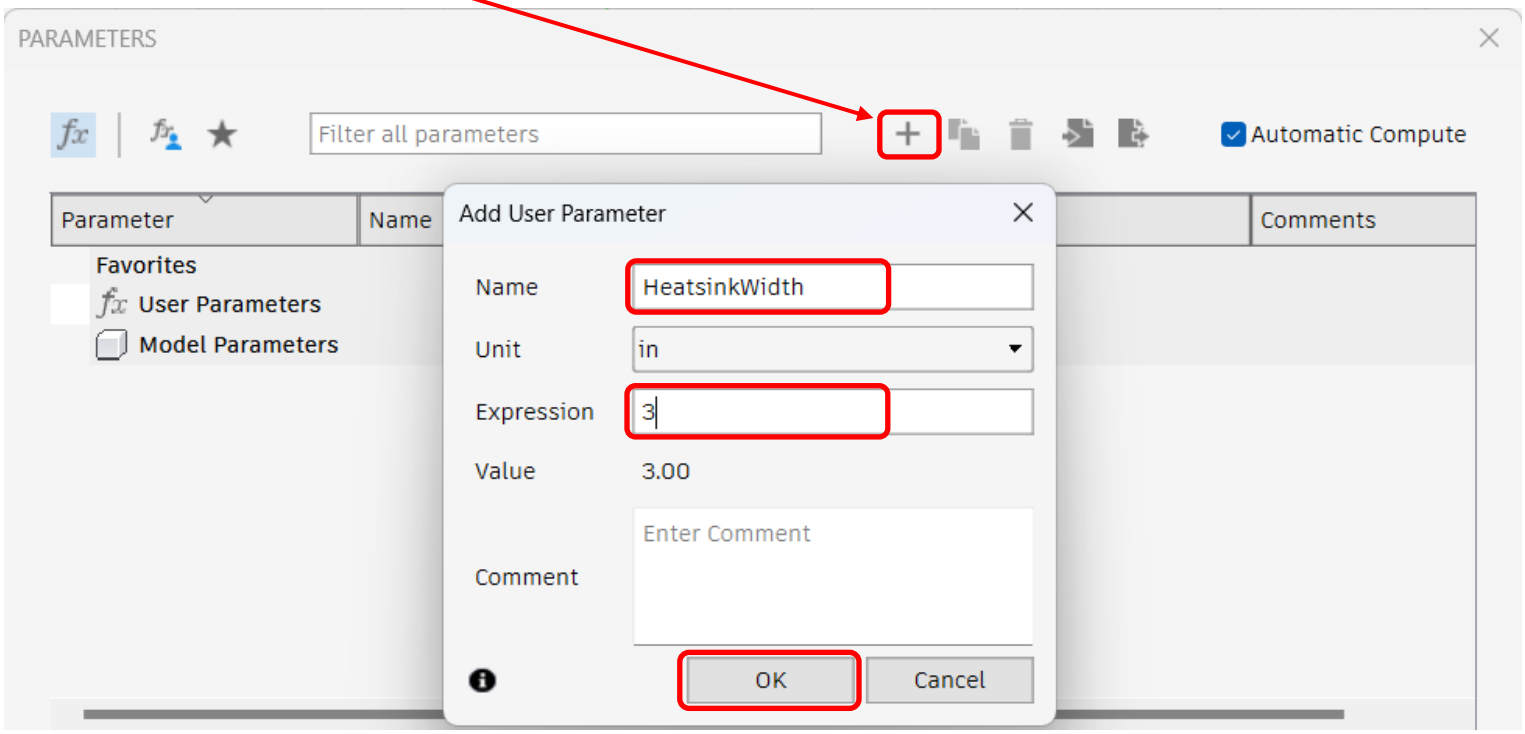


Setting User Parameters

- from the **MODIFY** menu, select **Change Parameters**



- if a window appears about **Parametric Text**, click its **OK** button
- click the **+** icon
- for **Name** enter **HeatsinkWidth**, for **Expression** enter **3**, and click **OK**

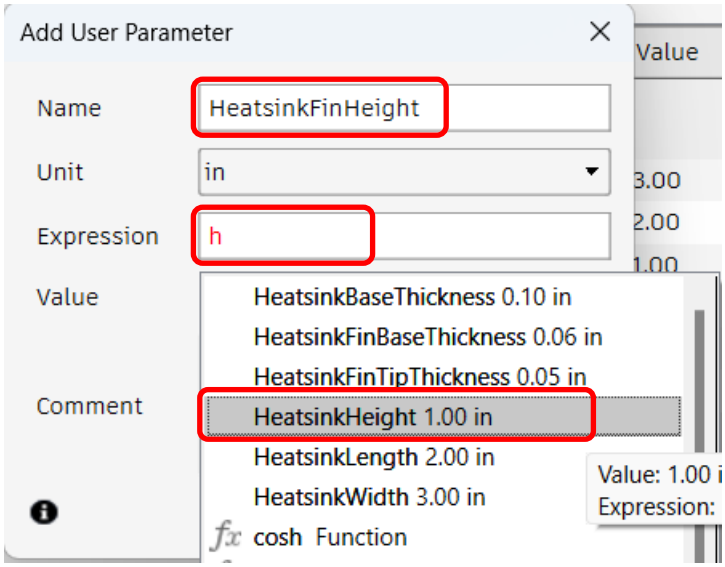


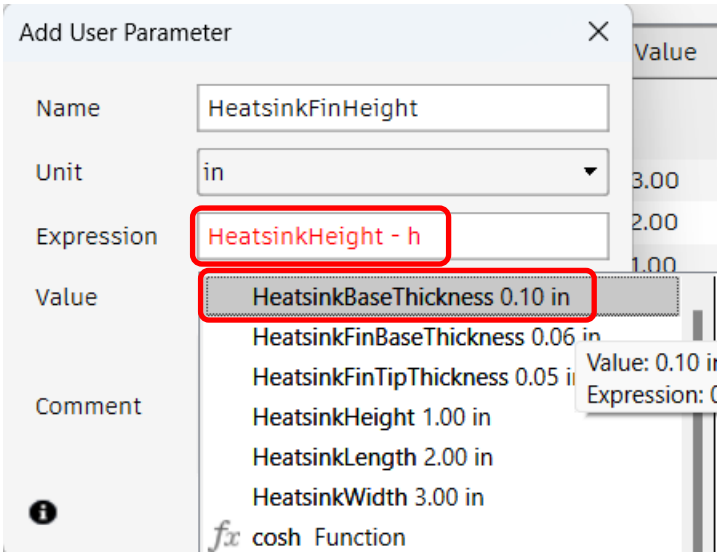
- use the **+** icon again to add the following **Names** and **Expressions**

HeatsinkLength 2
HeatsinkHeight 1
HeatsinkBaseThickness 0.1
HeatsinkFinBaseThickness 0.06
HeatsinkFinTopThickness 0.05

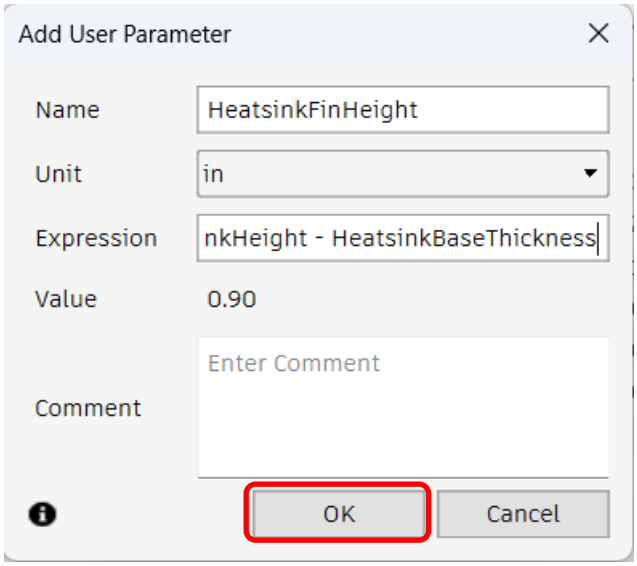
- use the **+** icon again to add a parameter name of **HeatsinkFinHeight**. For its Expression follow these steps.
- type **h** and select **HeatsinkHeight**
- as shown on the right side picture, type **- h** (note the minus sign) and then select **HeatsinkBaseThickness**

Note that Fusion presents functions, such as cosh (hyperbolic cosine), which can also be part of an Expression. Wouldn't it be neat to use a hyperbolic trigonometric function? That would be slay!

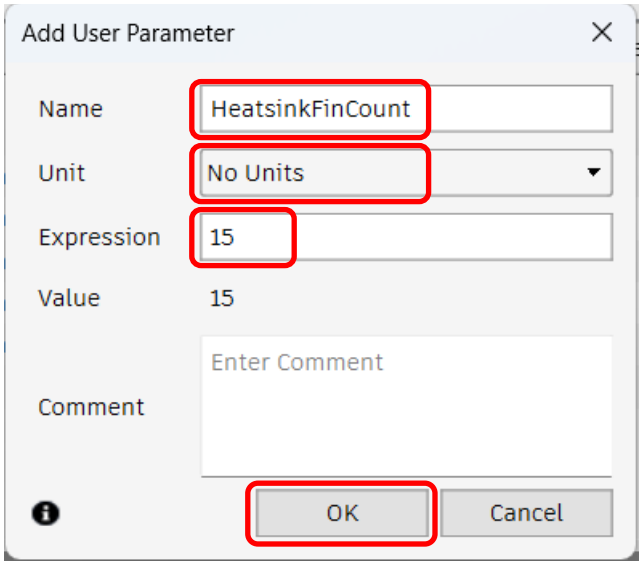




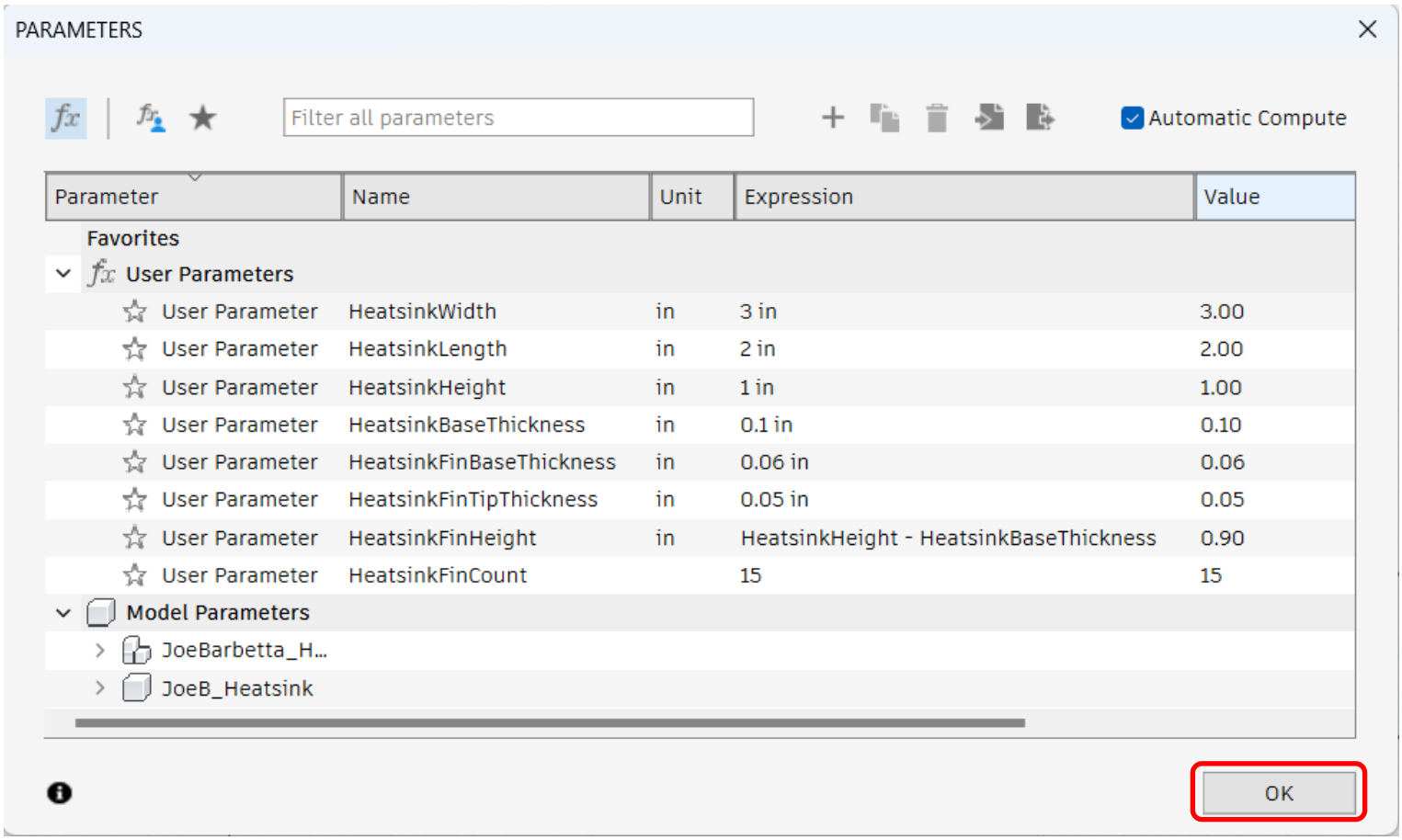
The Expression will look like that below. Note that some of the expression will not fit in the box.
- click **OK**



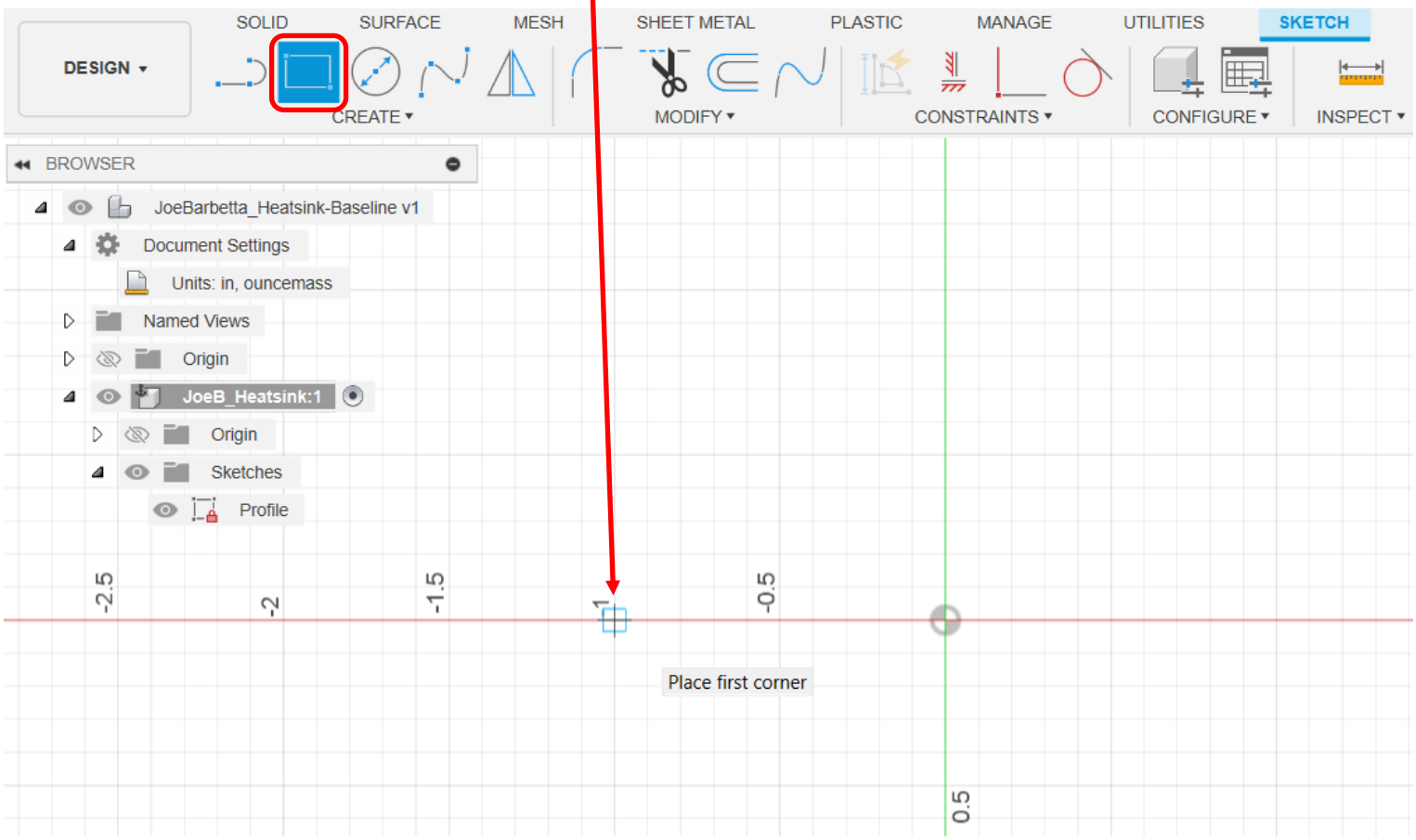
- use the **+** icon again to add one last parameter name of **HeatsinkFinCount** with a value of **15**.
- change the **Unit** to **No Units**
- click **OK**



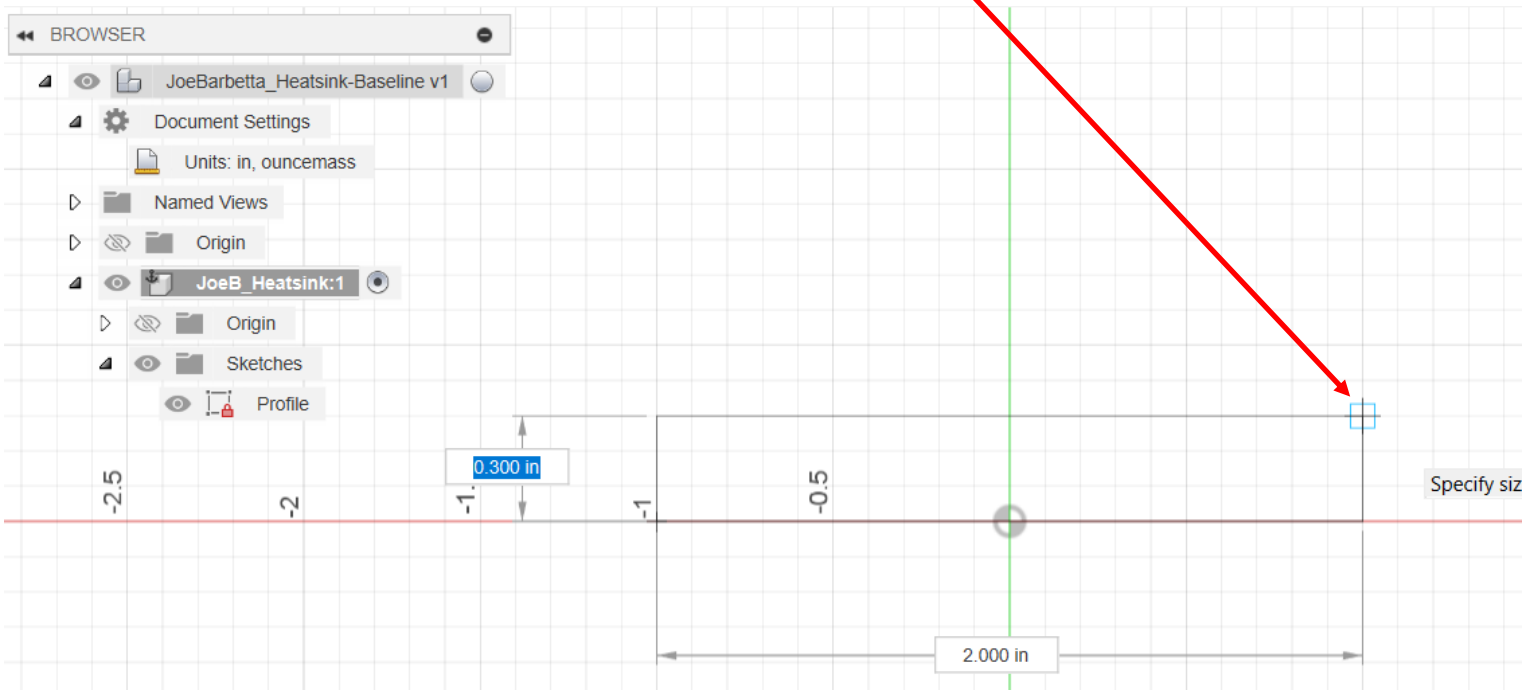
- verify the Parameters. One can click on a Name or Expression to edit a Parameter. One can also select a Parameter and click on the Delete icon to remove it from the list.
- click **OK**



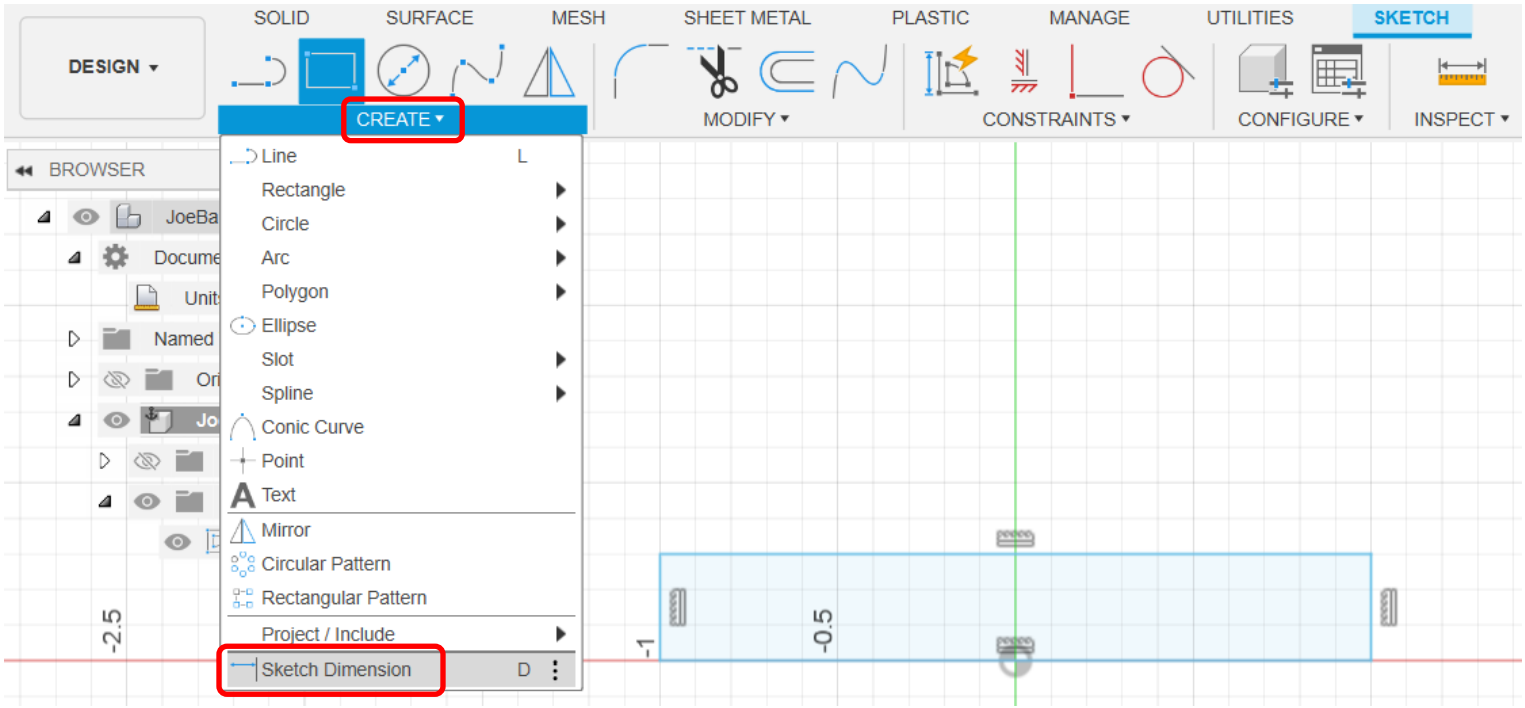
- zoom in to achieve a view similar to that below. Not the values on the axes. It is OK if the values show on the other half of the axes.
- select the **Rectangle** tool and click on the **point** indicated by the arrow. The position is not critical, but it must be on the axis.



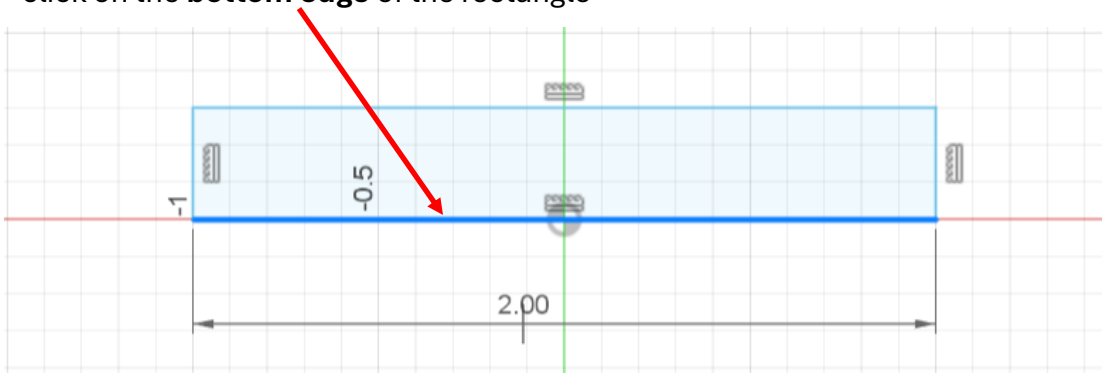
- drag the other corner **up and to the right and** click at **another point** without entering values. Here the rectangle is shown as 2.000 by 0.300, but these values are not important.



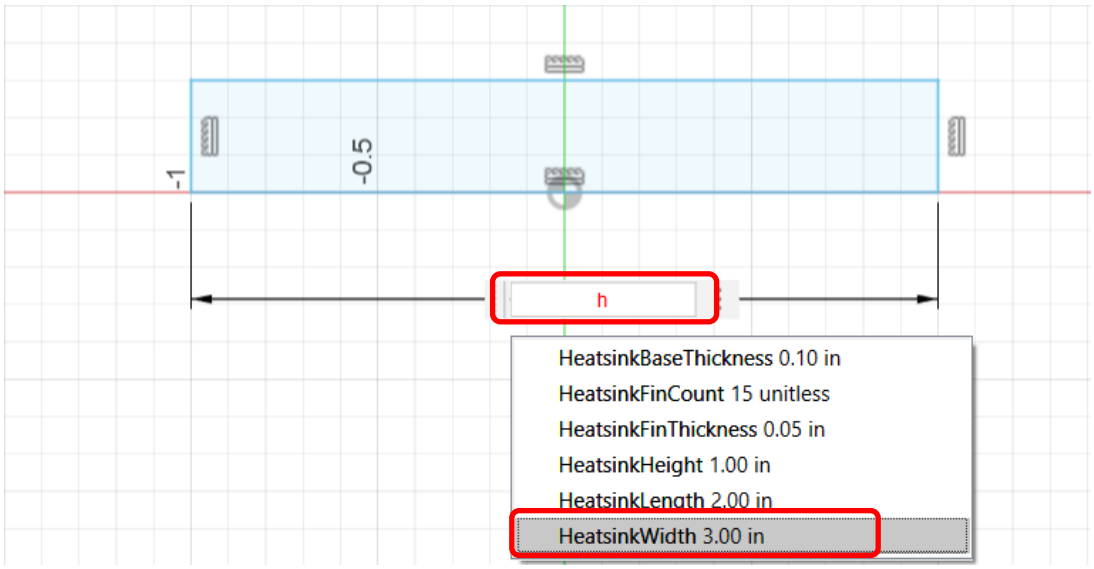
- at the bottom of the **CREATE** menu, select **Sketch Dimension**



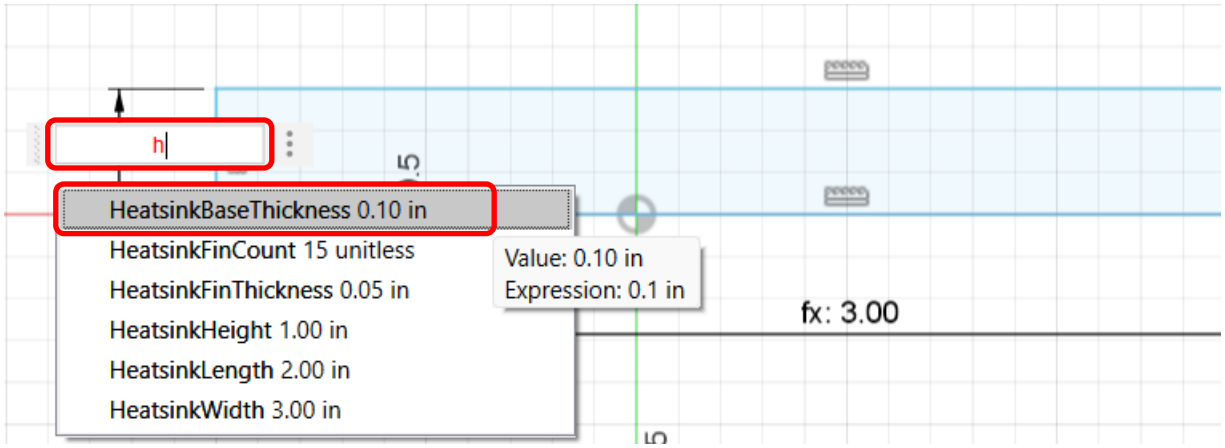
- click on the **bottom edge** of the rectangle



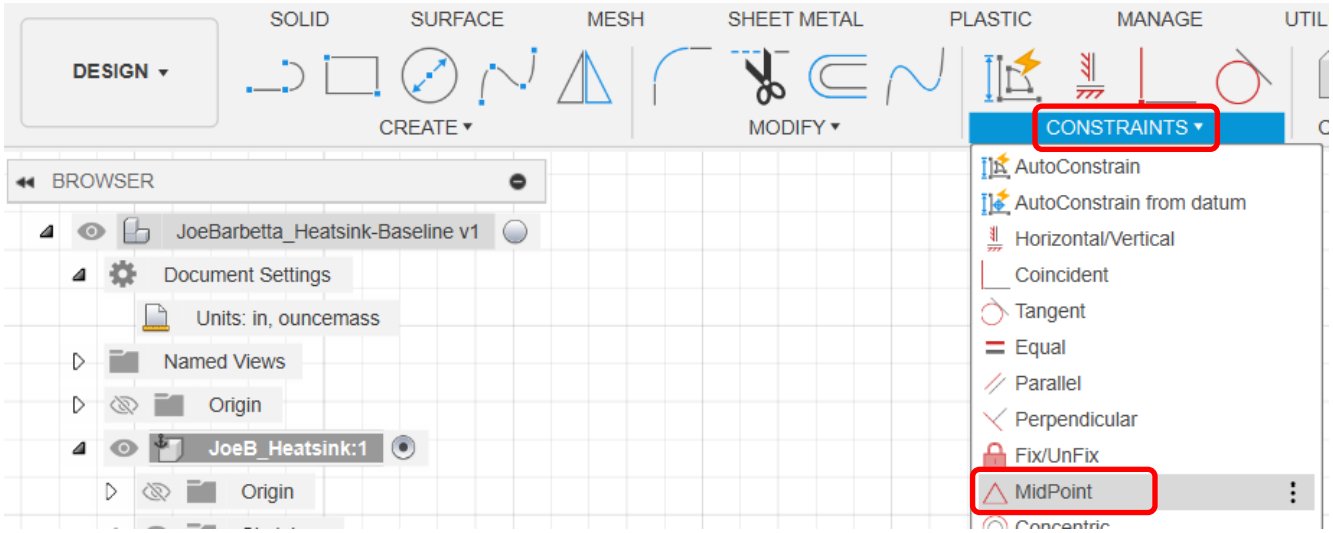
- type **h** in the value box and select **HeatsinkWidth**. Note that the rectangle may not be centered on the origin anymore, which is OK.



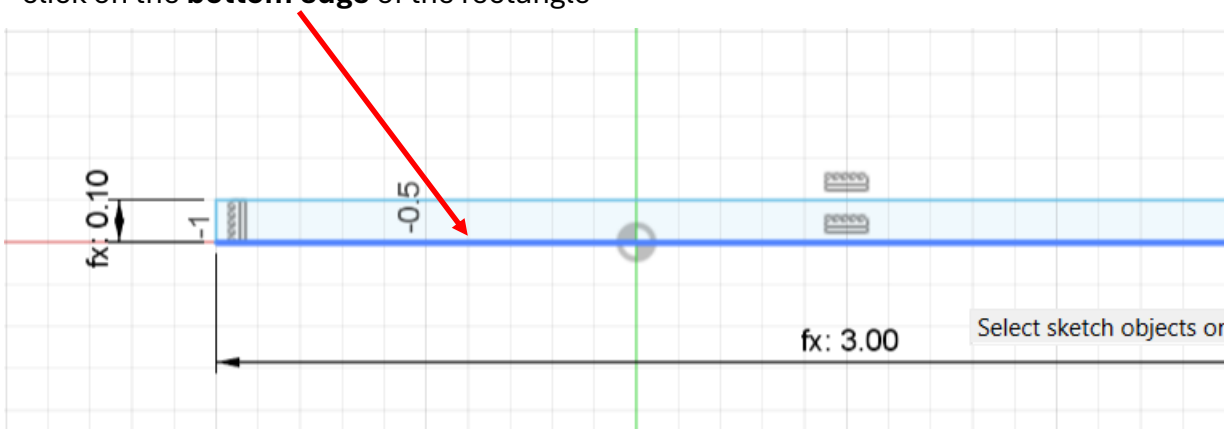
- from the **CREATE** menu, select **Sketch Dimension** again
- click on the **left side** of the rectangle
- type **h** in the value box and select **HeatsinkBaseThickness**



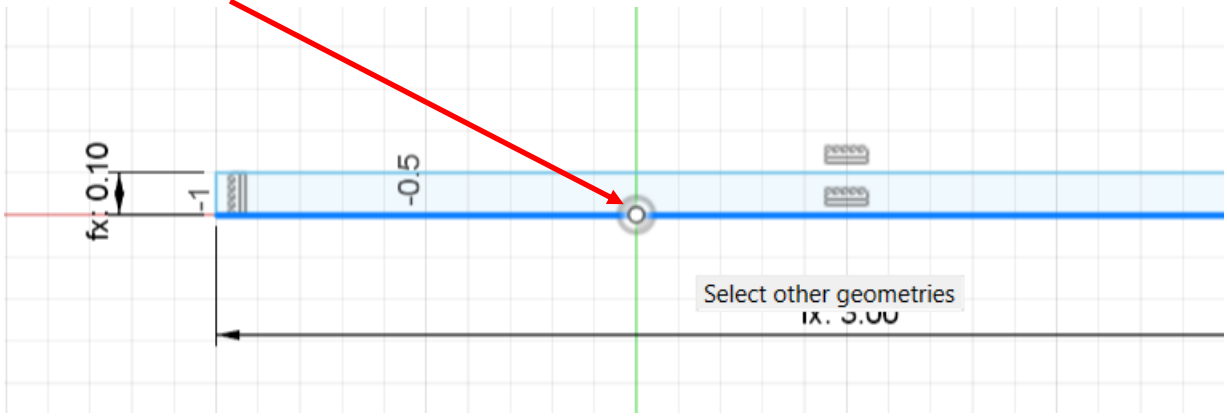
- from the **CONSTRAINTS** menu, select **MidPoint**



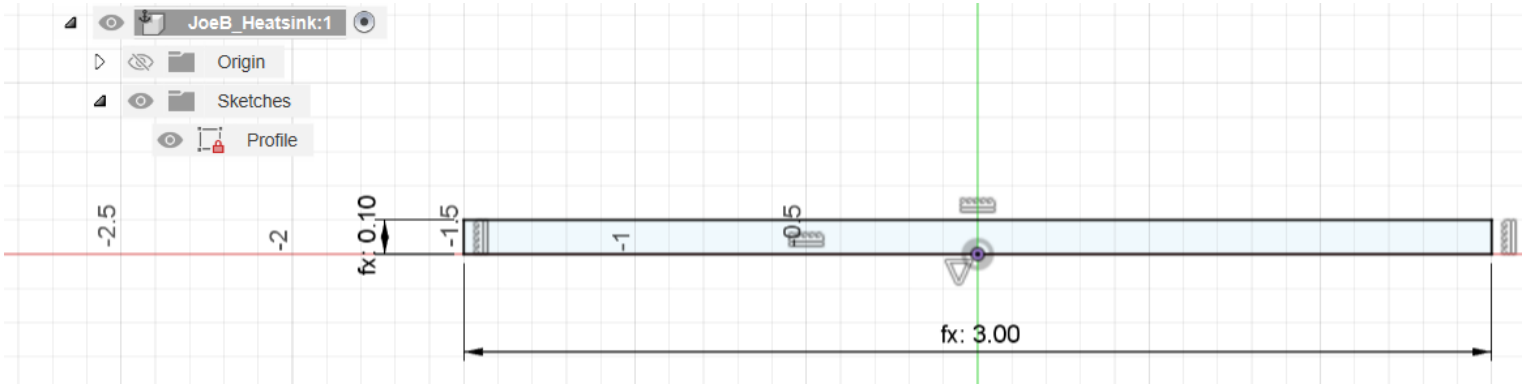
- click on the **bottom edge** of the rectangle



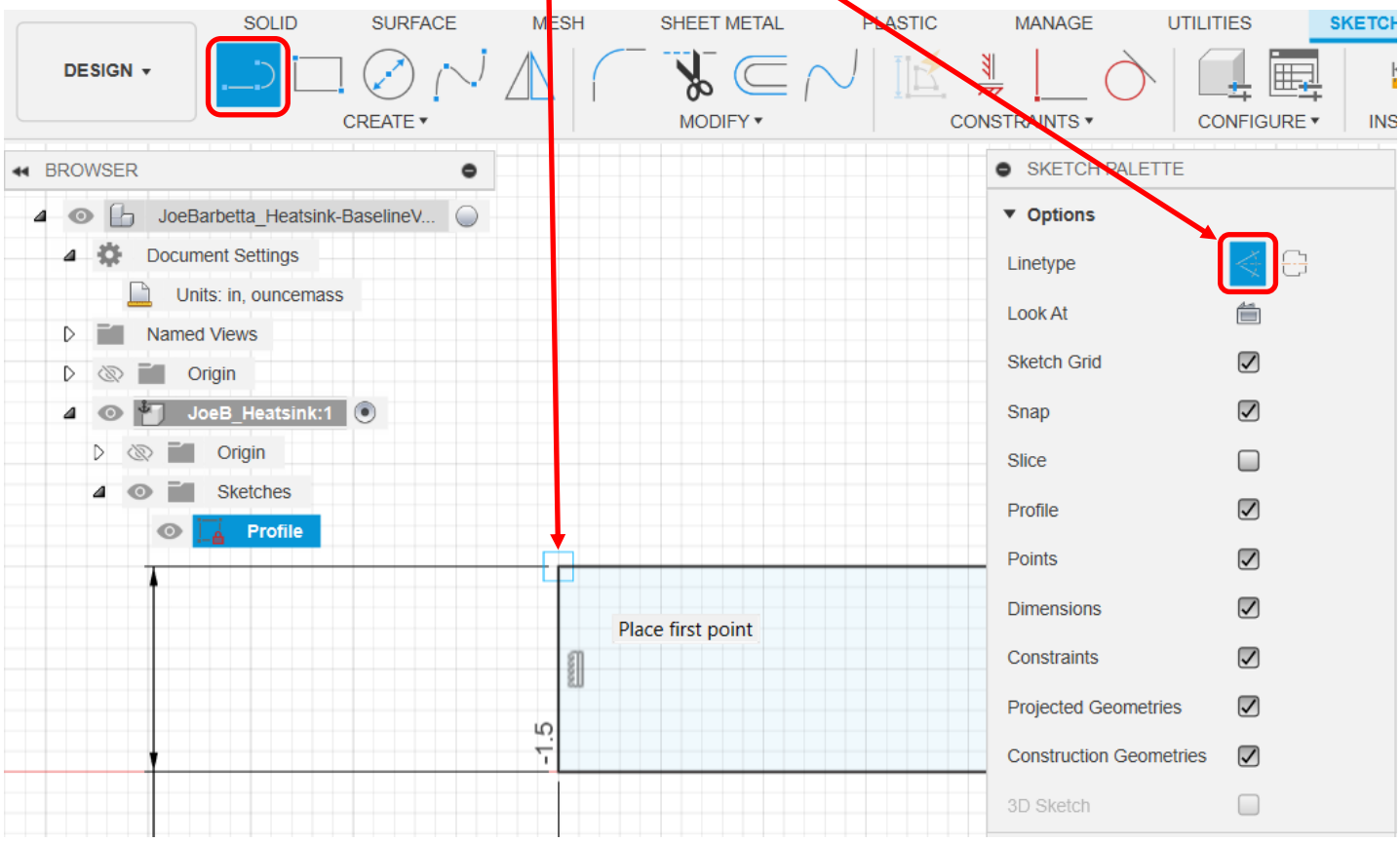
- click on the **Origin**. The rectangle should jump to the left or right to be centered horizontally on the Origin.



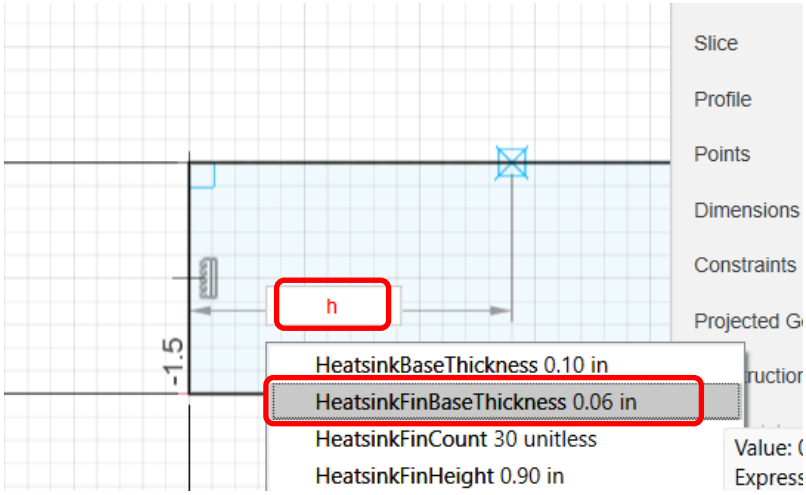
- the rectangle should now be **centered horizontally** on the **Origin**.



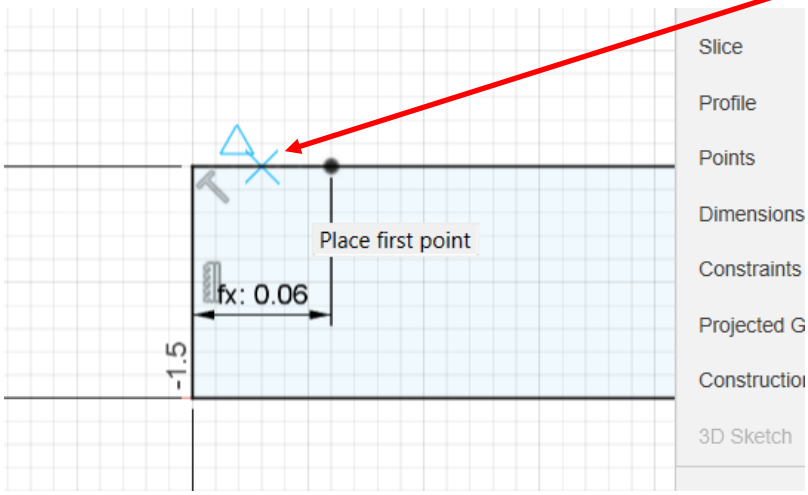
- zoom into the **left end** of the rectangle
- click on the **Construction** line icon to highlight it blue
- select the **Line** tool and click on the **top left corner** of the rectangle



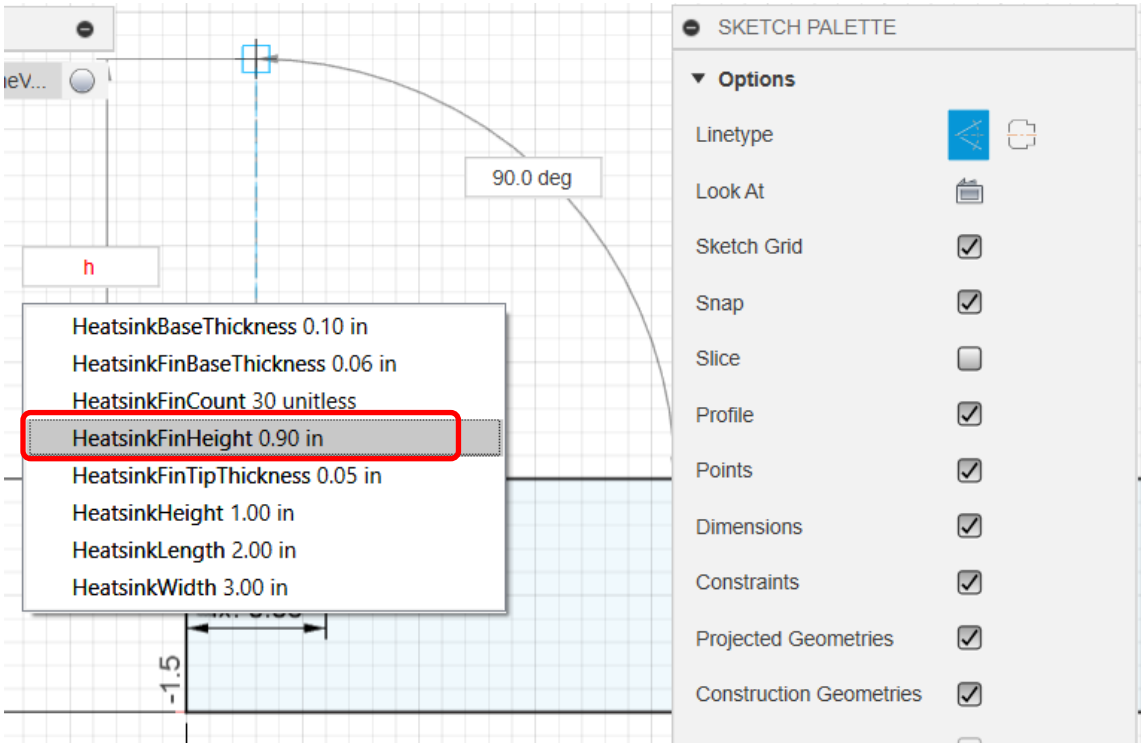
- **extend the line to the right**, type **h**, and select **HeatsinkFinBaseThickness**, and press the **Enter key**



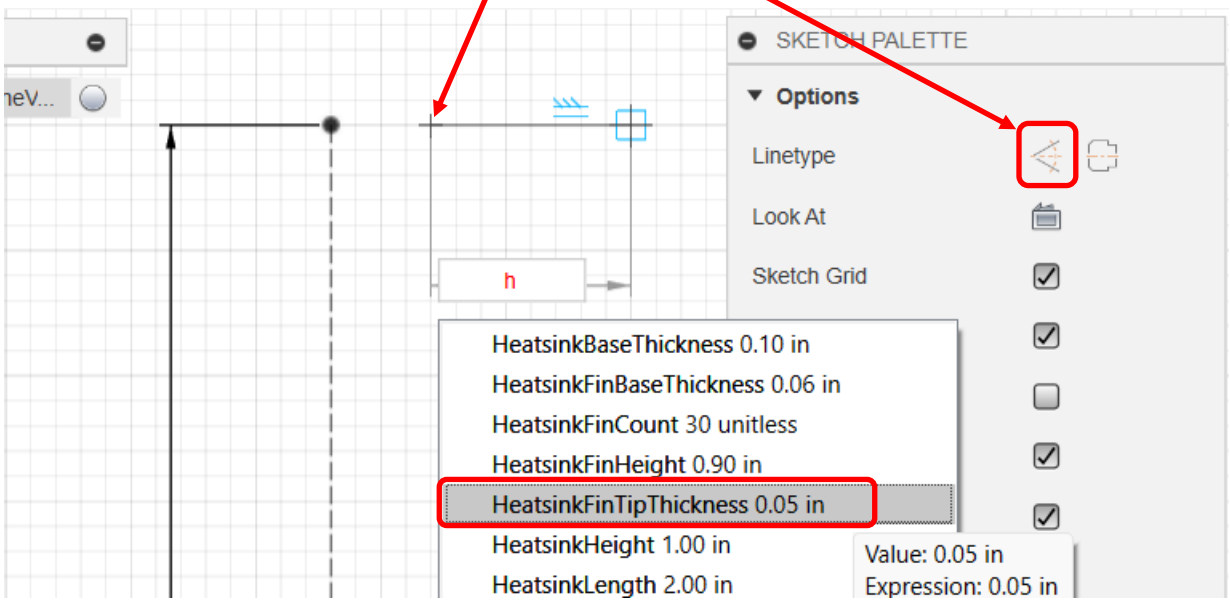
- select the **Line** tool again
- move over the line just created and **click when the blue triangle appears**, which indicates the center point



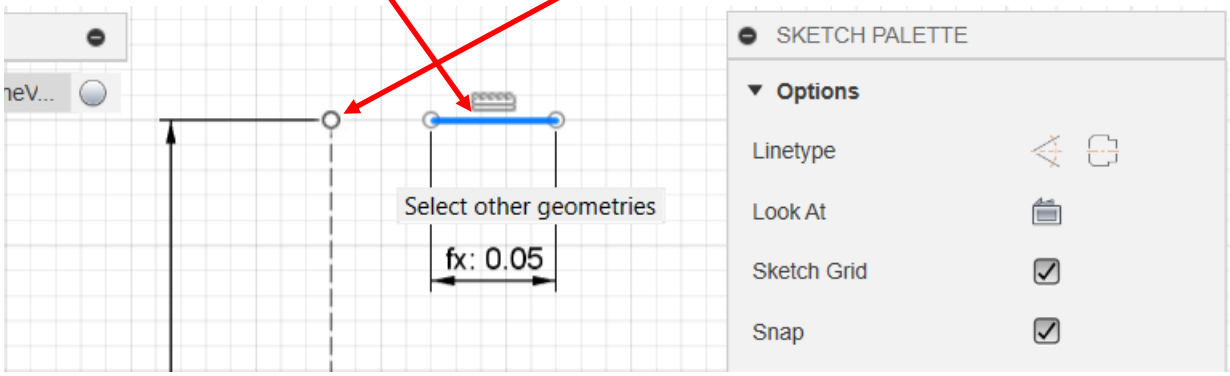
- **extend the line upward**, type **h**, use the **arrow key** to select **HeatsinkFinHeight**, and press the **Enter key**



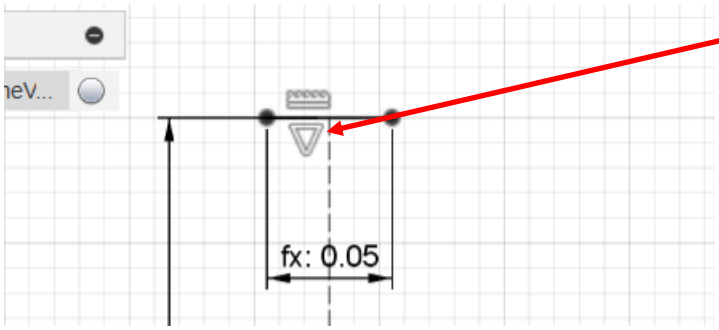
- click on the **Construction** line icon to remove the blue highlighting
- hold down the mousewheel button (or select the Hand icon at the bottom of the Fusion screen) and **pan the view to the top of the line just created**
- select the **Line** tool and click on a **point to the right of the line endpoint**
- **extend the line to the right**, type **h**, use the **arrow key** to select **HeatsinkFinTipThickness**, and press the **Enter key**

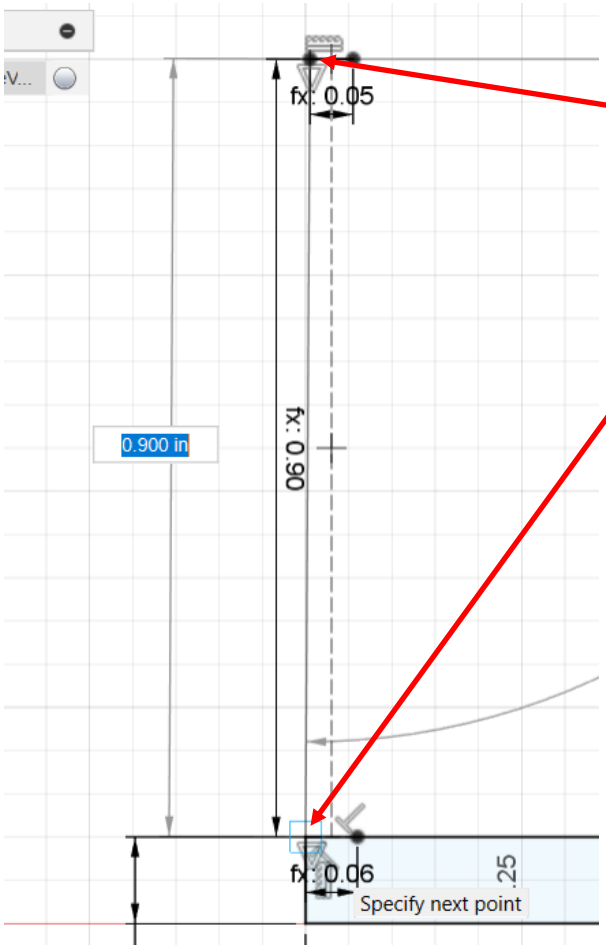


- from the **CONSTRAINTS** menu select **MidPoint**
- click on the **line just created** and then click on the **circle at the top of the vertical line**. This should cause the line to jump to become centered horizontally on the vertical line, as the next picture shows.
- press the **Esc key**



This is the result of the **MidPoint** operation. Note the MidPoint (inverted triangle) symbol.

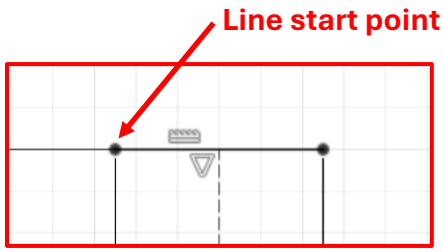




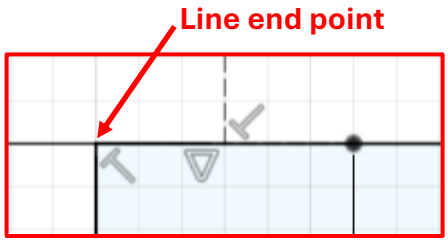
- zoom out to access the lines just created

- select the **Line** tool and click on the **left end of the top horizontal line segment**

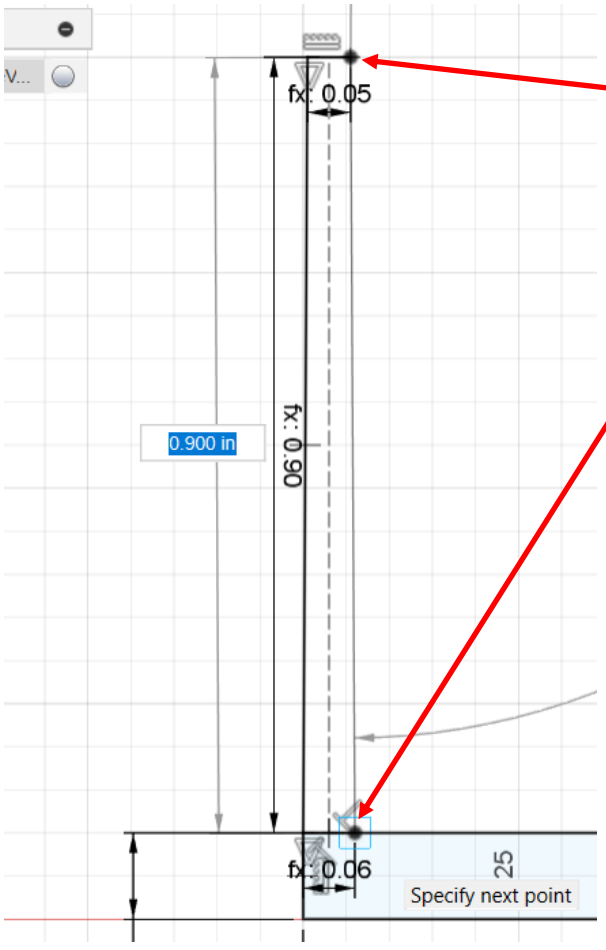
- extend the line downward and click on the **left end of the bottom horizontal line segment**. No value will be entered.



Zoomed in view of top

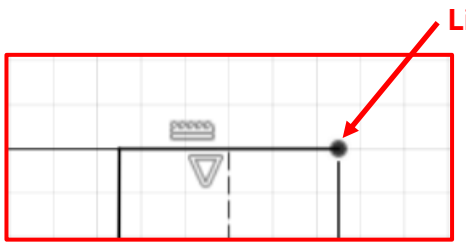


Zoomed in view of bottom

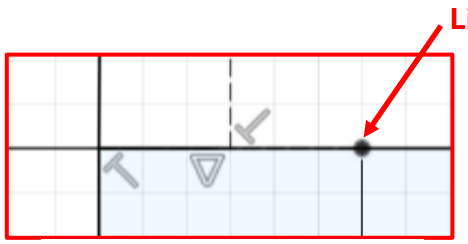


- select the **Line** tool again and click on the **right end of the top horizontal line segment**

- extend the line downward and click on the **right end of the bottom horizontal line segment**. No value will be entered.

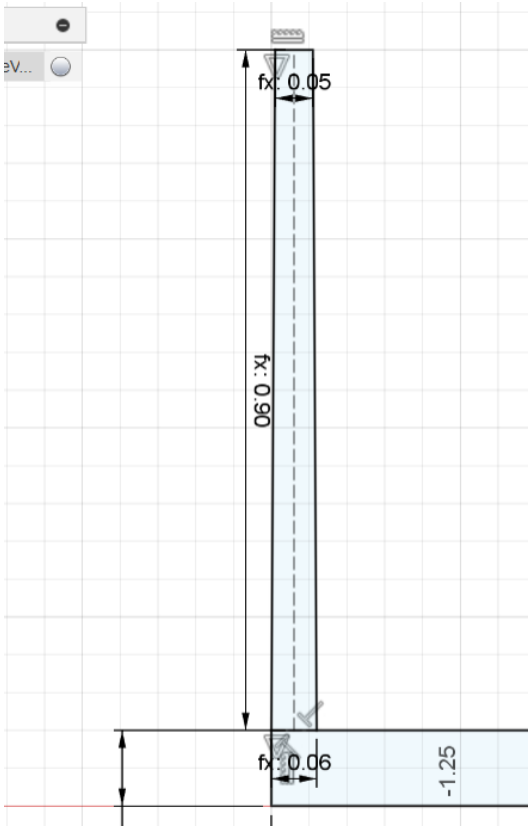


Zoomed in view of top

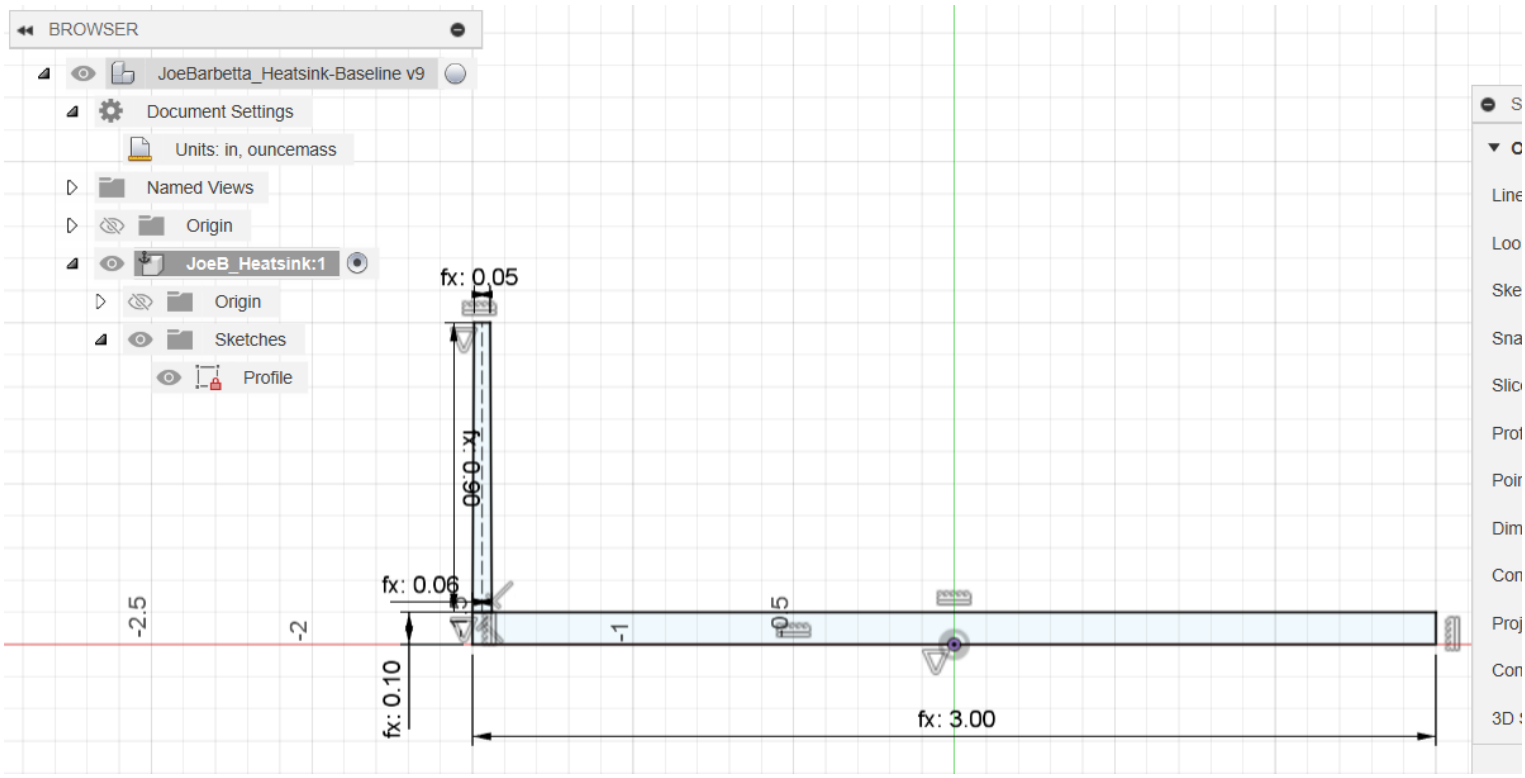


Zoomed in view of bottom

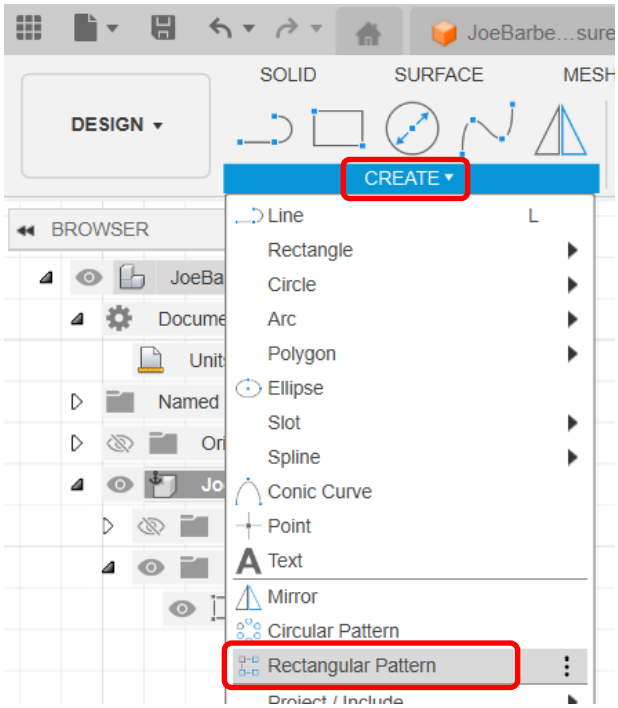
This is the result of the recent line operations. This is a profile for a fin. It should be automatically filled in with a light blue color.



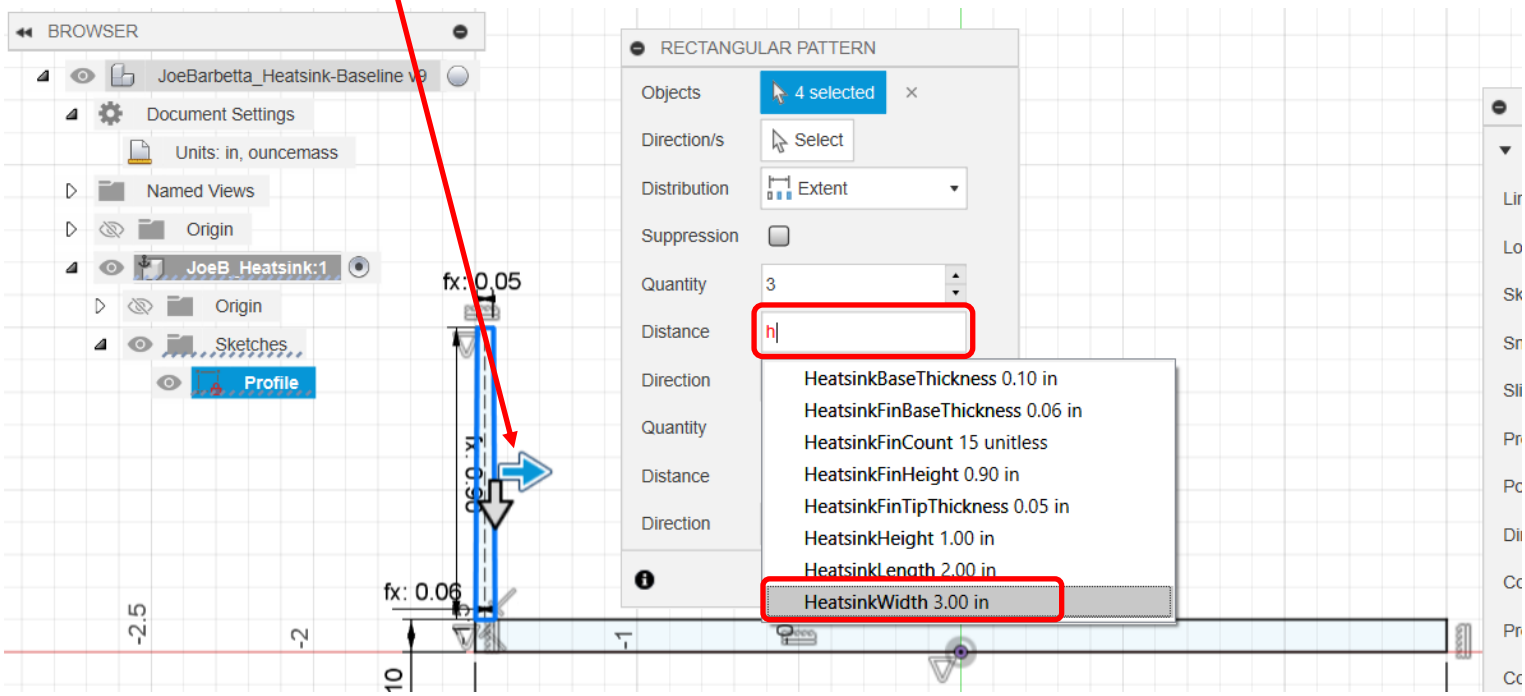
- zoom out to view the entire heat sink base and fin



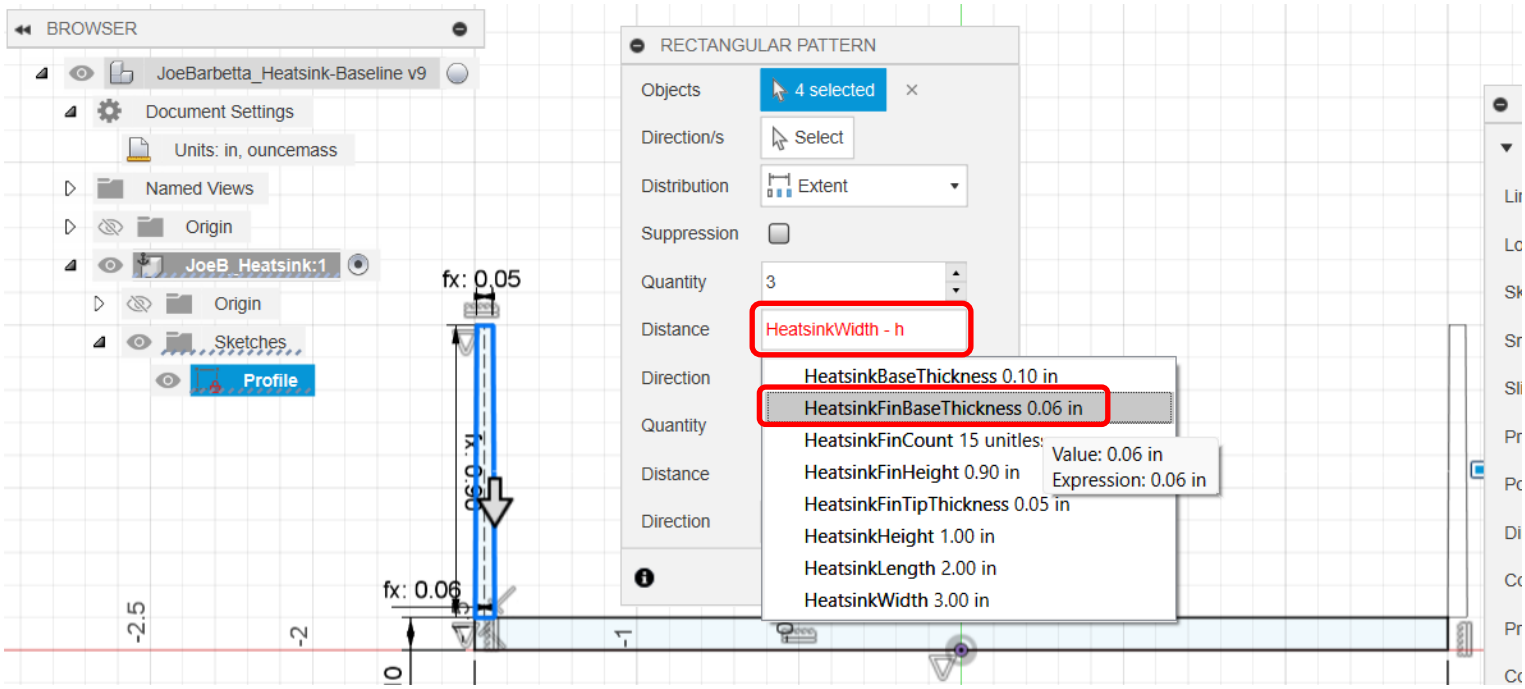
- fromt the **CREATE** menu, select **Rectangular Pattern**



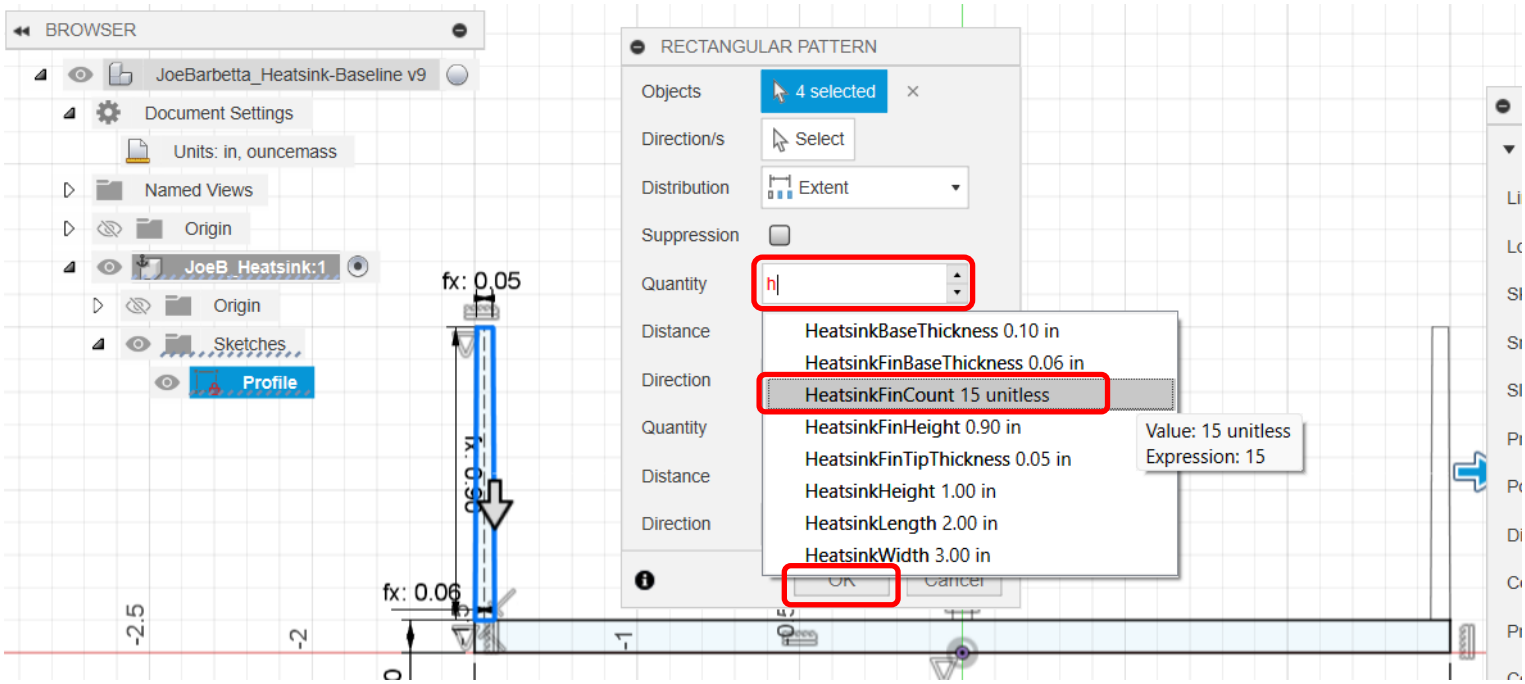
- **double-click** on one of the **solid lines** of the fin to highlight them blue
- click on the **horizontal blue arrow**
- for **Distance** type **h** and select **HeatsinkWidth**
- type a **minus sign** followed by **h** and select **HeatsinkFinBaseThickness**, as shown in the next picture



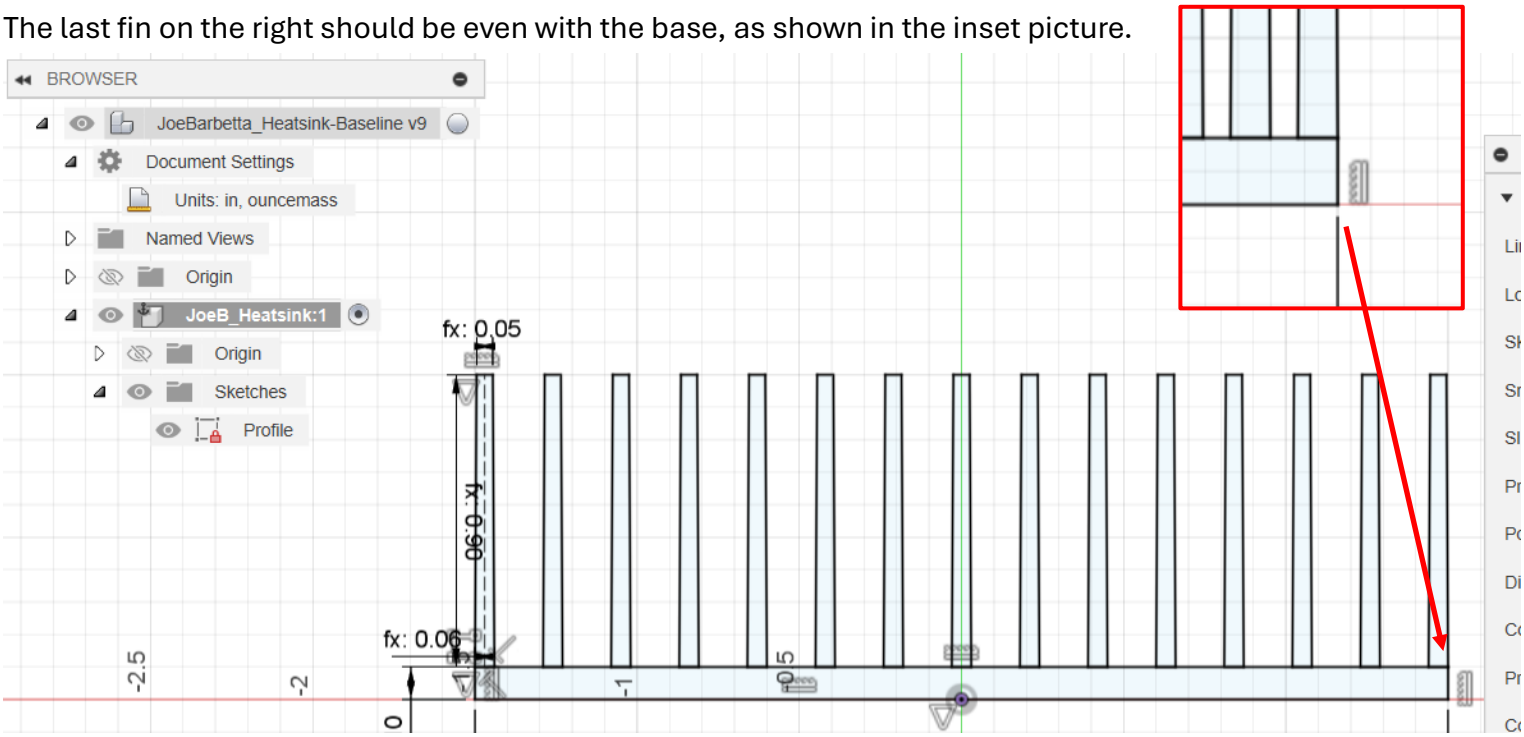
- as mentioned above previous picture



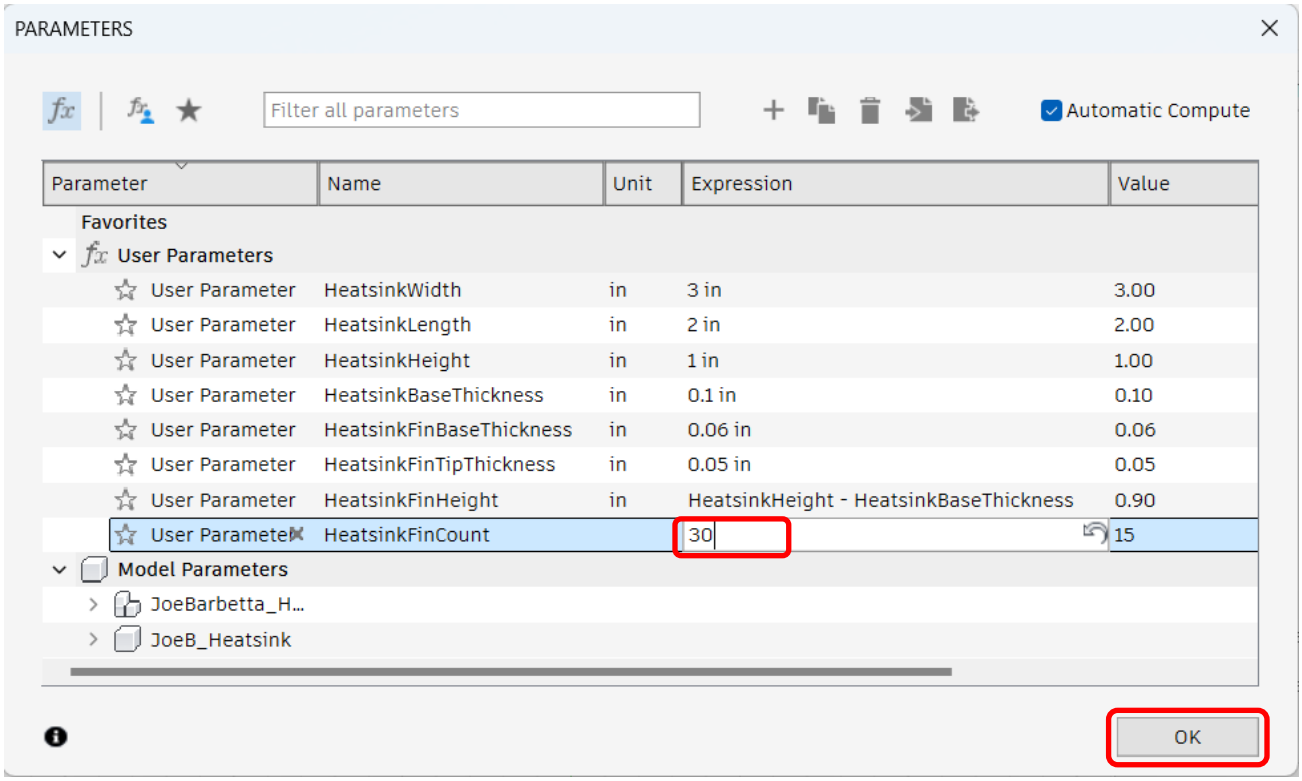
- in the **Quantity** box, type **h**, and select **HeatsinkFinCount**
- click **OK**



This should be the result of the Pattern Operation.
The last fin on the right should be even with the base, as shown in the inset picture.

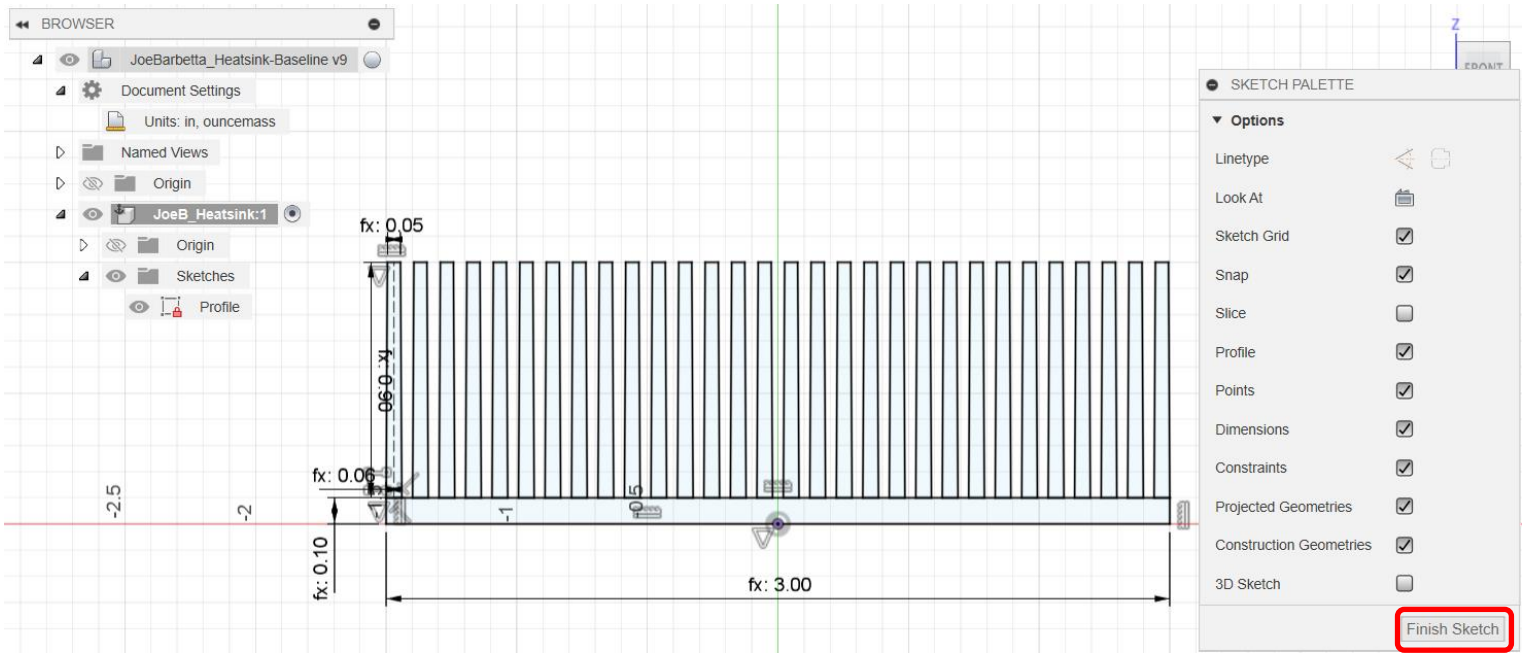


- from the **MODIFY** menu, select **Change Parameters**
- click on the Expression for HeatsinkFinCount and double its value. Here 15 was changed to 30.
- click **OK**

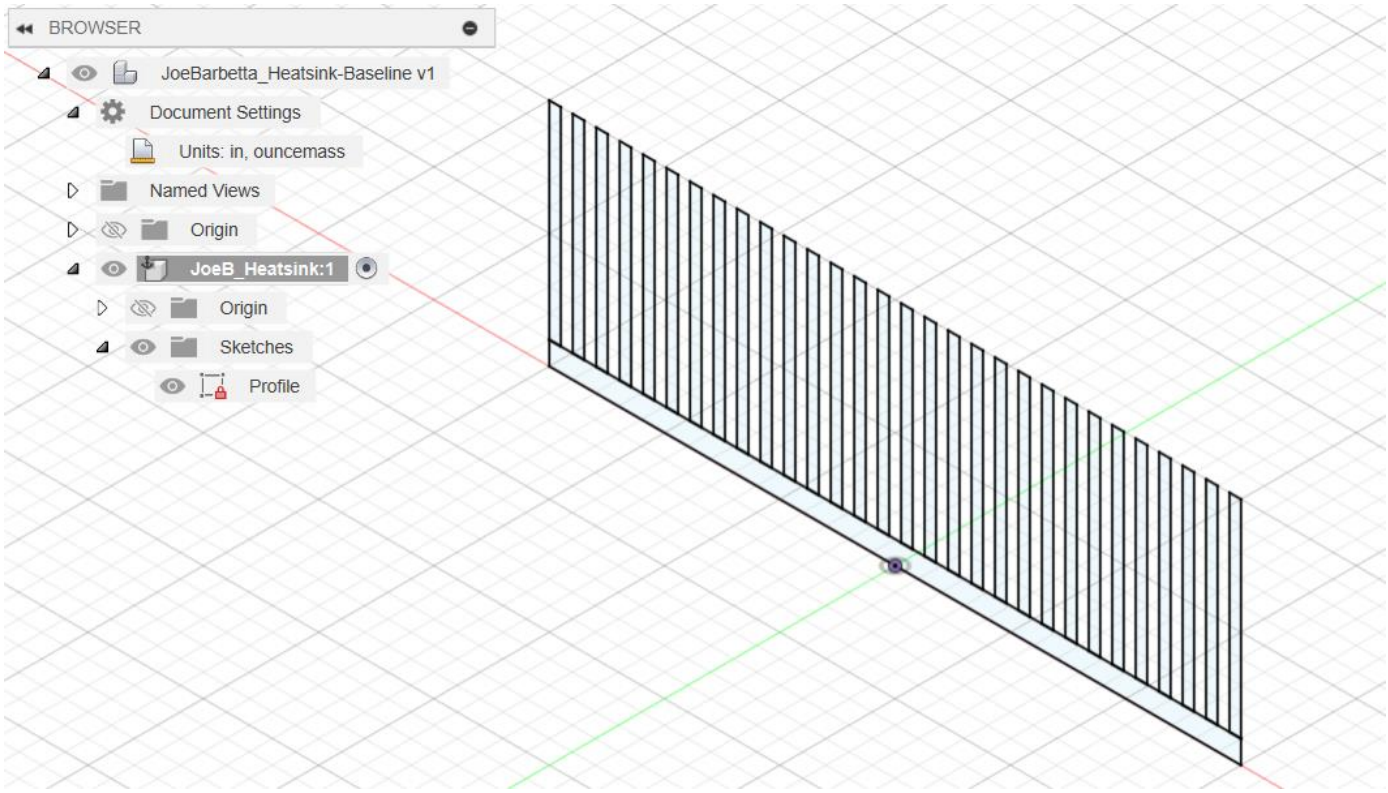


- yell **“Dude, that’s a lot of fins!”**
- click **Finish Sketch**

The reason for doubling the fin count is to allow changes to the HeatsinkFinCount Parameter to affect the number of fins for the final extruded heatsink.

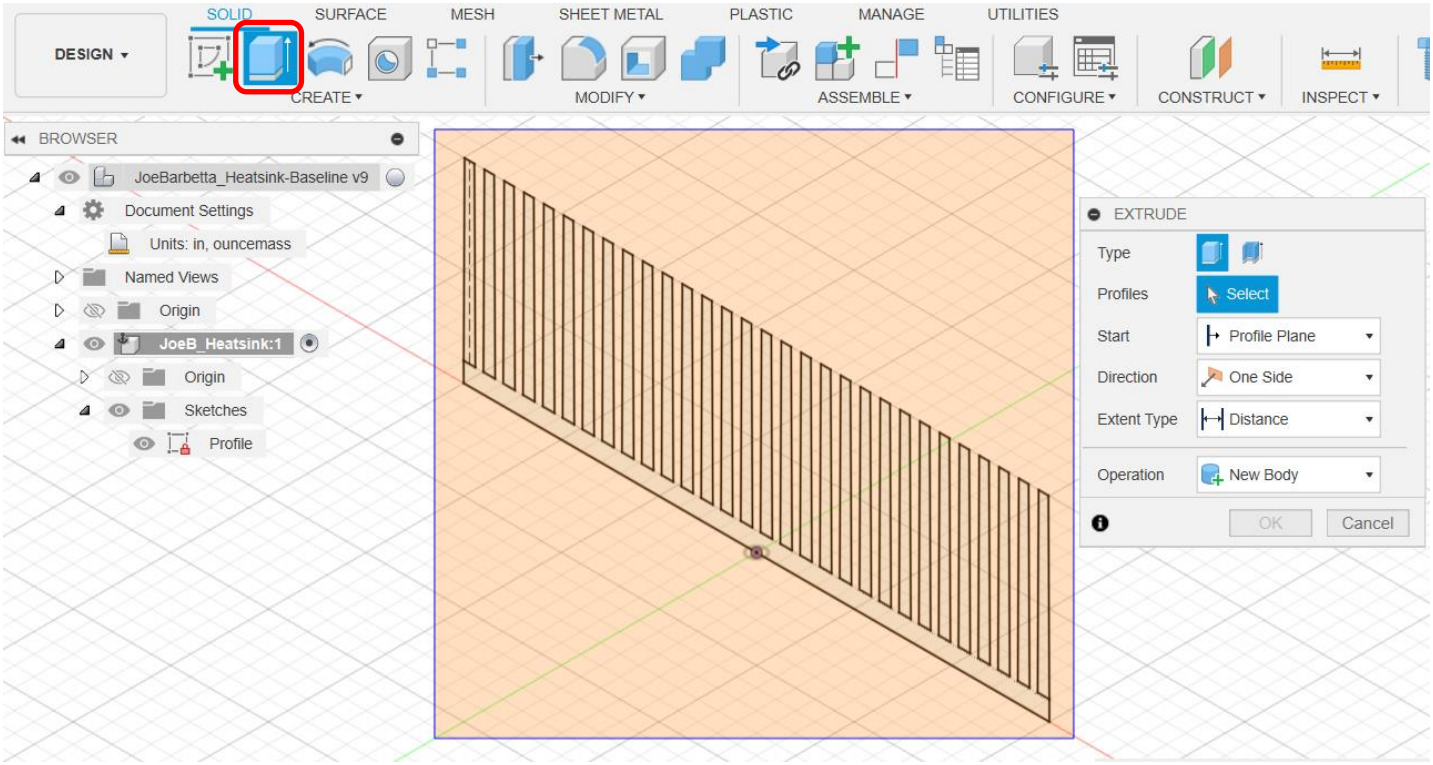


- click on the **Home** icon at the **View Cube**
- zoom to achieve a view similar to this, where the entire profile is visible.

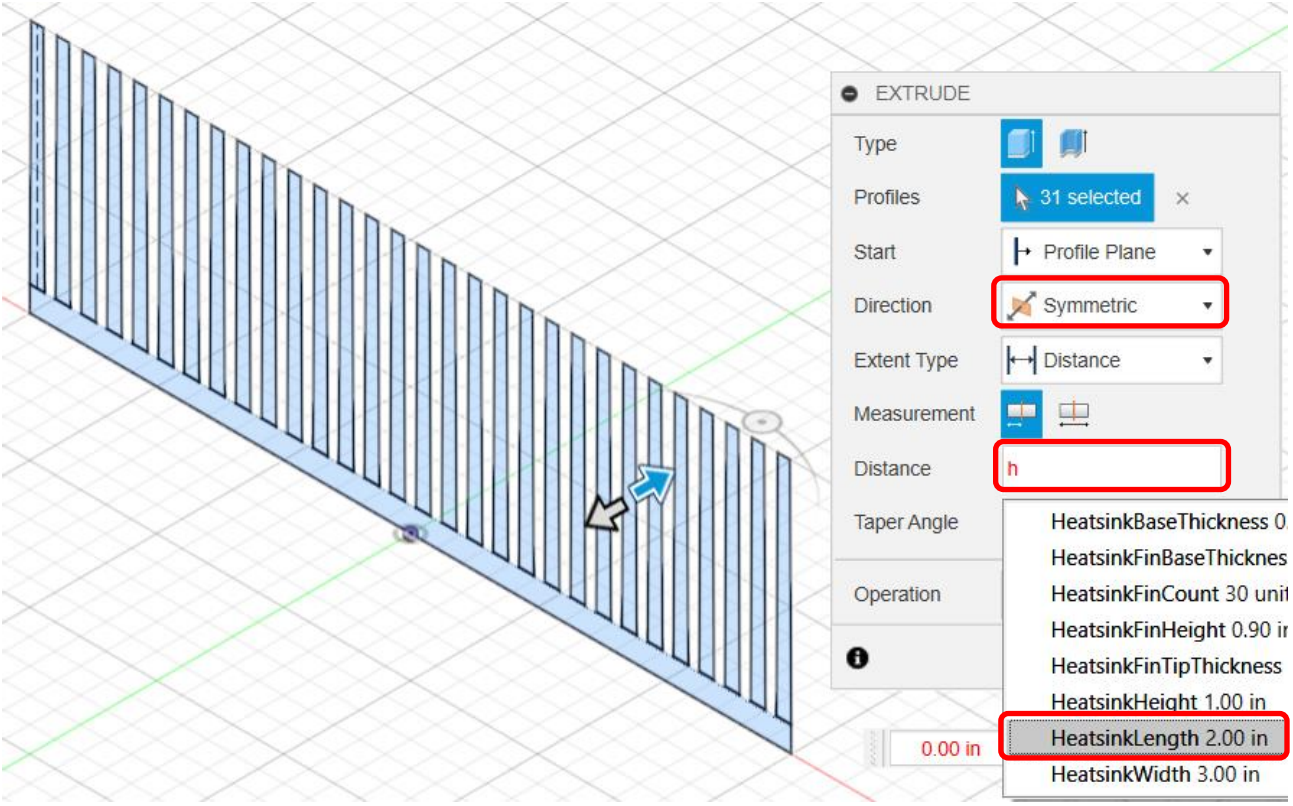


Extruding the Heat Sink

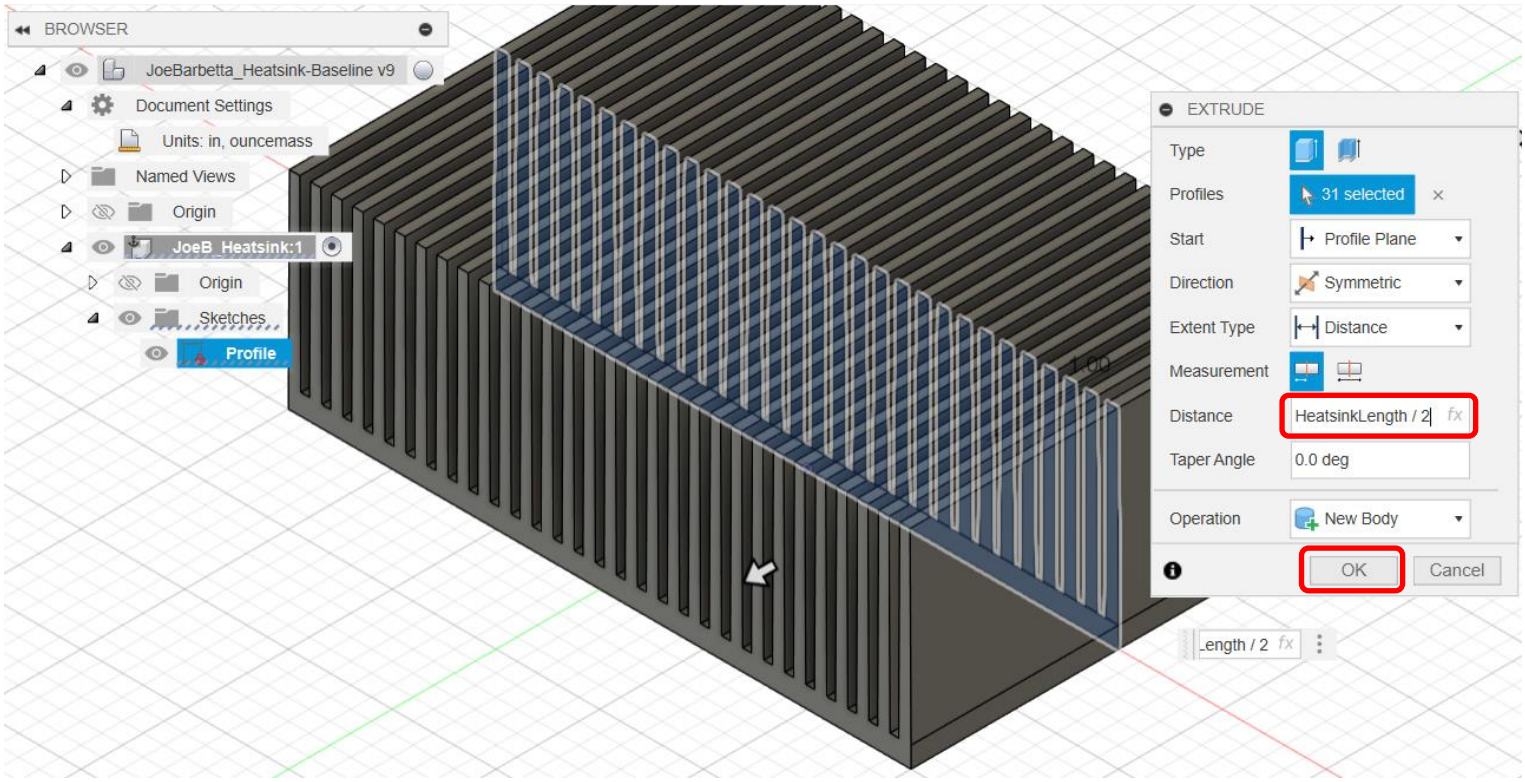
- select the **Extrude** tool
- drag a selection rectangle around the **entire profile**



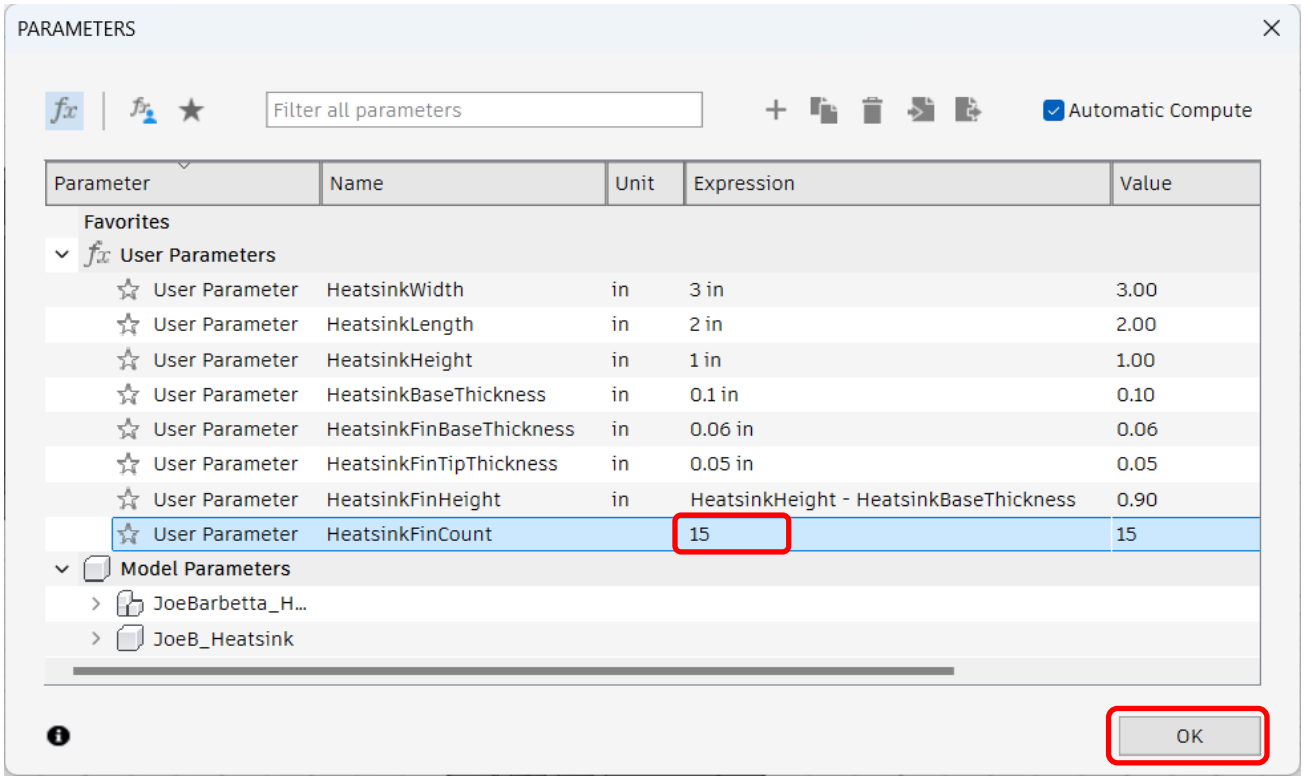
- change **Direction** to **Symmetric**
- in the **Distance** box type **h** and select **HeatsinkLength**



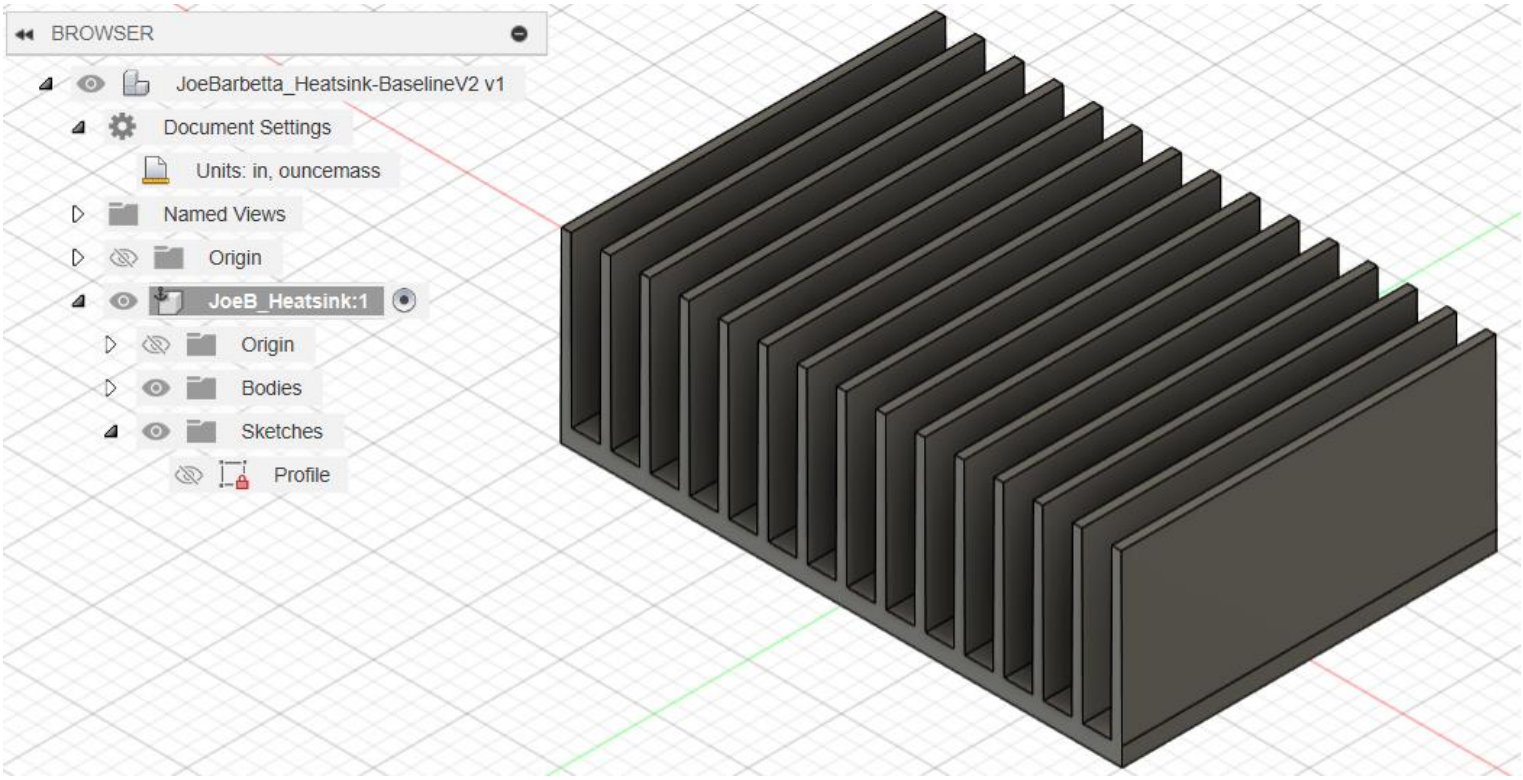
- after HeatsinkLength is selected in the Distance box type **/2** after it to divide the value by 2.
- click **OK**



- from the **MODIFY** menu, select **Change Parameters**
- click on the Expression for HeatsinkFinCount and change the value back to **15**
- click **OK**



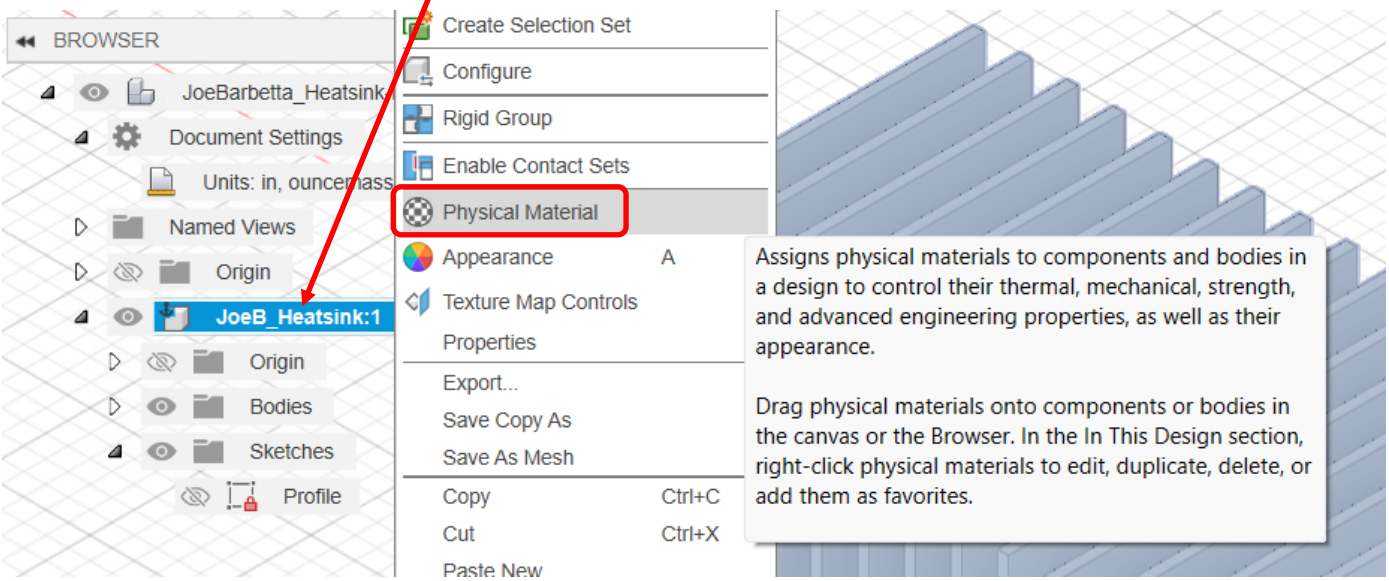
The heat sink should now show with 15 fins.



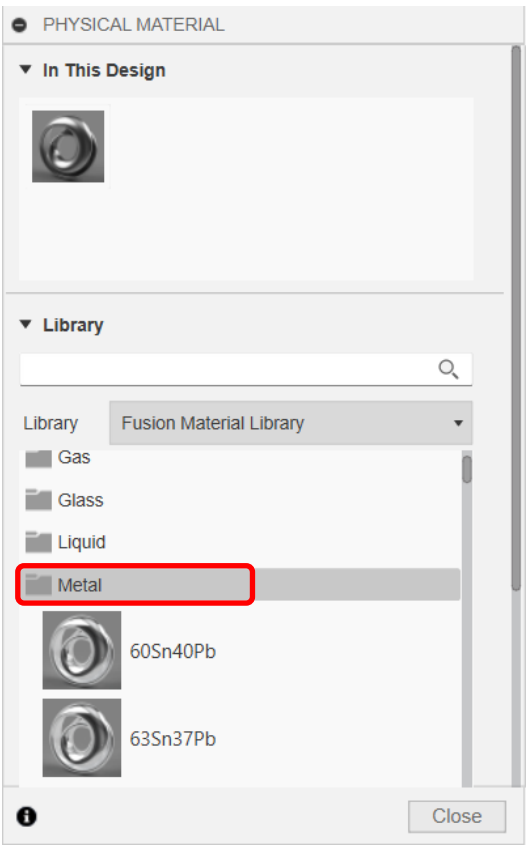
Setting the Material

By default Fusion sets a body to steel. The great majority of heatsinks are made of aluminum and in particular an alloy of aluminum designated as 6063. Note that 6061 is a very common alloy of aluminum. 6063 is similar, but it has higher thermal conductivity and is easier to extrude (not the Fusion extrude, but the extrusion manufacturing method. 6063 is not as strong as 6061, however, for a heatsink strength has a much lower priority compared to its thermal properties.

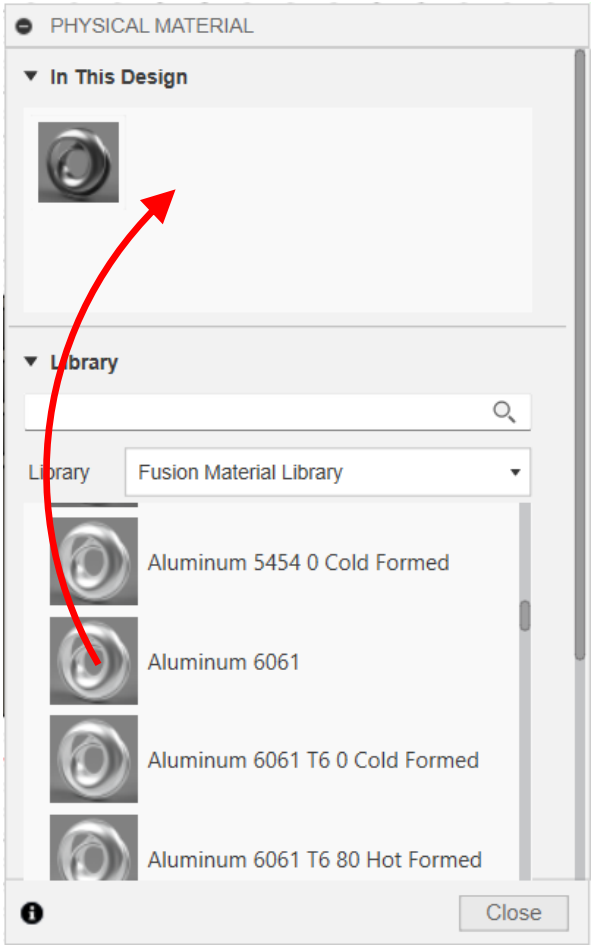
- right-click on the heatsink **Component** name and select **Physical Material**



- scroll down to the **Metal** folder and click on it



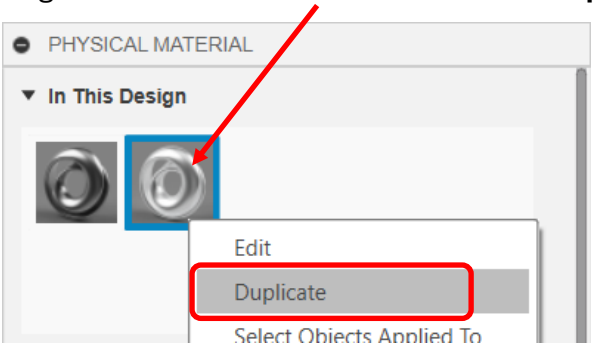
- sroll down to **Aluminum 6061**
- drag its icon into the top **In This Design** section next to the existing icon



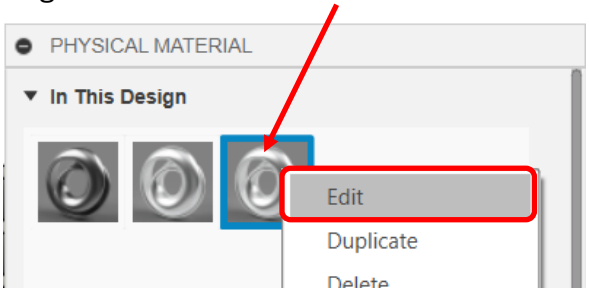
As of this writing, Aluminum 6063 is not in the Material Library.

We will select Aluminum 6061 and then change its name and thermal properties for 6063.

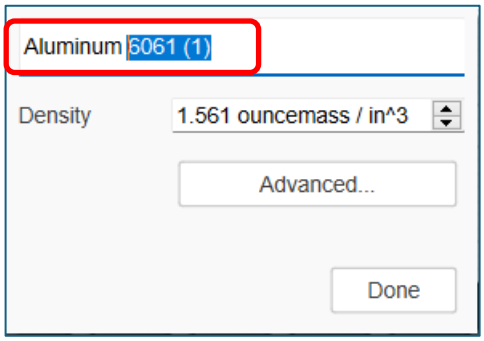
- right-click on the **new icon** and select **Duplicate**

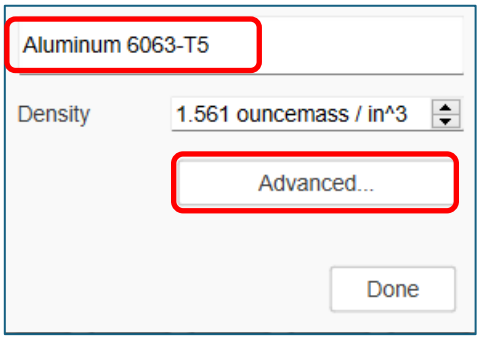


- right-click on the **newest icon** and select **Edit**



- select the **6061 (1)** text and change it to **6063-T5**
- click **Advanced...**



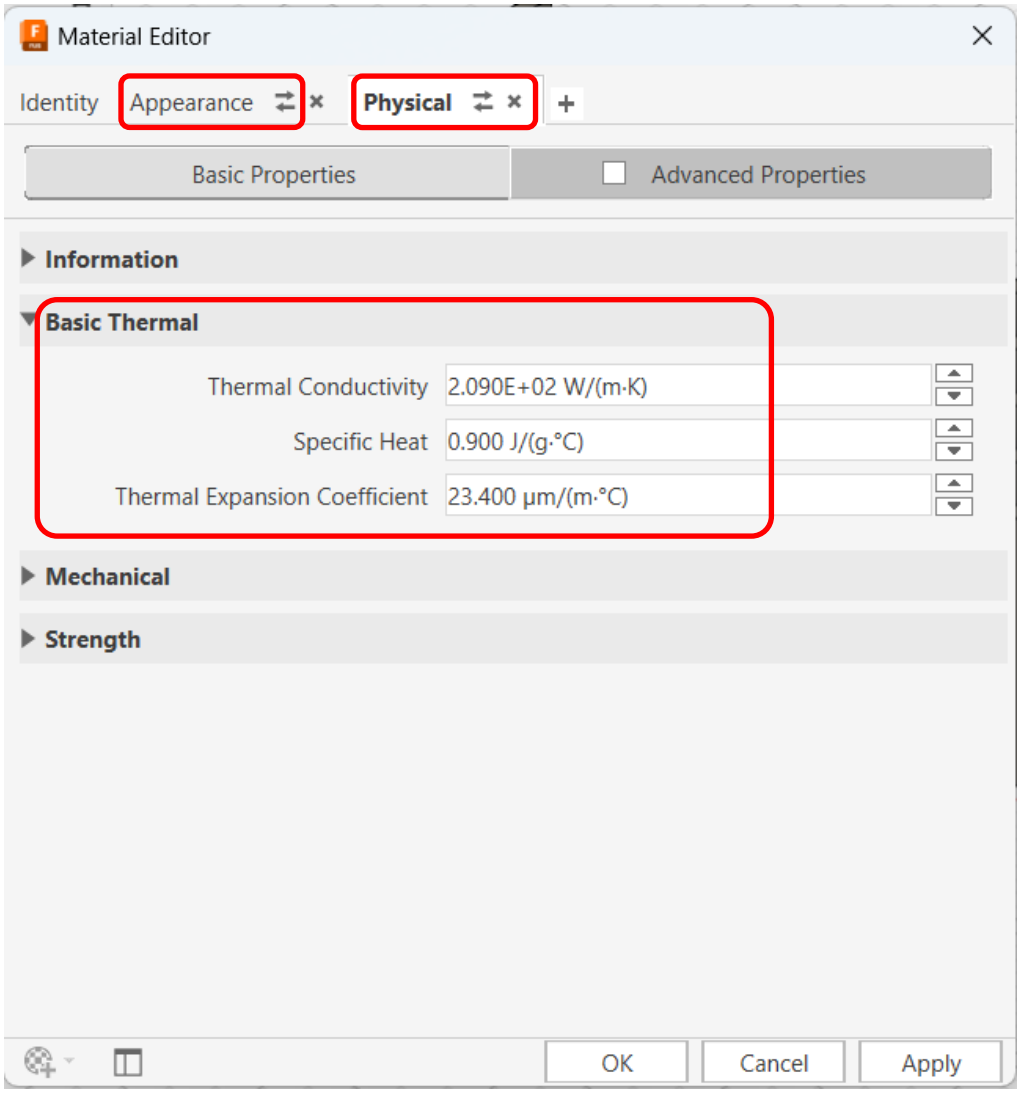


The **-T5** suffix indicates the ***temper*** of the aluminum due to heat treatment. Note that 6061 typically has a **-T6** temper.

- click on the **Physical** tab and change the **Basic Thermal** properties to

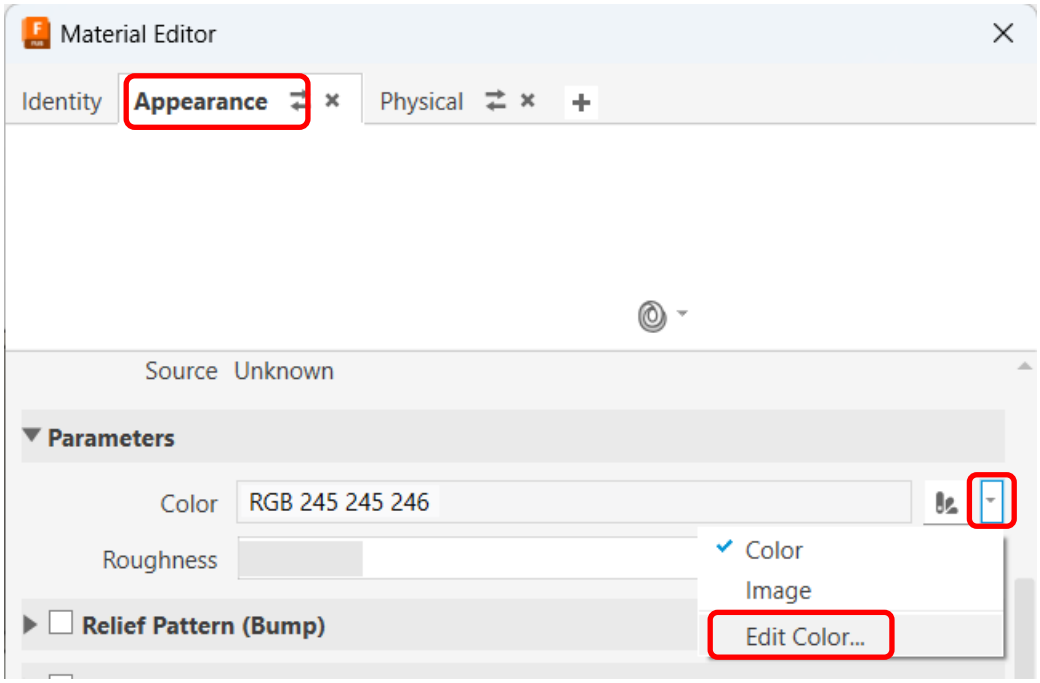
Thermal Conductivity 2.090E+02
Specific Heat 0.900
Thermal Expansion Coefficient 23.400

- click on the **Appearance** tab next

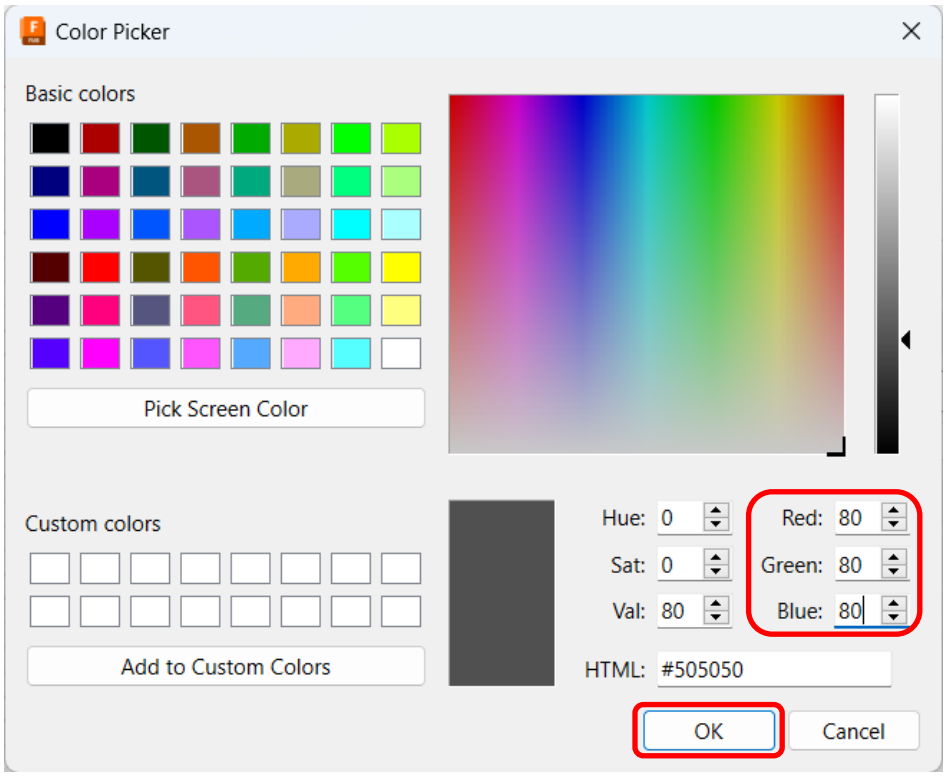


For our ***static*** thermal simulation, we only need **Thermal Conductivity**, however, we may as well set the 2 others.

- click on the **Appearance** tab
- click on the drop-down menu button and select **Edit Color...**



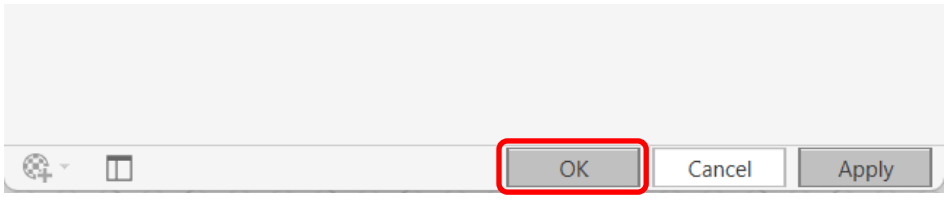
- change the **Reg, Green, and Blue values to 80**, which will result in a dark gray, and click **OK**



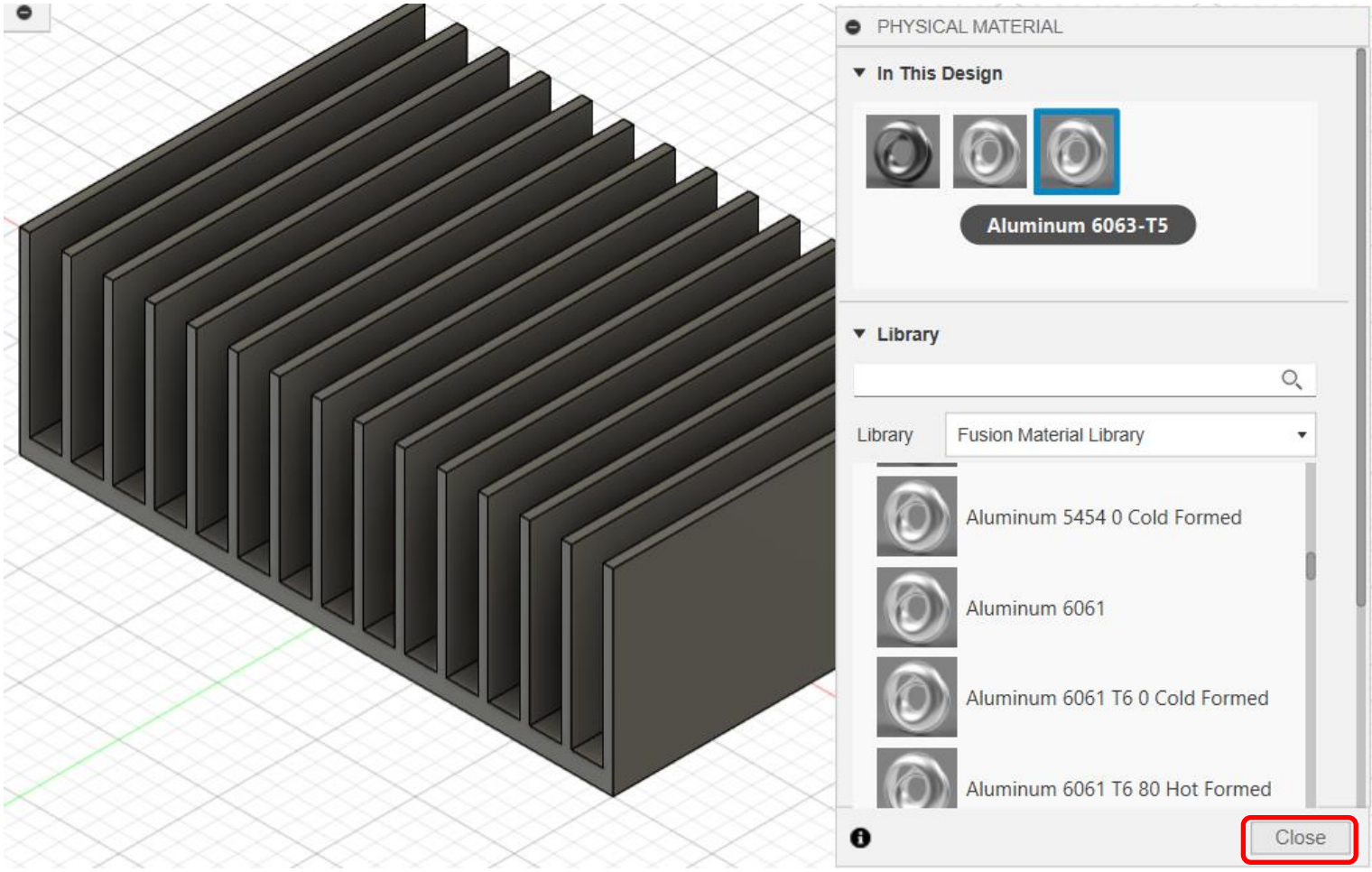
Heatsinks are often anodized black to increase the ***emissivity*** (closer to 1) to provide more effective ***radiational cooling***. However, anodizing the aluminum results in a high emissivity, even if it isn't dyed and remains as the natural aluminum color.

As far as the Fusion simulation, an emissivity value is entered as a value. Setting the color here is just for aesthetics.

- click on OK at the bottom of the Material Editor window. If it is grayed out, click Cancel.

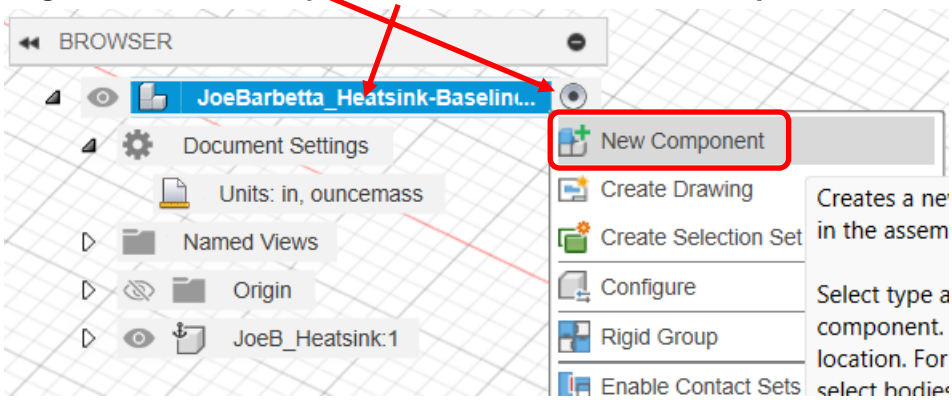


- the color of the heat sink should darken. Click **Close**

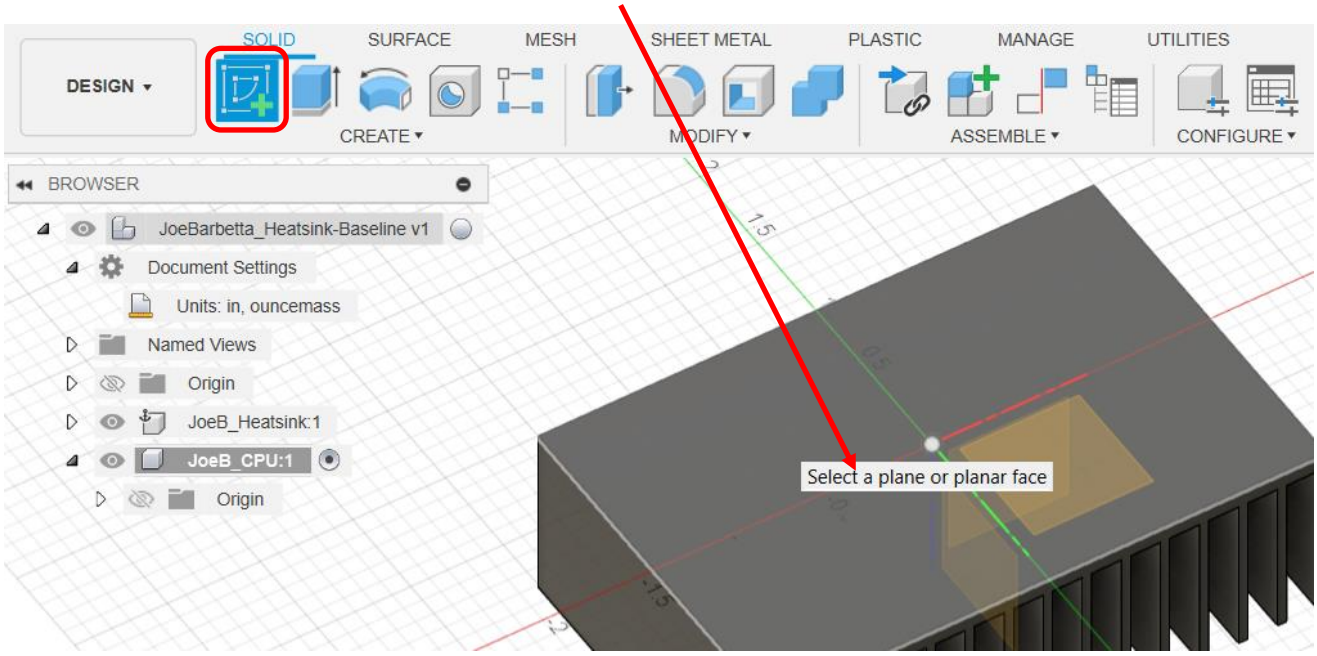


Creating the CPU Component

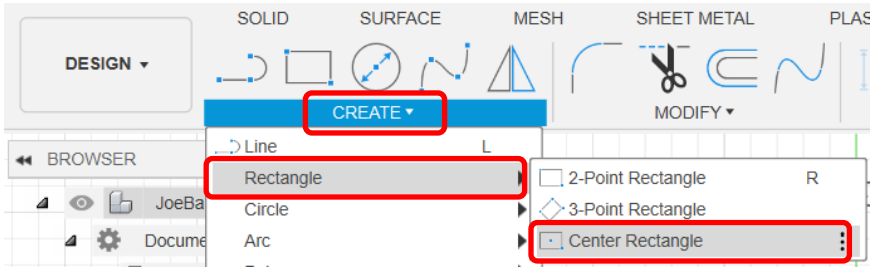
- click on the **circle at the right of the Project Name** to activate it
- right-click on the **Project Name** and select **New Component**



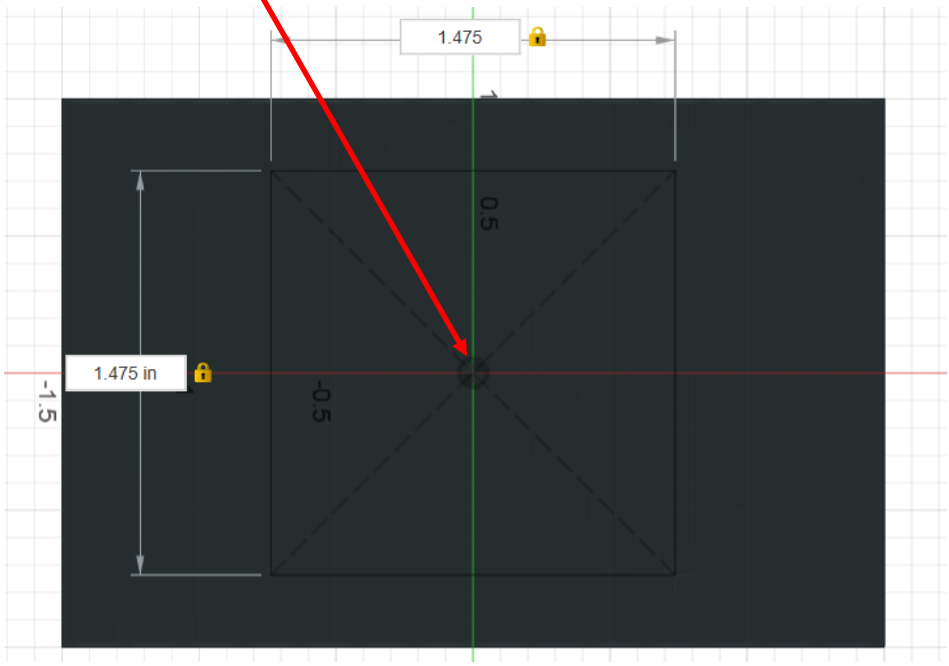
- use the **View Cube** to rotate the view to access the underside of the heat sink
- select **Create Sketch** and click on the **underside** of the heat sink



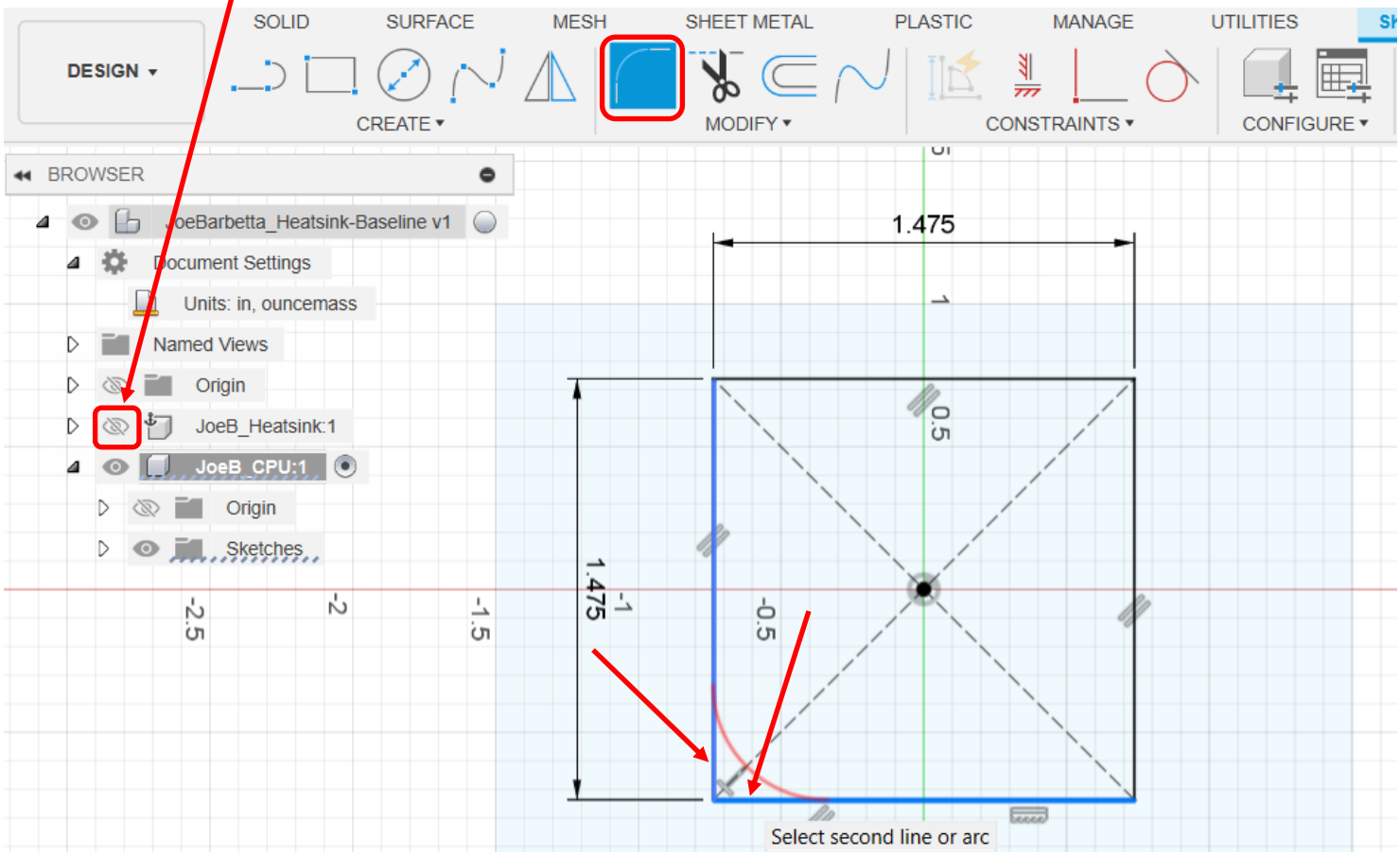
- from the **CREATE** menu, select **Rectangle** and **Center Rectangle**



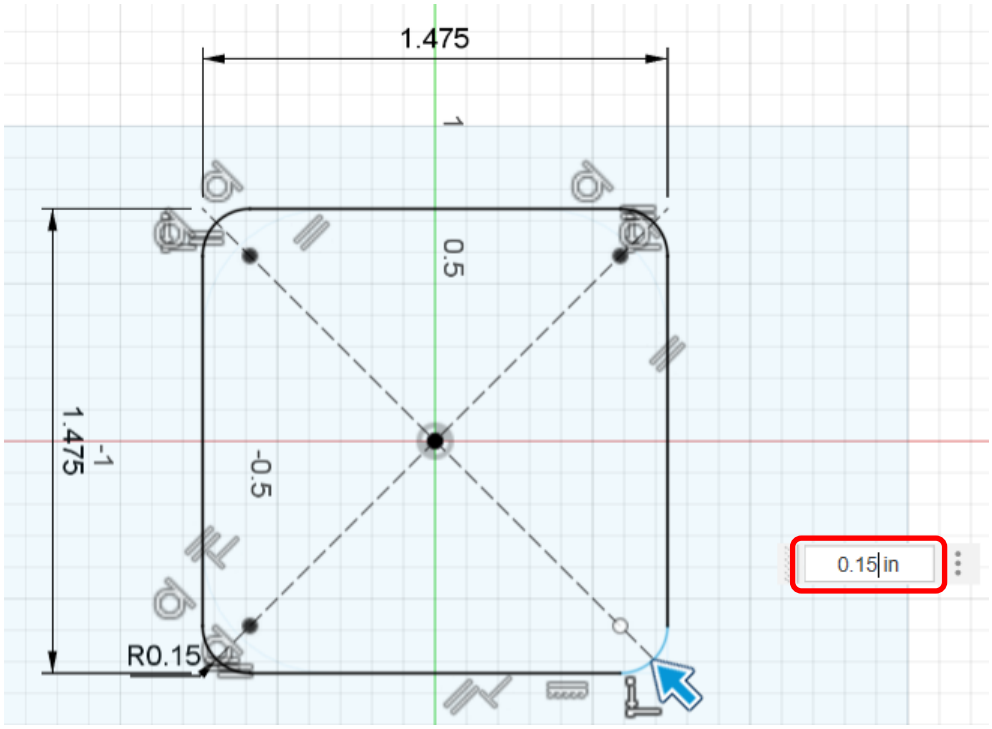
- click on the **Origin** and extend the rectangle down and to the right
- type **1.475**, press the **Tab key**, type **1.475**, and press the **Enter key**



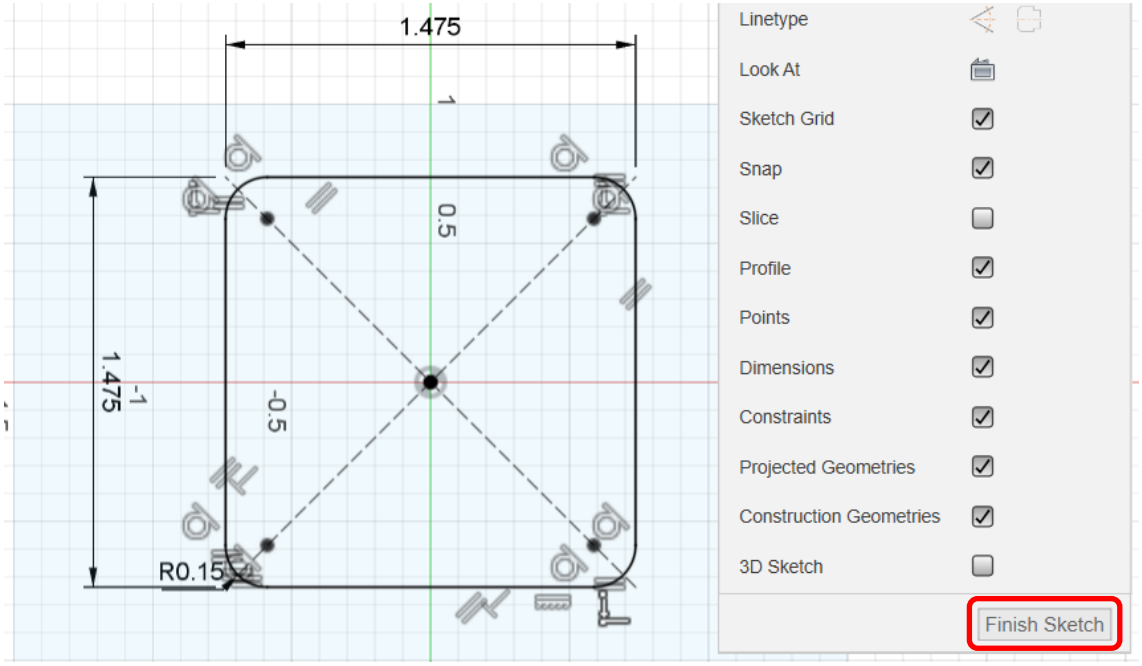
- click on the **eye icon** for the **Heatsink Component** to hide it
- select the **Fillet** tool. If it is not visible, find it in the MODIFY menu.
- click on the **ends of the 2 lines** near the bottom left corner



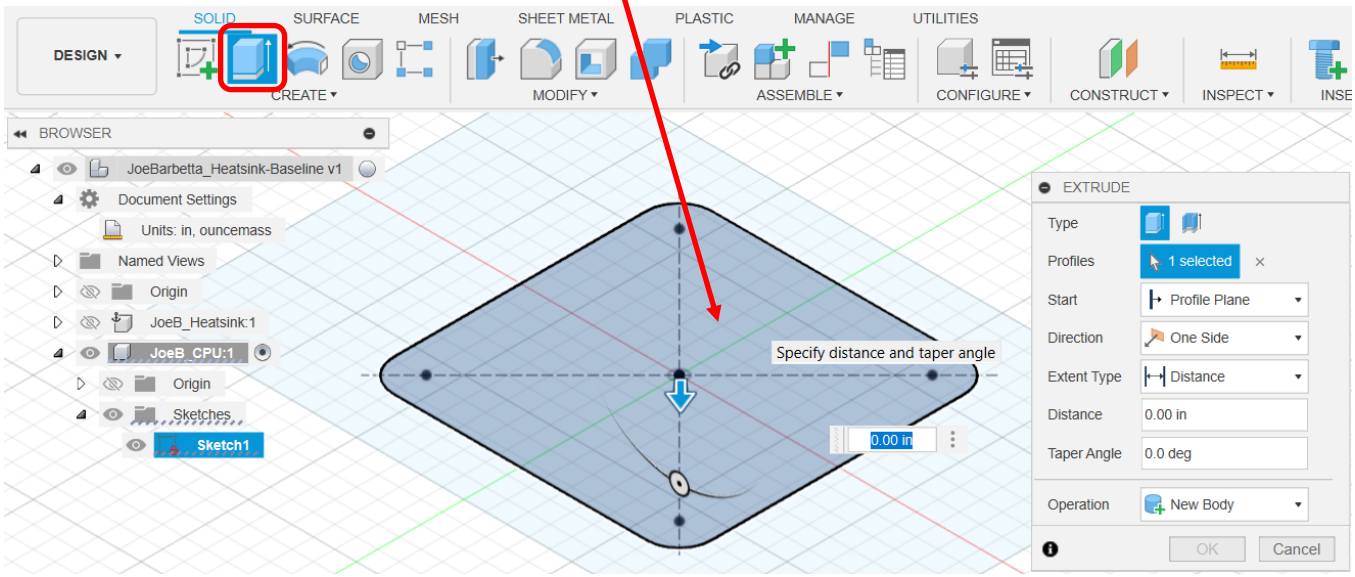
- click on the **lines near the 3 other corners**, type **0.15**, and press the **Enter key**
- gnore any warning mesage that pops up



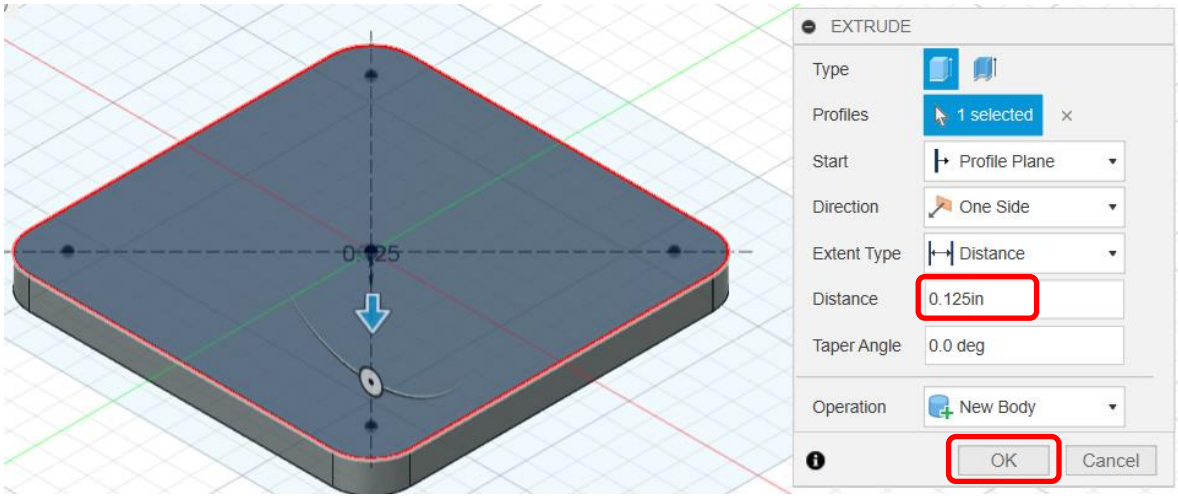
- click **Finish Sketch**



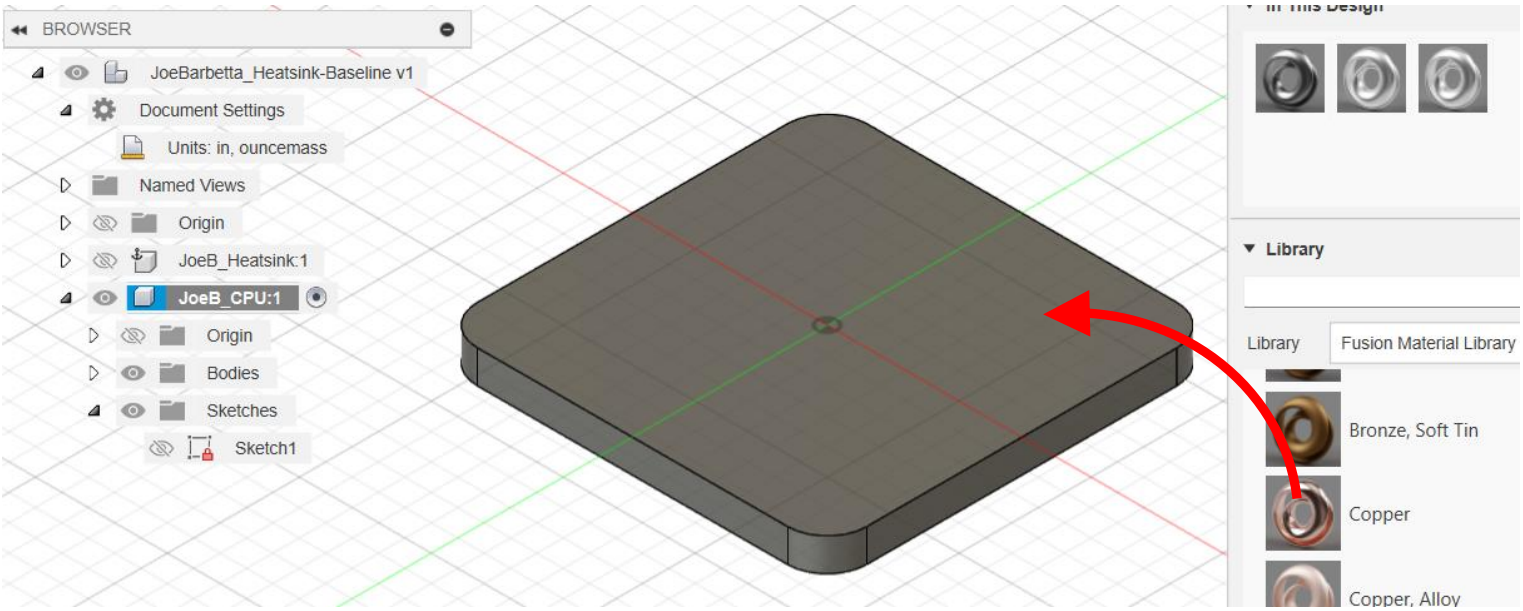
- select the **Extrude** tool and click on the **interior** of the rectangular region



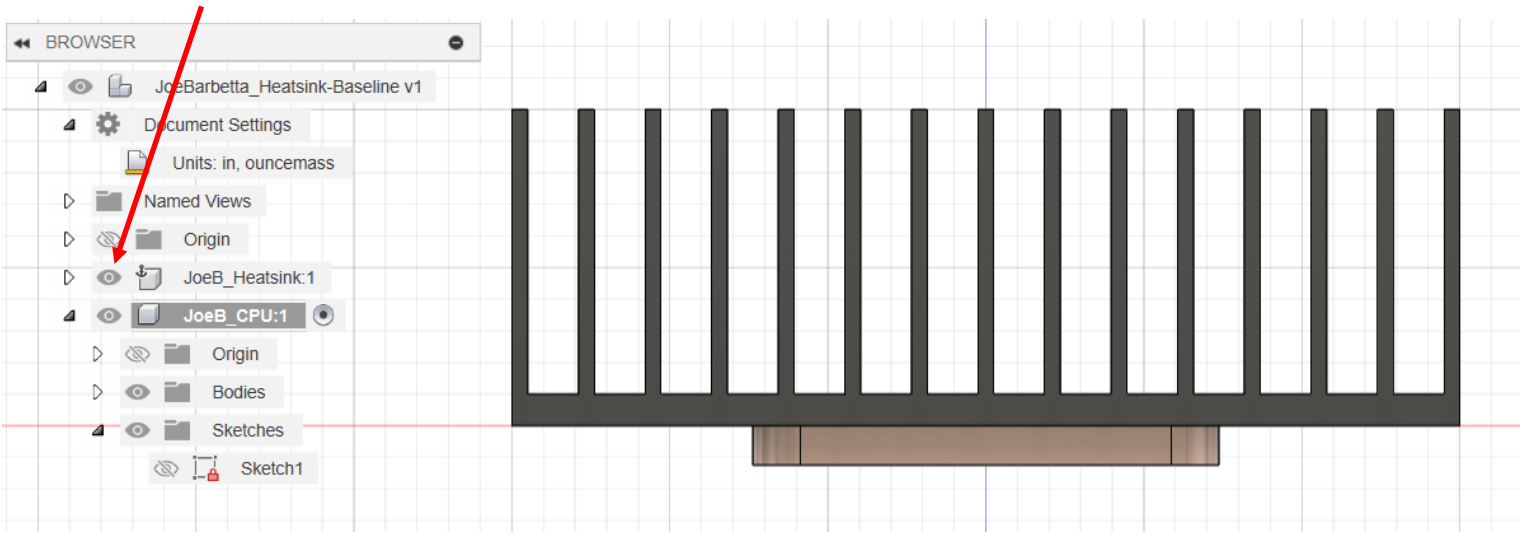
- enter **0.125** for **Distance** and click **OK**



- right-click on the **CPU Component name** and **select Physical Material**
- scroll down in the **Metal** folder to **Copper** and **drag its icon onto the CPU**

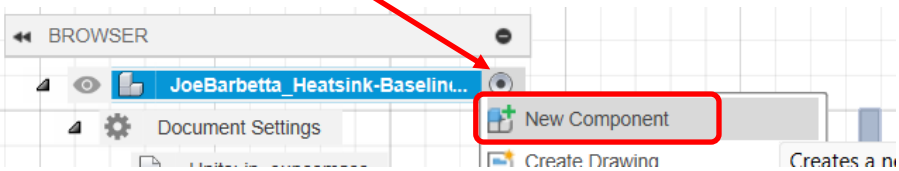


- click on the **eye** icon for the **Heatsink Comp**onent to make it visible again

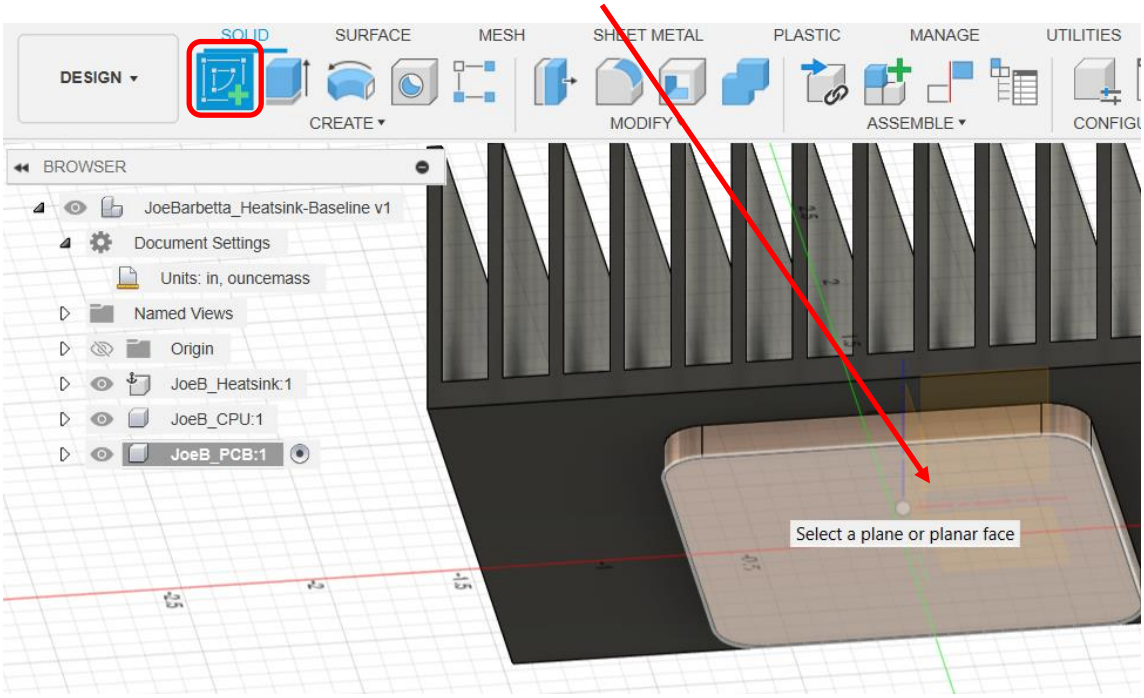


Creating the PCB (Printed Circuit Board)

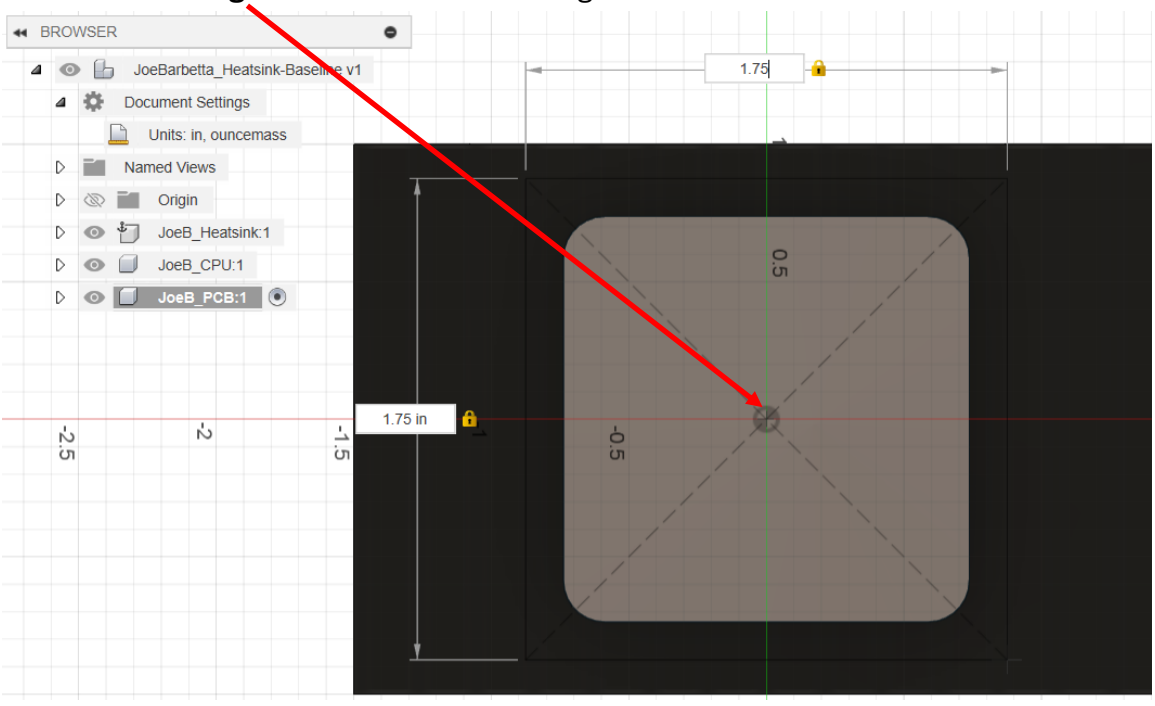
- click on the **circle** for the Project Name to activate it
- right-click on the name and select **New Component** and name the component with your name followed by "_PCB", e.g. **JoeB_PCB**



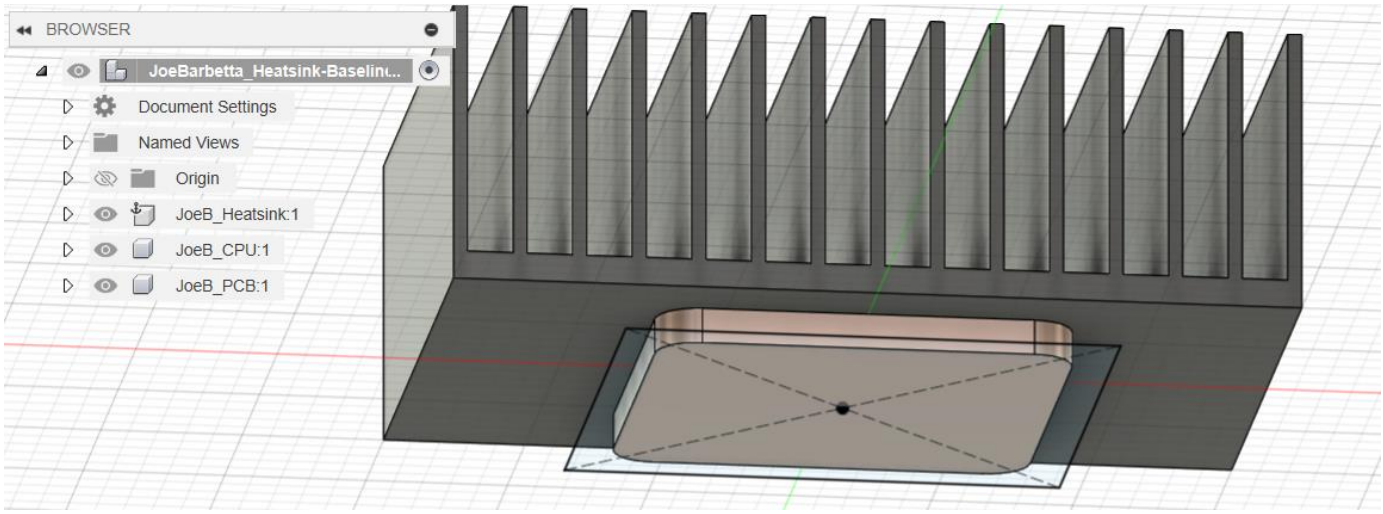
- select **Create Sketch** and click on the **bottom of the CPU**



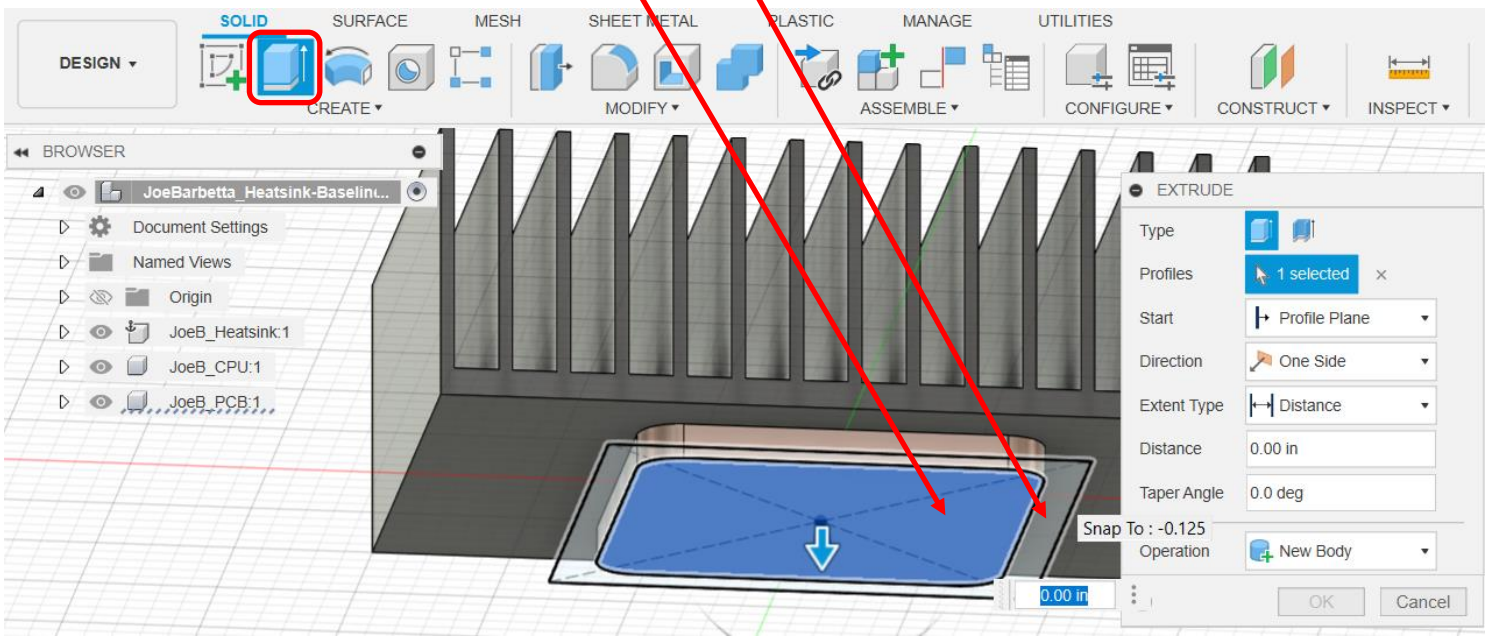
- from the **CREATE** menu, select **Rectangle** and **Center Rectangle**
- click on the **Origin** and extend the rectangle outward and set **both dimensions to 1.75**



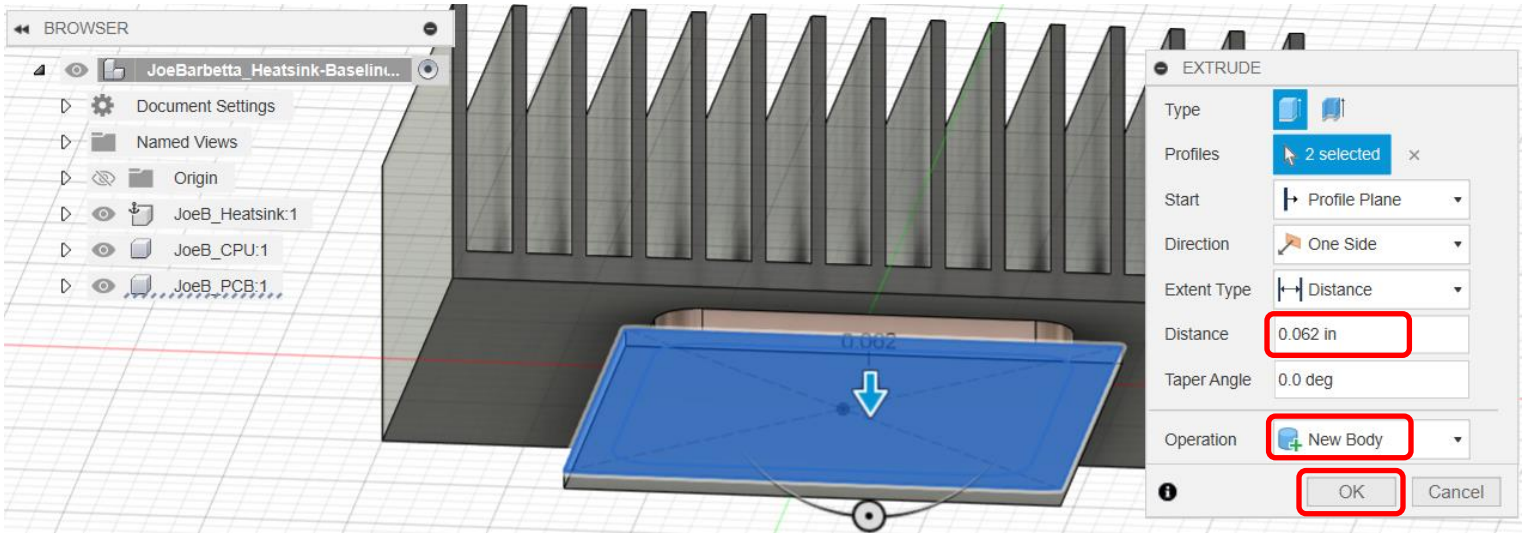
- click **Finish Sketch** and adjust the view to that shown below



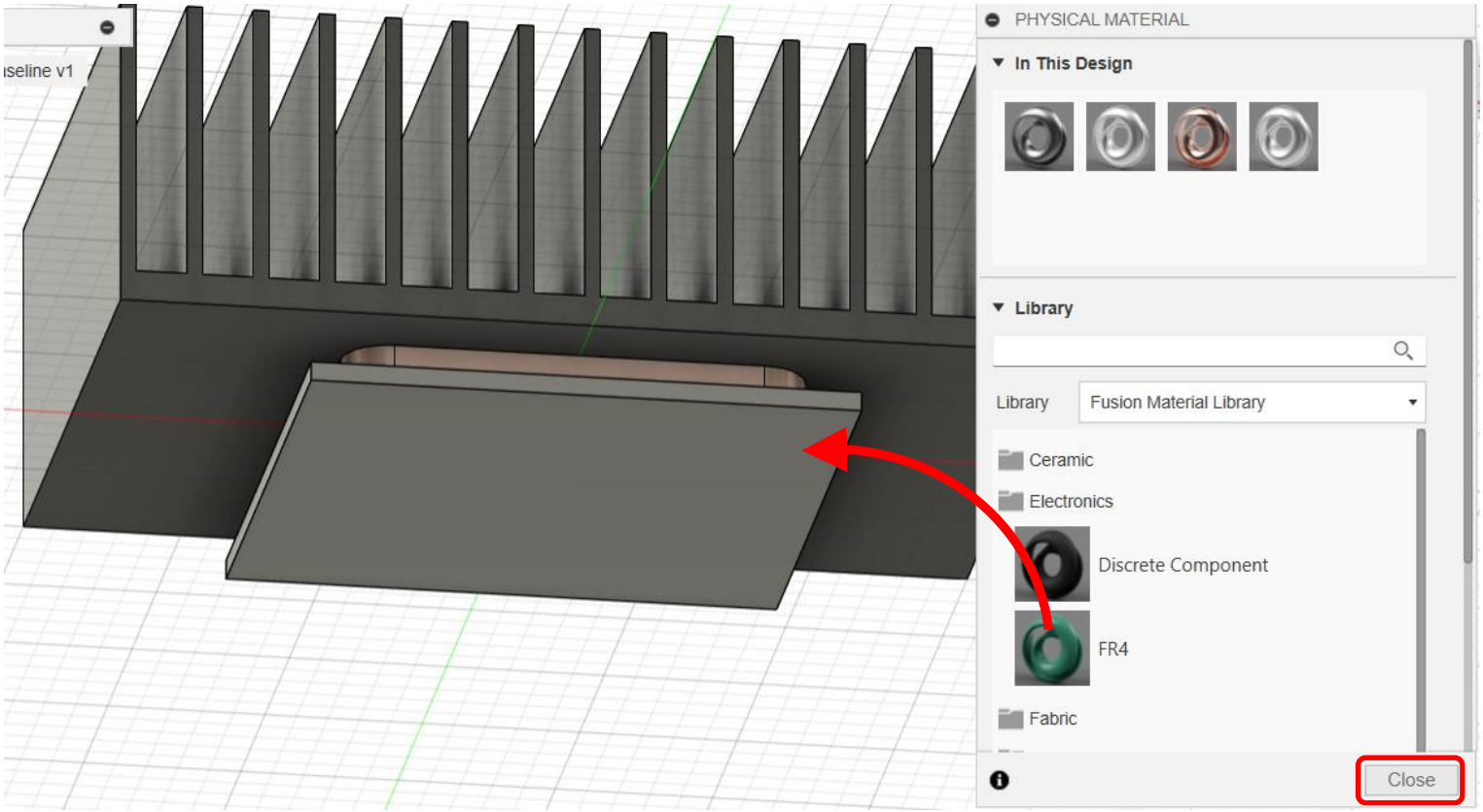
- select the **Extrude** tool and click on the **inner and outer regions** to turn them blue



- enter **0.062** for **Distance**, ensure that **Operation** is **New Body**, and click **OK**

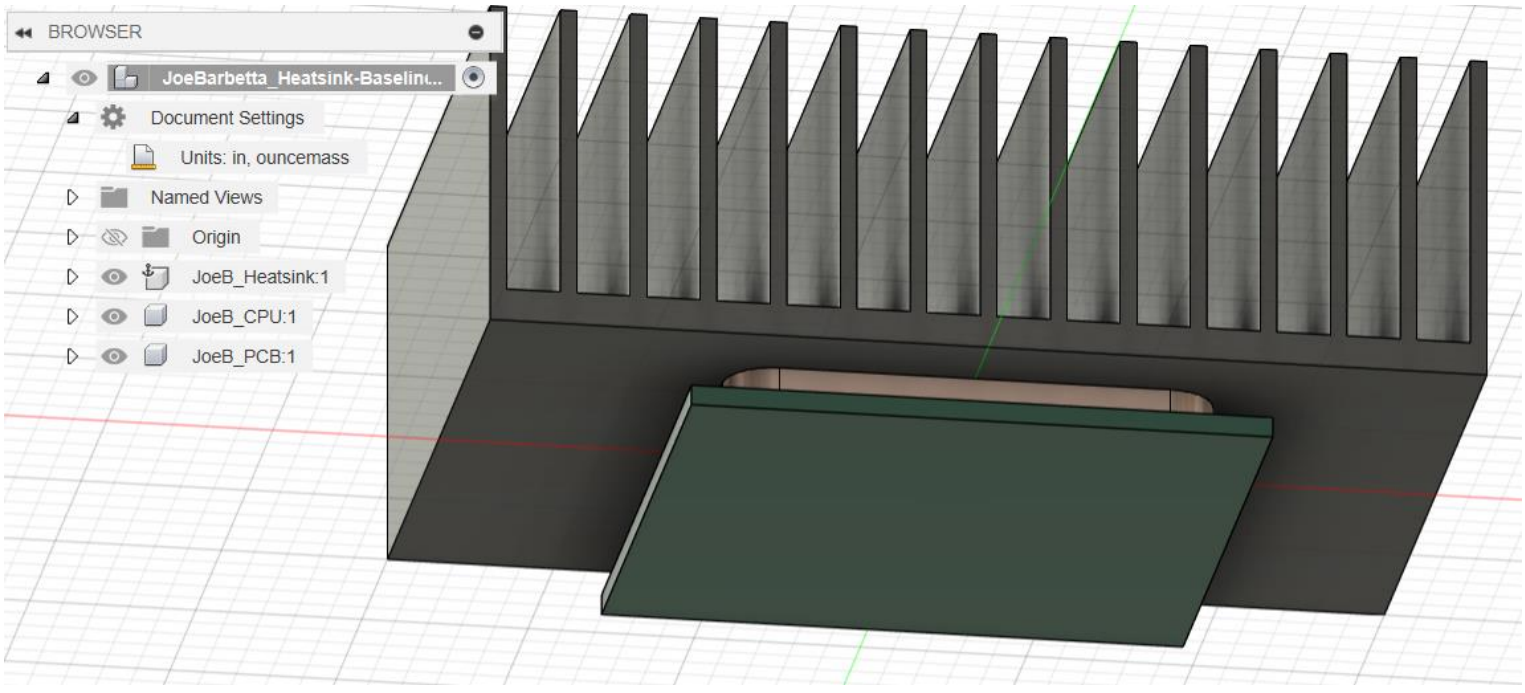


- right-click on the **CPU Component name** and **select Physical Material**
- scroll down to the **Electronics** folder and click on it
- **drag the FR4 icon onto the PCB** and click **Close**

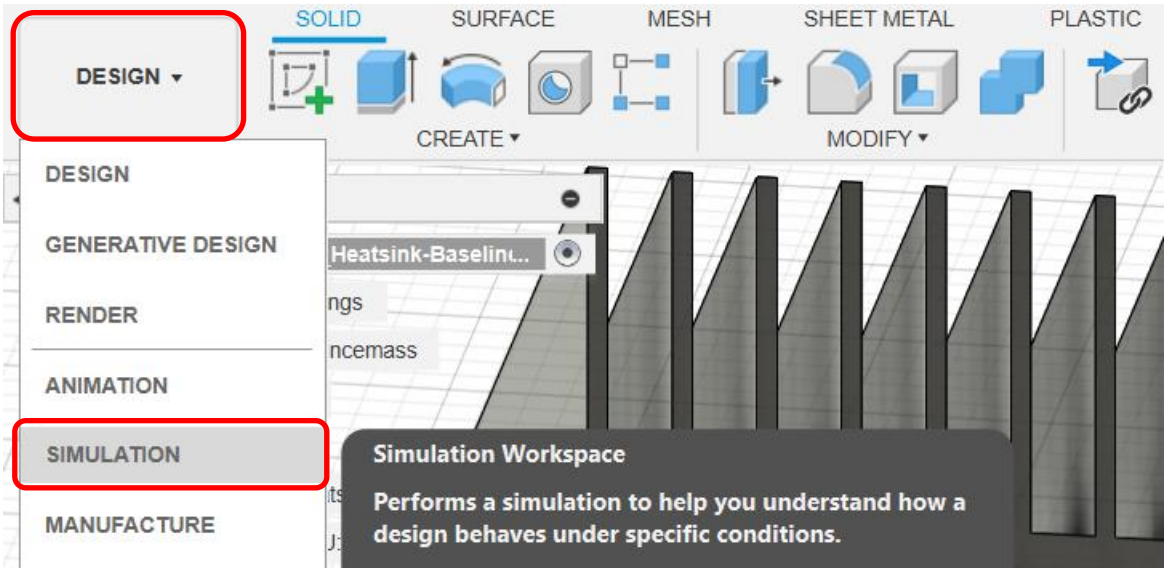


The PCB now shows as green.

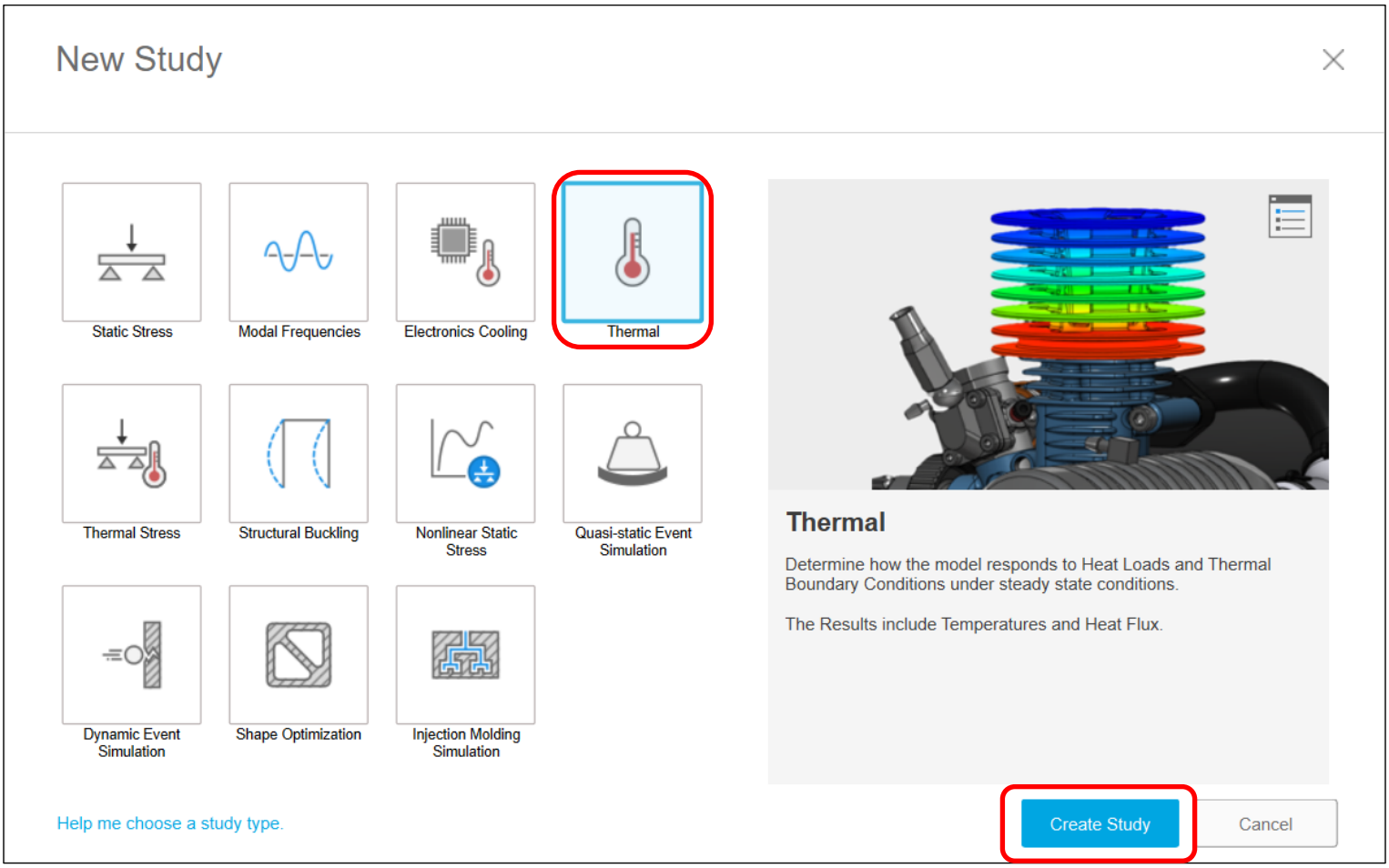
Note that a PCB (Printed Circuit Board) would be much larger. It thickness would be the same, but it would cover a much larger area to support many other electronic components. A small section is added to represent the PCB.



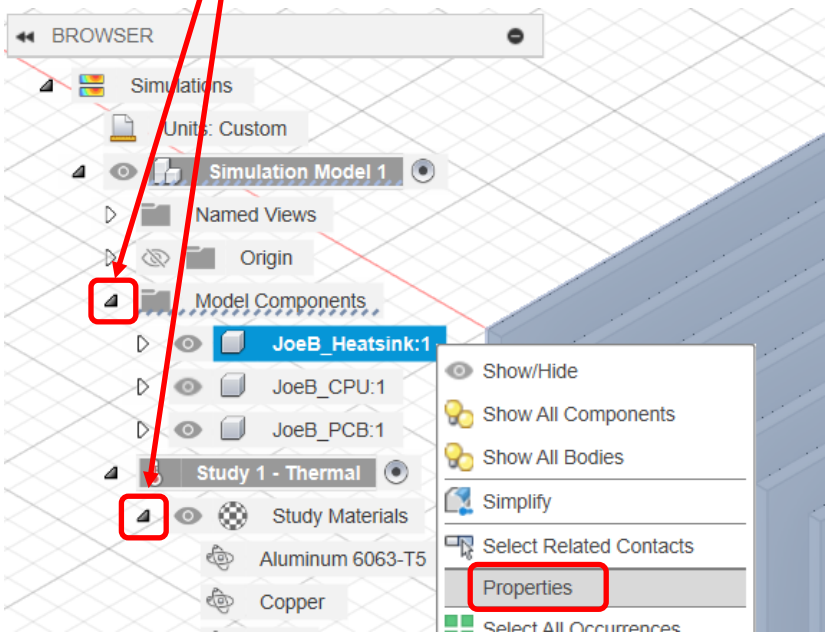
- change the **Workspace** from **DESIGN** to **SIMULATION**

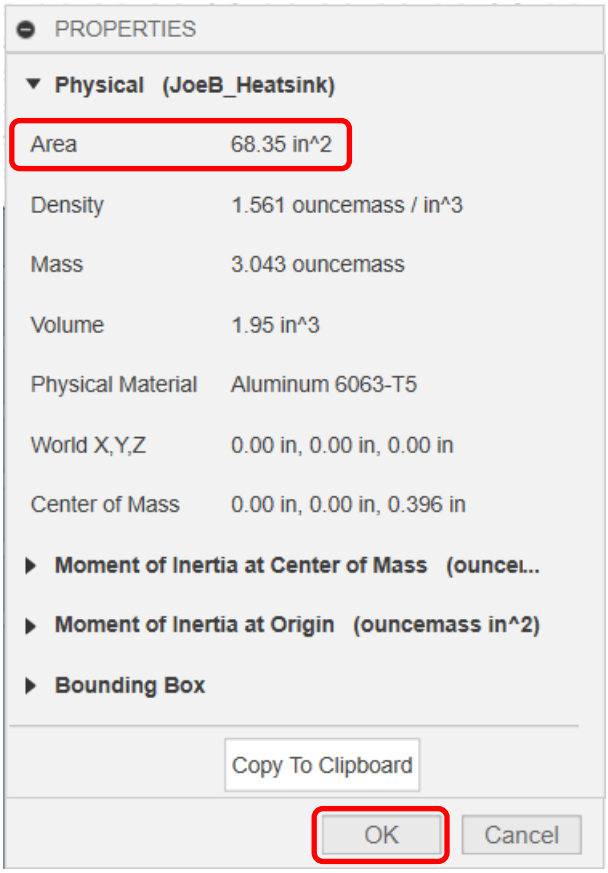


- yell **"Dude, that's a lot of simulations!"**
- select **Thermal** and click the bottom right **Create Study**



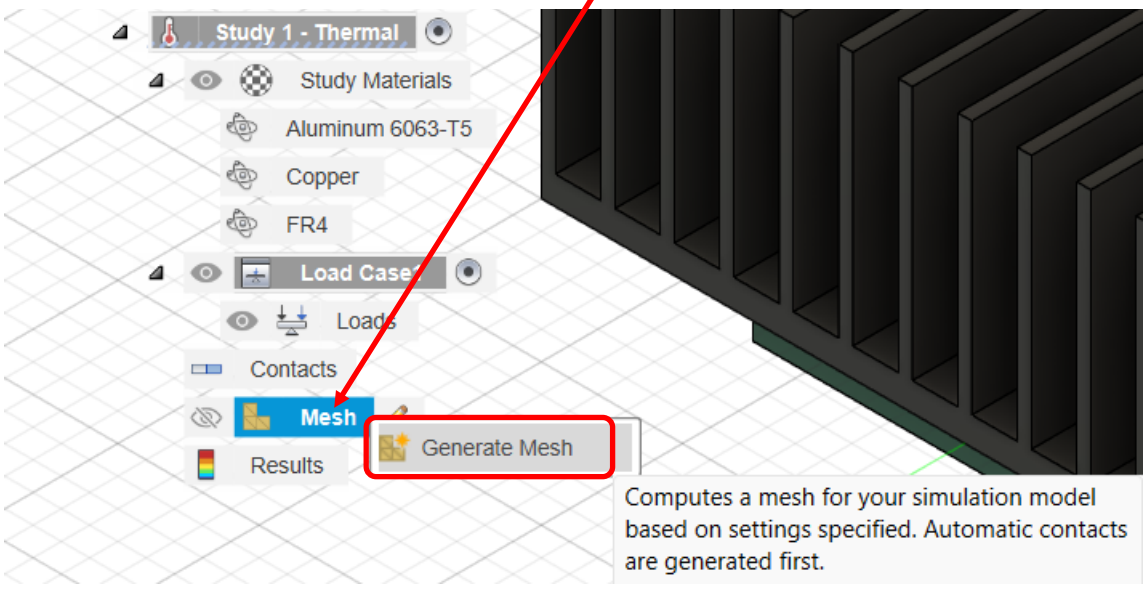
- click on the **arrows** for **Model Components** and **Study Materials** to open these folders
- right-click on the **Heatsink Component** and select **Properties**
- note the **Physical** properties shown. **Area** is ***surface area*** shown here in in^2. Click **OK** to close this window.





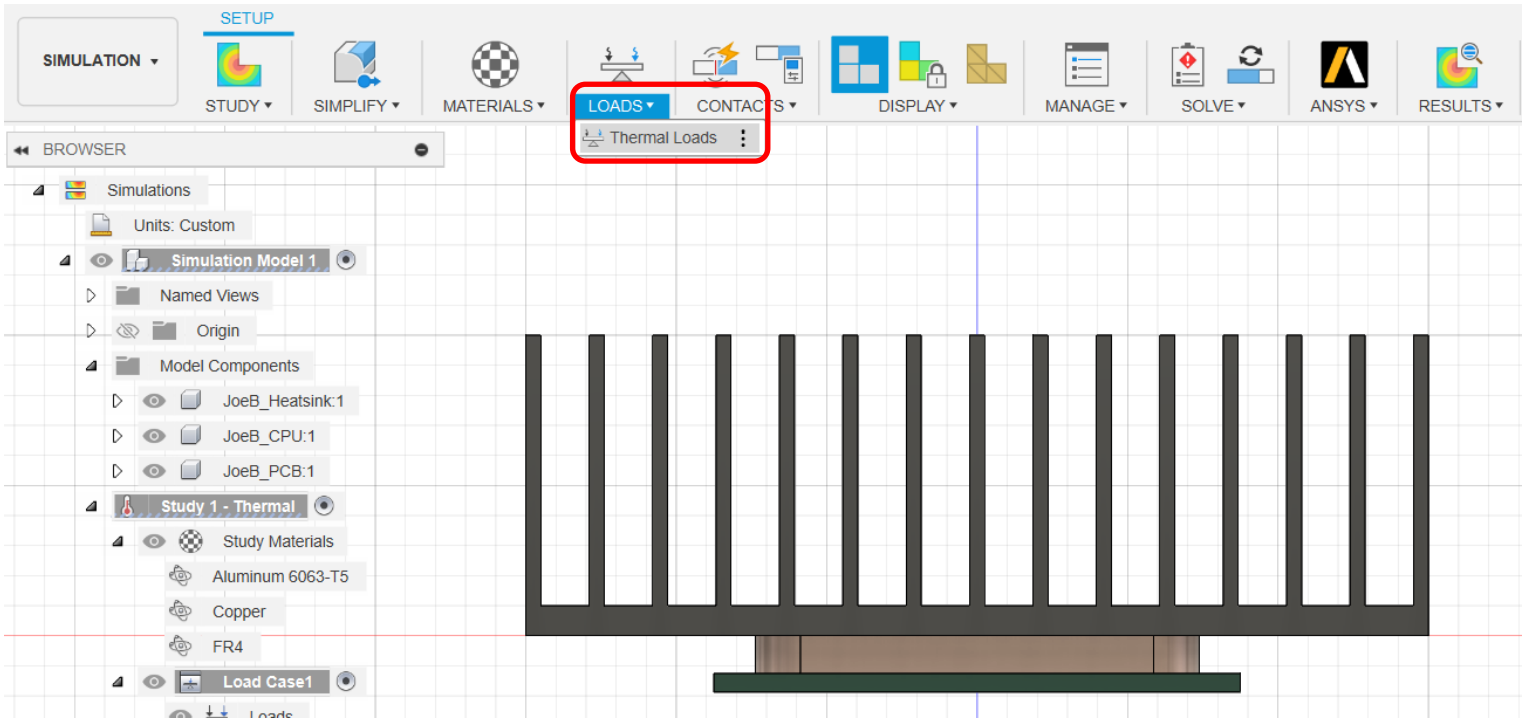
- at the bottom of the BROWSER, right-click on **Mesh** and **select Generate Mesh**

Whenever design changes are made to the heatsink, such as changing fin count, or any dimension of the heatsink, a new mesh must be generated.

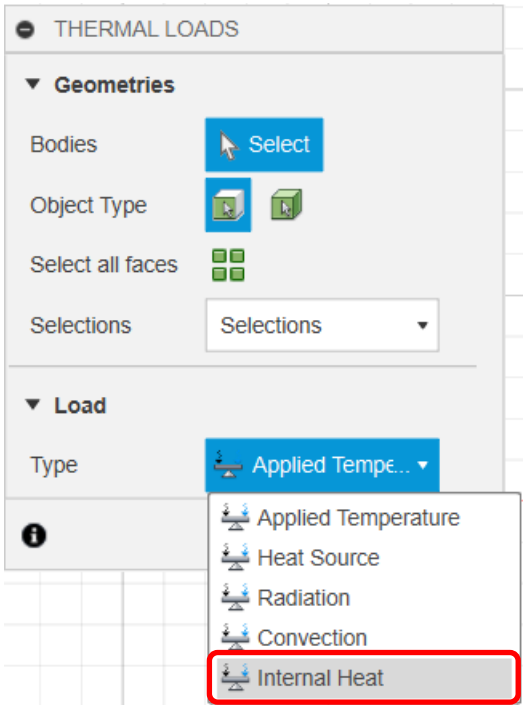


Applying a Thermal Load for Internal Heat

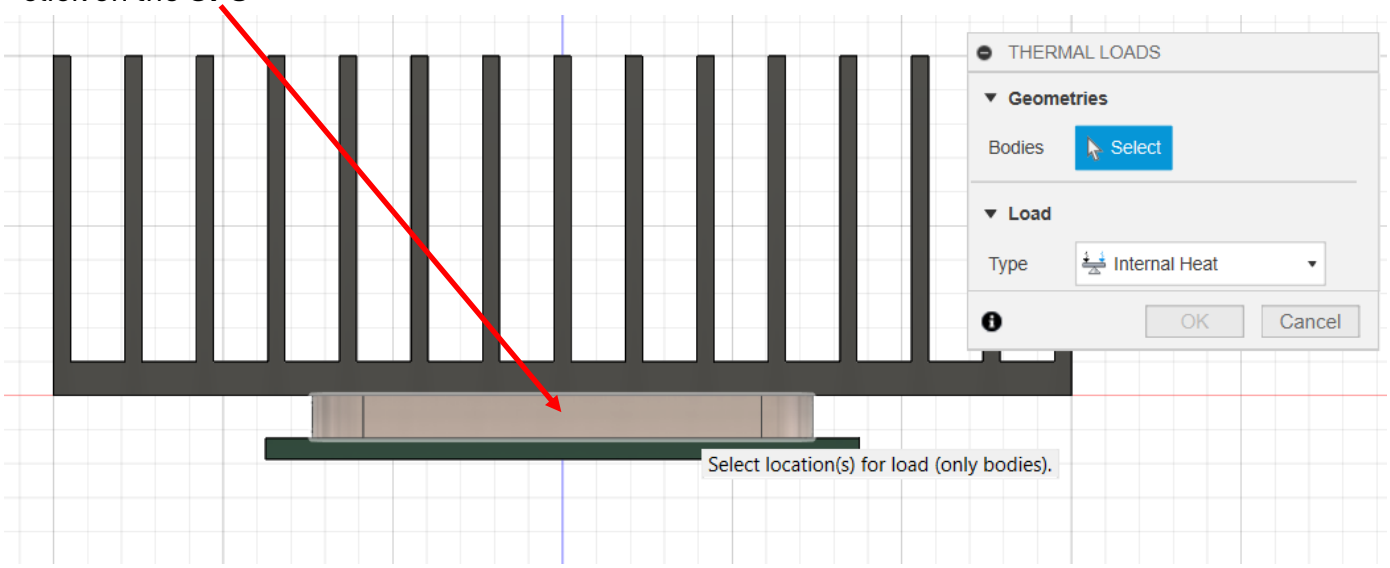
- click on the **FRONT** face of the **View Cube** to achieve the below view
- from the **LOADS** menu select **Thermal Loads**



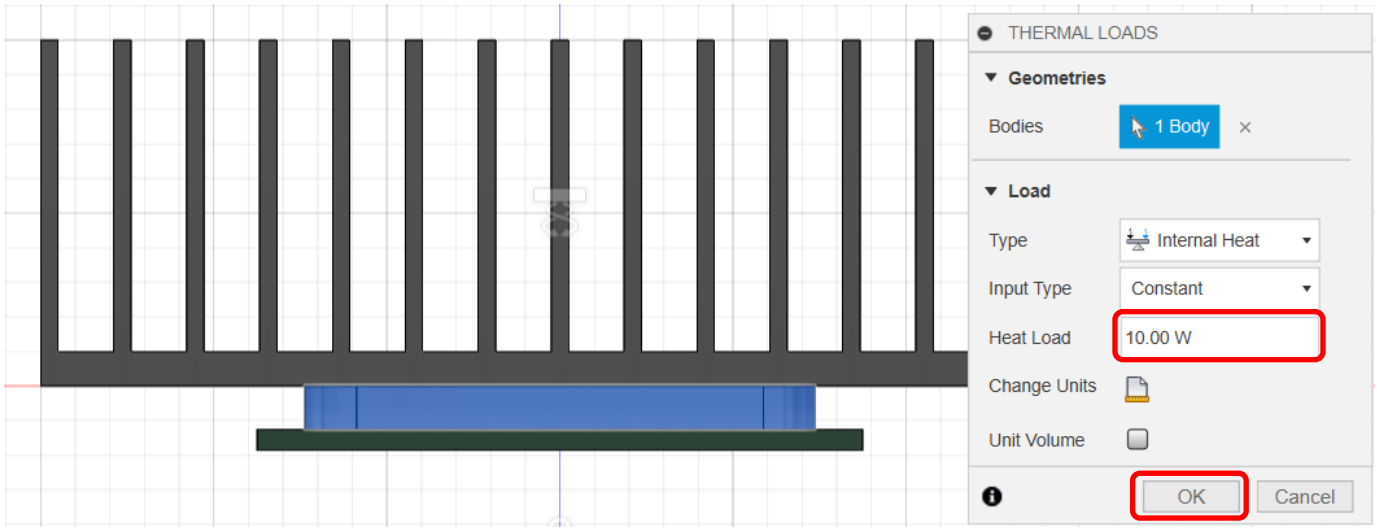
- for **Type** select **Internal Heat**



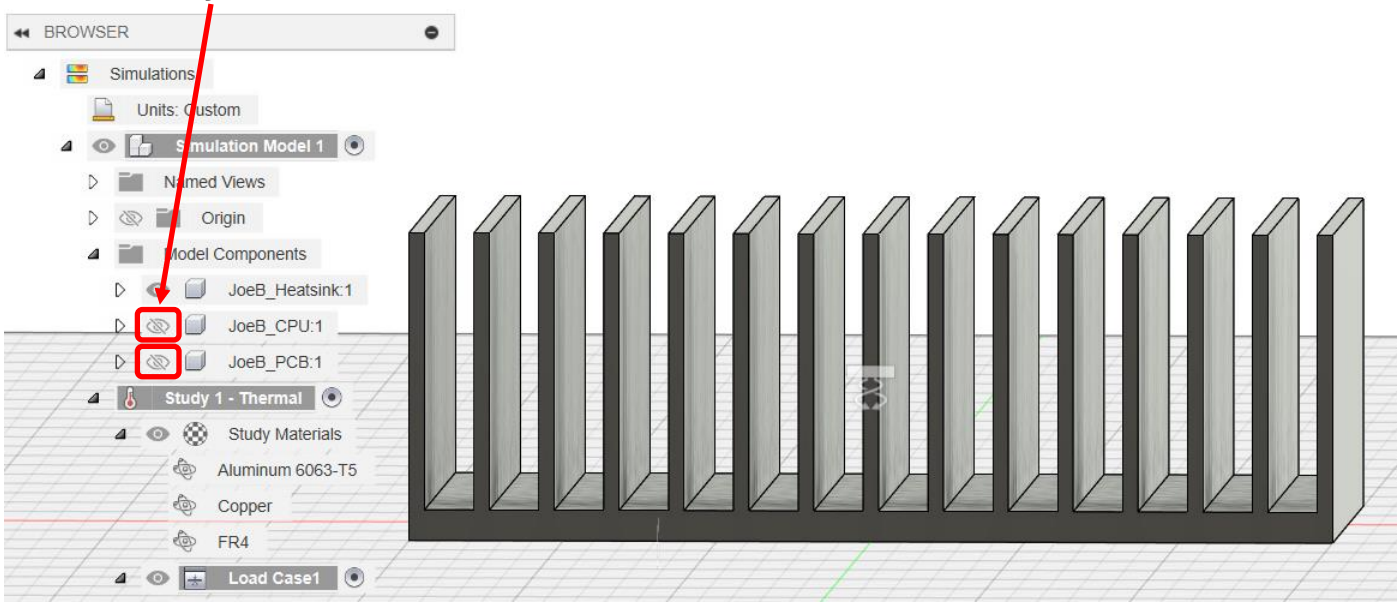
- click on the **CPU**



- set the **Heat Load** to **10.00** W (Watts) and click **OK**

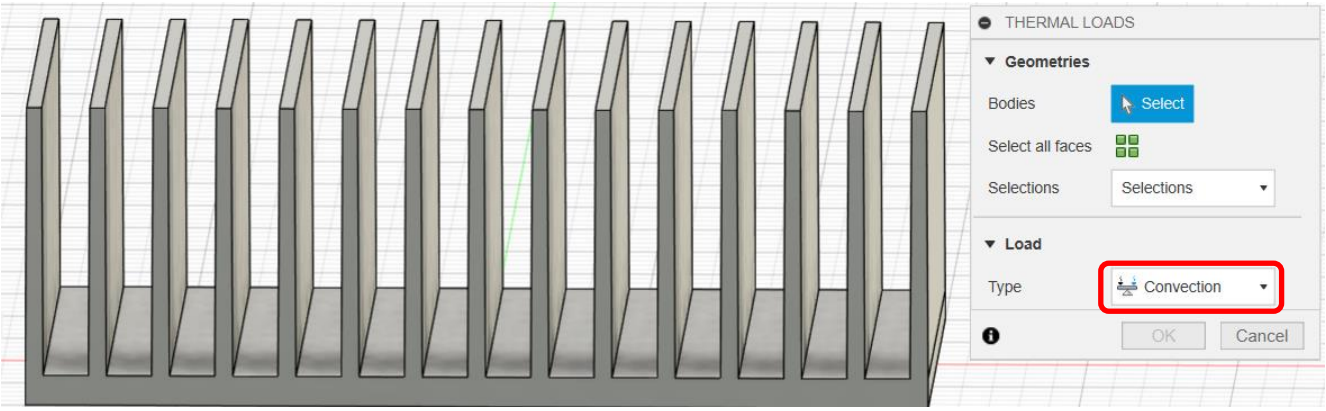


- click on the **eye** icons for the **CPU** and **PCB** to hide them

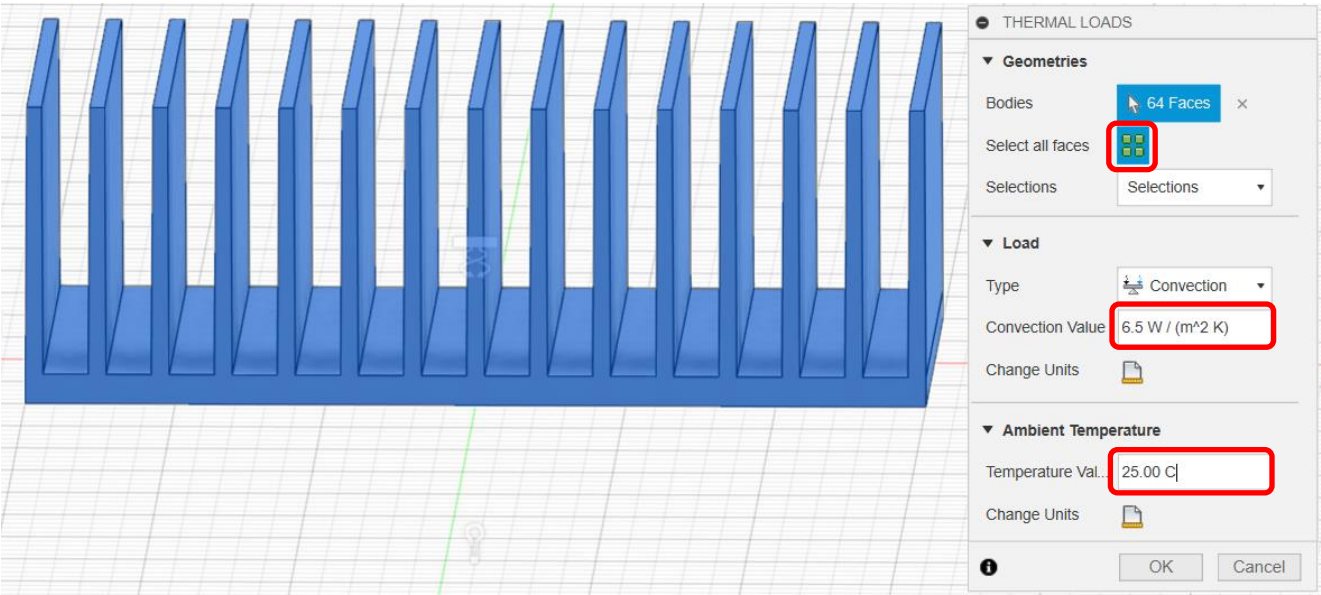


Applying a Thermal Load for Convection

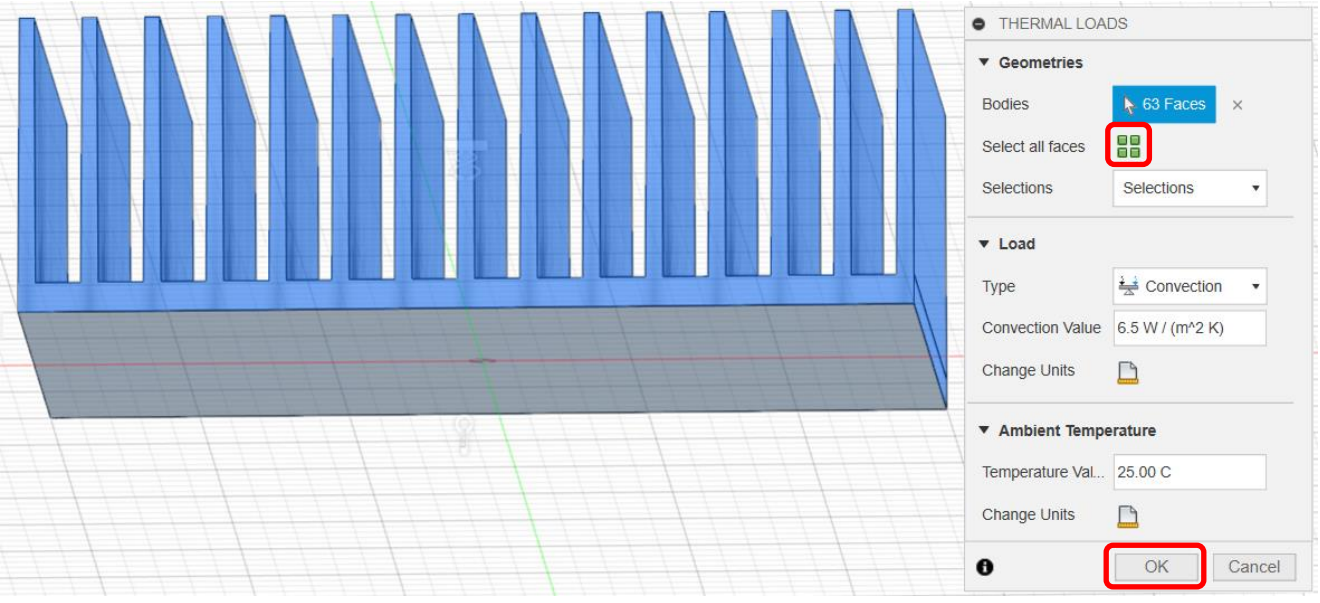
- from the **LOADS** menu select **Thermal Loads** and set the **Type** to **Convection**



- click on the **Select all faces** icon to highlight it and click anywhere on the heatsink
- set the **Convection Value** to **6.5** and the **Ambient Temperature value** to **25**



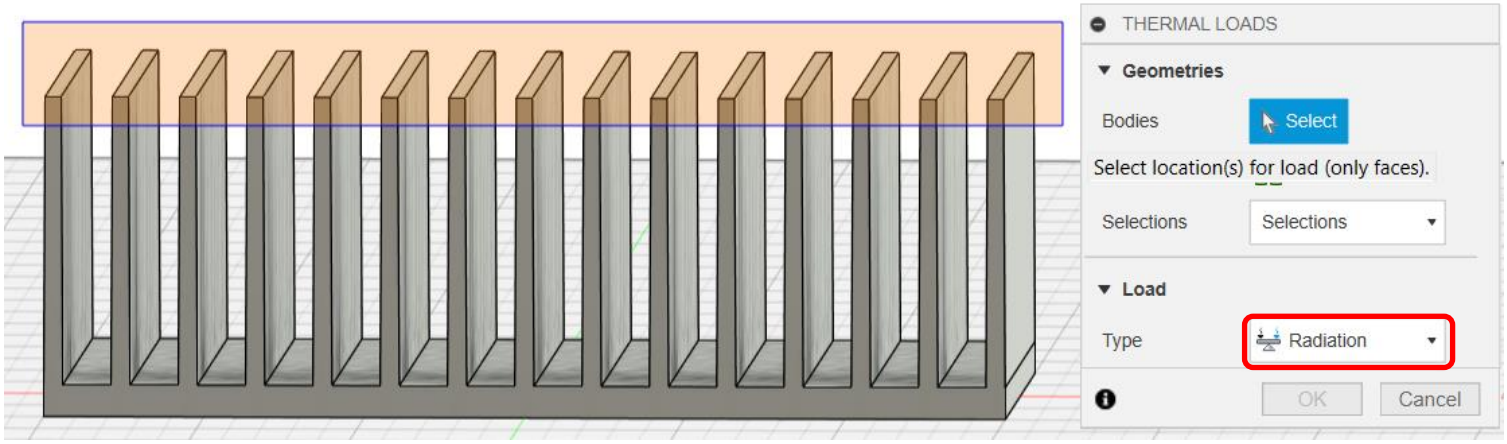
- turn the **View Cube** to access the **bottom of the heat sink**
- click on the **Select all faces** icon to **remove the highlighting** and click on the **bottom surface** and click **OK**



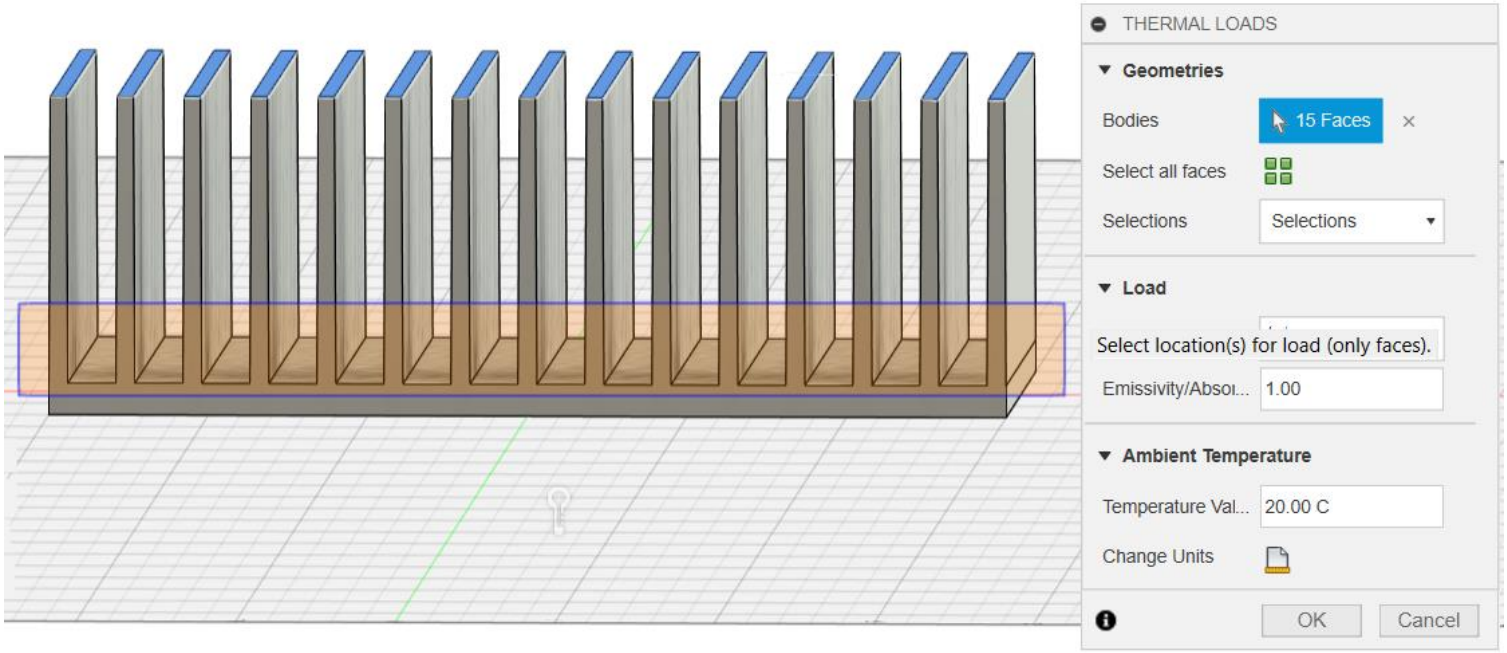
Applying a Thermal Load for Radiation

It should be noted that the steps used here will select all the faces except the interior fin surfaces. This may seem odd at first, but the radiation emitted by an interior fin surface will be absorbed by the opposite fin surface. If all the faces were selected, the simulation would not account for this. It will assume that every face will be emitting radiation away from the heat sink.

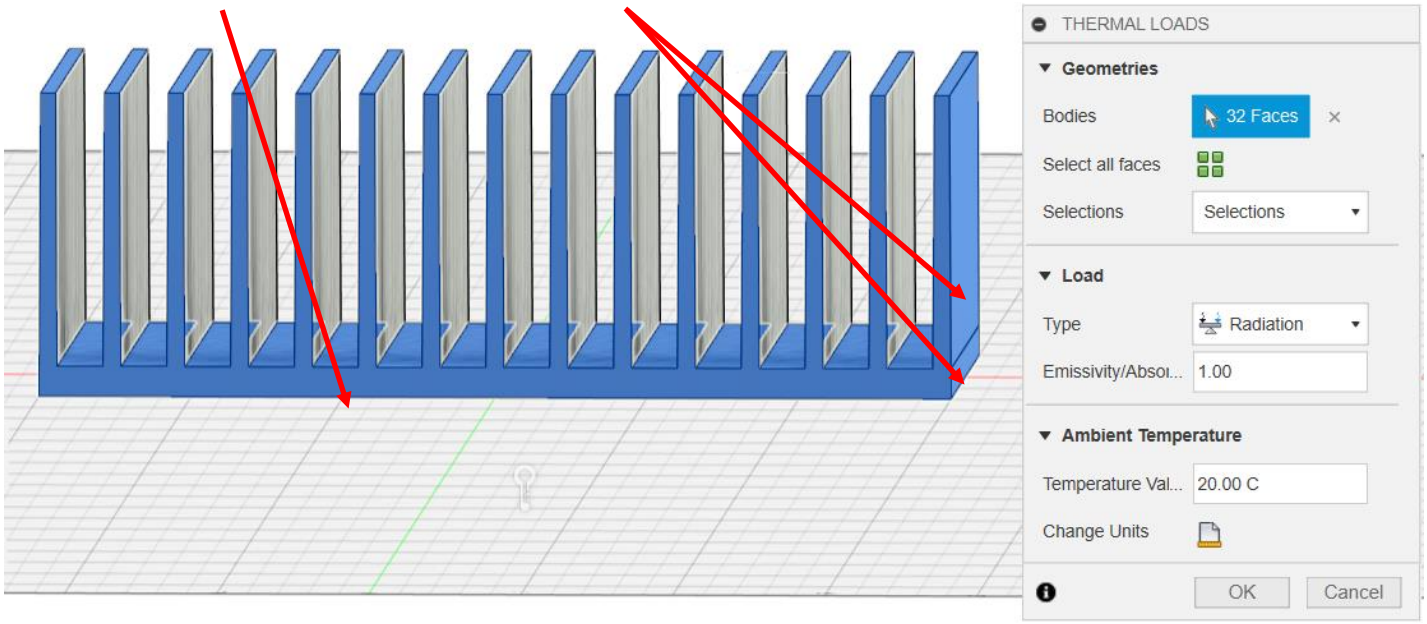
- from the **LOADS** menu select **Thermal Loads** and set the **Type** to **Radiation**
- drag the selection rectangle **over the tops of the fins**, which should result in the tops turning blue



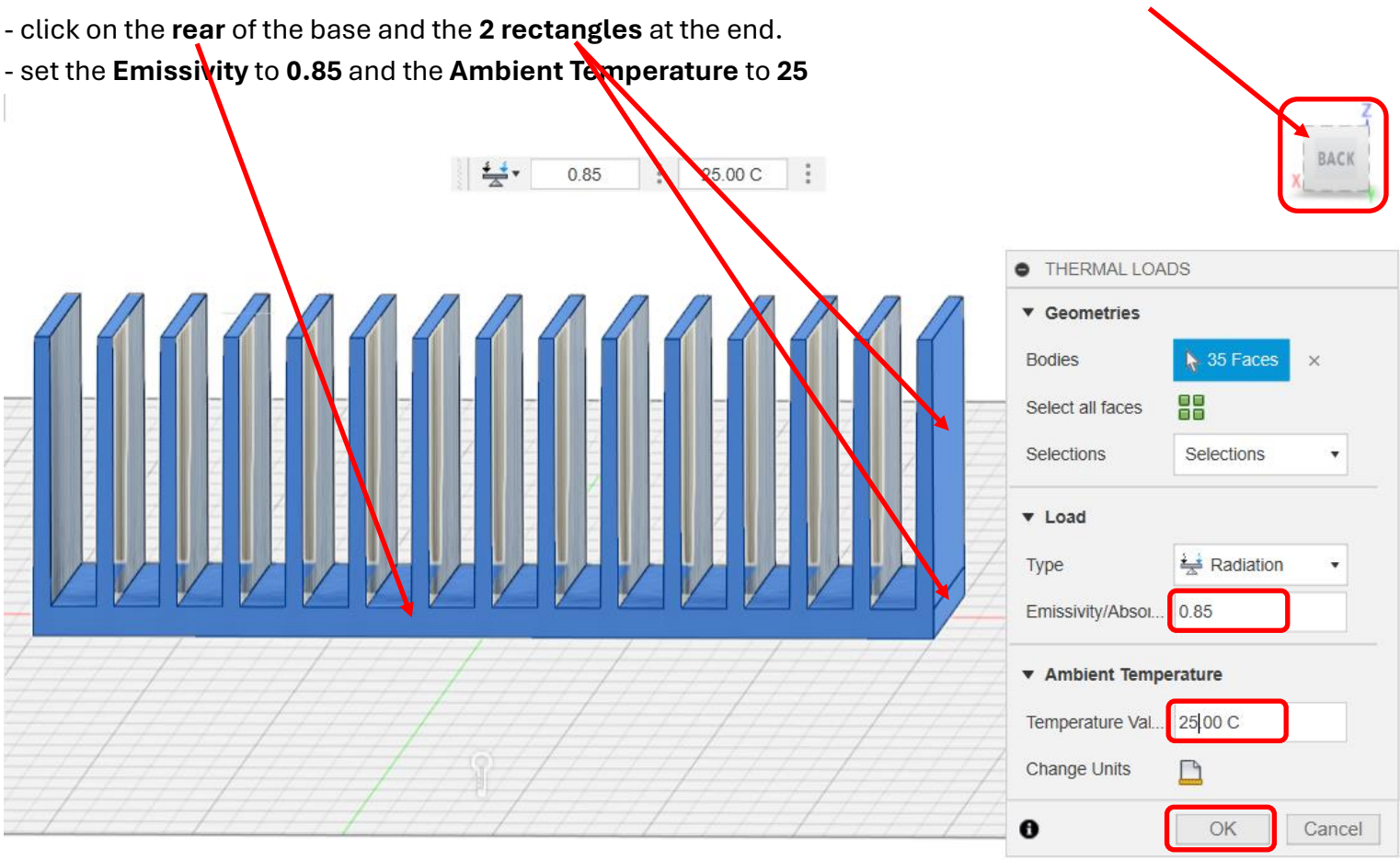
- drag the selection rectangle over the **bottoms of the fins**, which should result in the bottoms of the spaces between fins turning blue



- click on the **front of the base** and the **2 rectangles at the end**



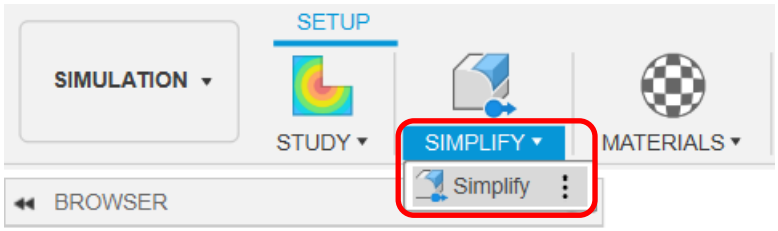
- use the **View Cube** to rotate the view to access the **back of the heat sink**. Note the View Cube here.
- click on the **rear** of the base and the **2 rectangles** at the end.
- set the **Emissivity** to **0.85** and the **Ambient Temperature** to **25**



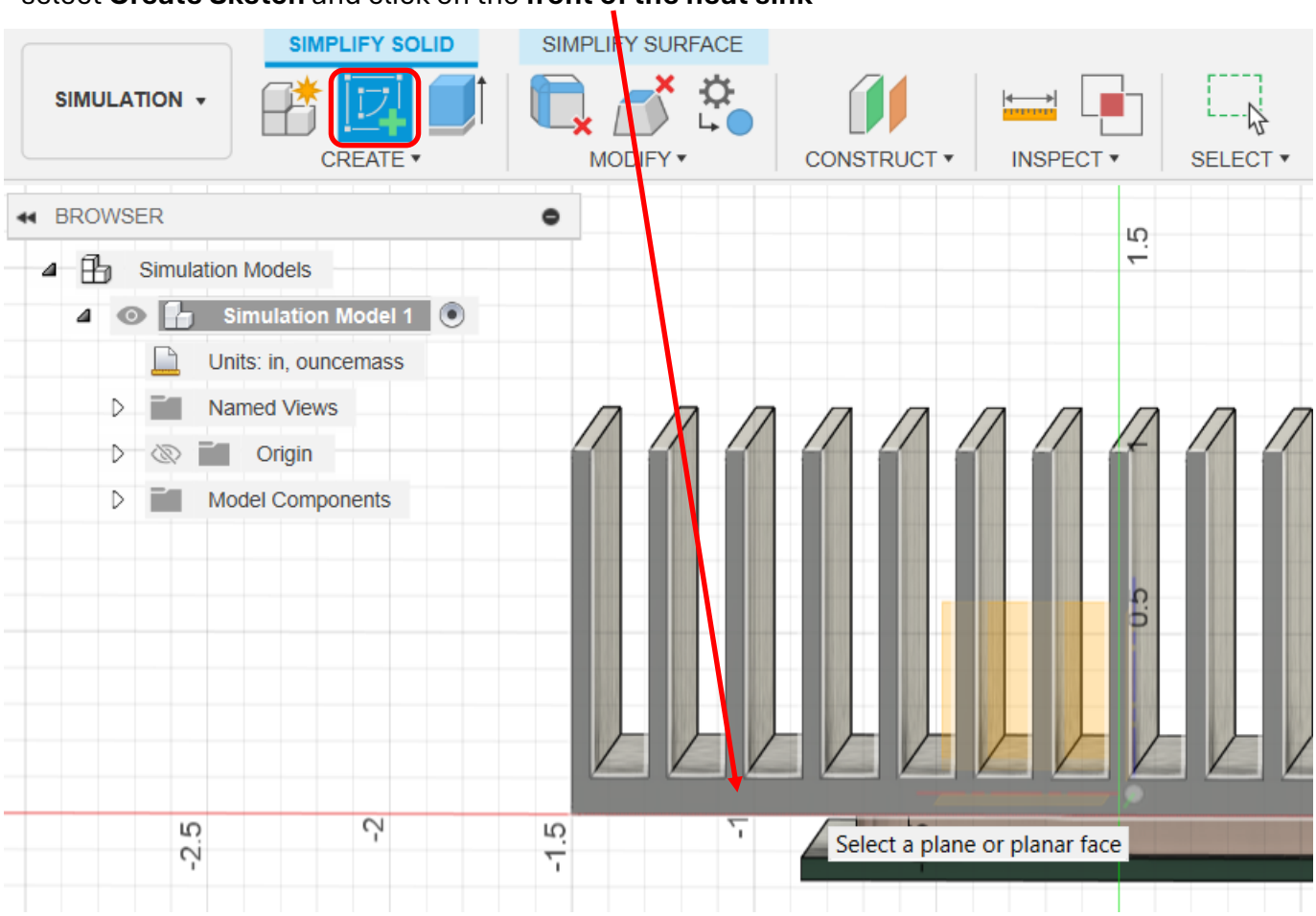
Simplifying the Simulation

In many application, the Simplify options will simplify the model by allowing one to remove features that are not needed for the simulation. One will note that the simplify tools are essentially design tools and will create a new model only used for the simulation. In our case, we will be adding a small block that we can set to an ambient temperature to act as a reference for the coloring of the temperatures.

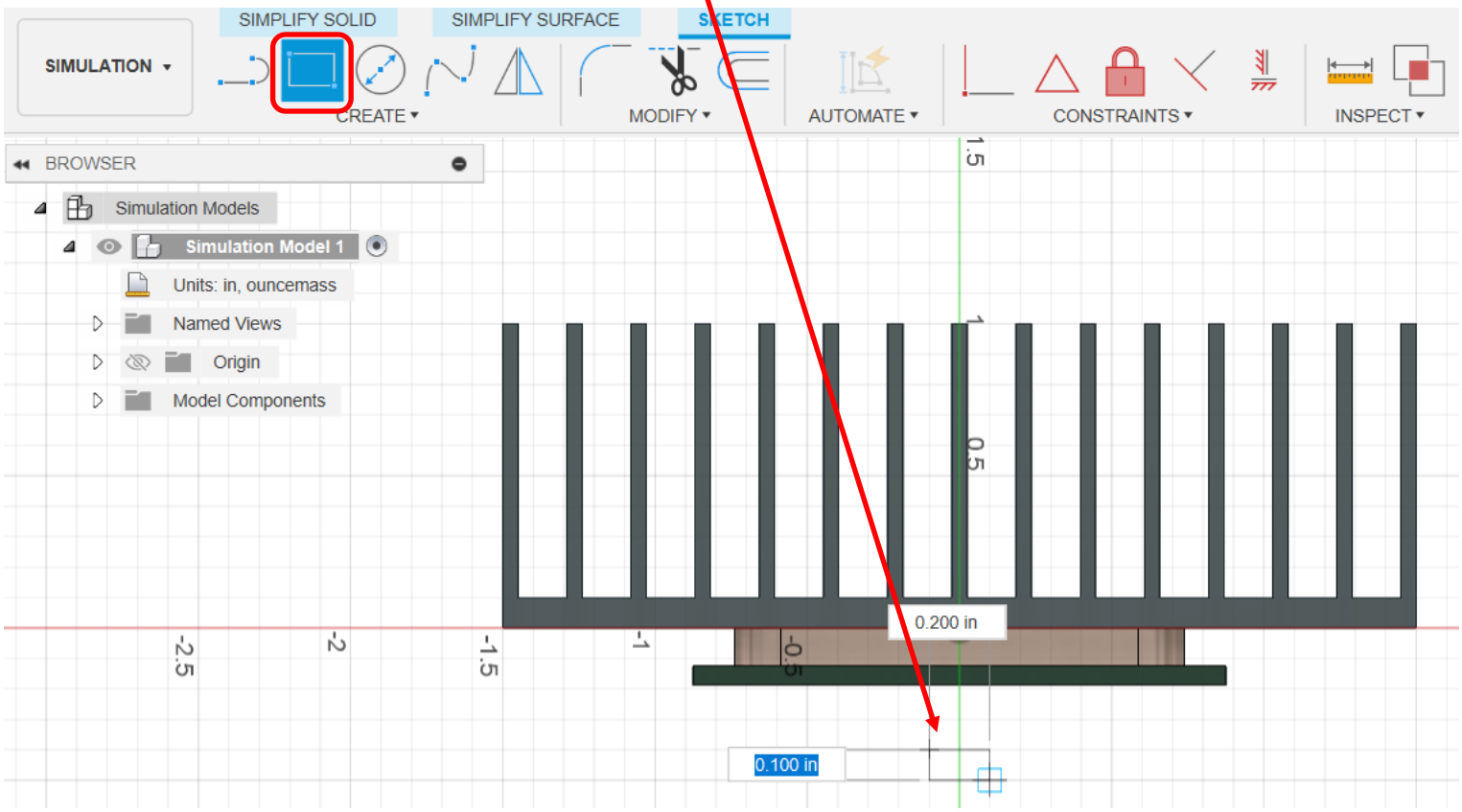
- from the top **SIMPLIFY** menu, select **Simplify**



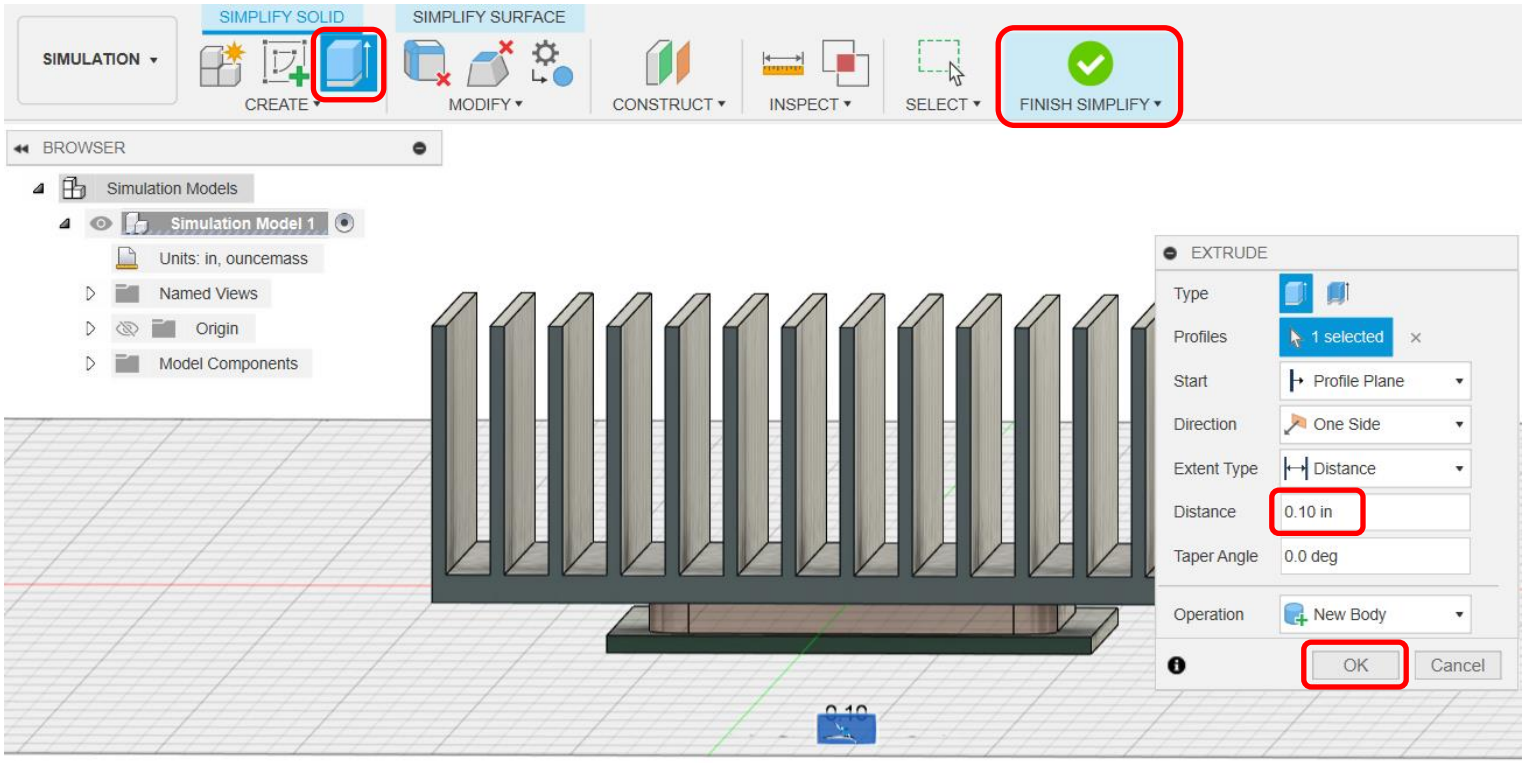
- select **Create Sketch** and click on the **front of the heat sink**



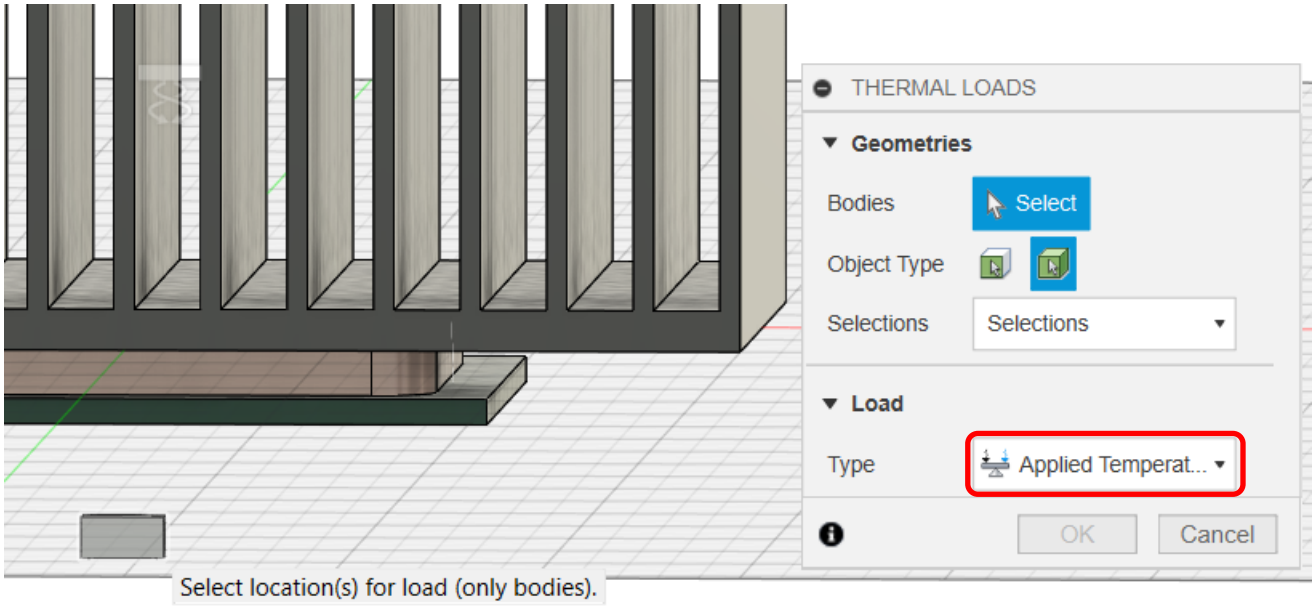
- select the **Rectangle** tool and create a **small rectangle under the PCB** that is **0.200 wide** by **0.100 high**. The position is not critical and neither are the dimensions. It is small so that many mesh elements won't be allocated to it.



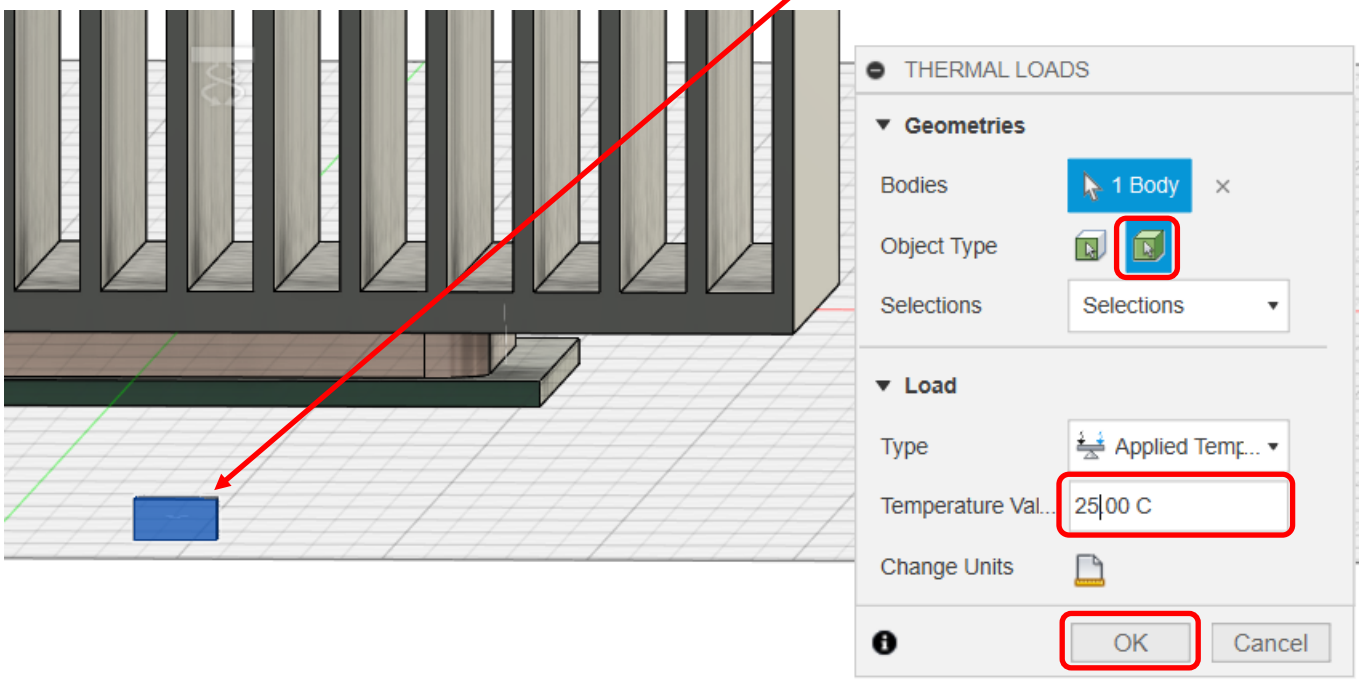
- select the **Extrude** tool
- if the small rectangle does not highlight blue, click on it
- set the **Distance** to **0.10**, and click **OK** and click on **FINISH SIMPLIFY**



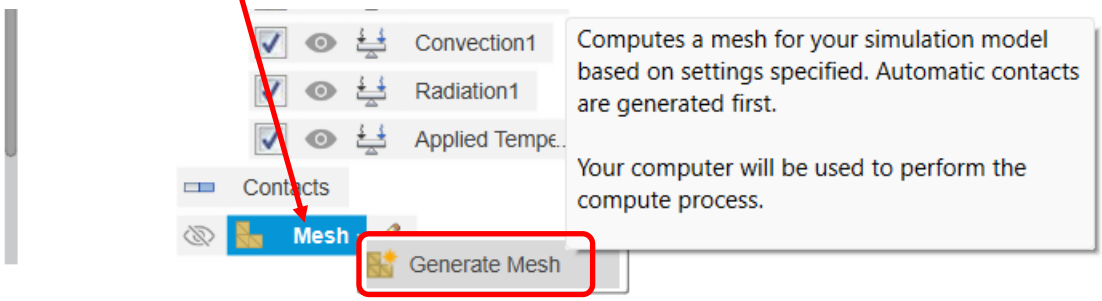
- from the **LOADS** menu select **Thermal Loads** and set the **Type** to **Applied Temperature**



- select the **Body** icon for **Object Type** and click on the **small block** that was just created
- set the **Temperature Value** to **25** and click **OK**

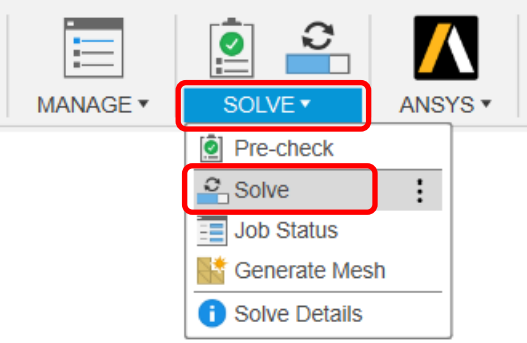


- right-click on **Mesh** and select **Generate Mesh**

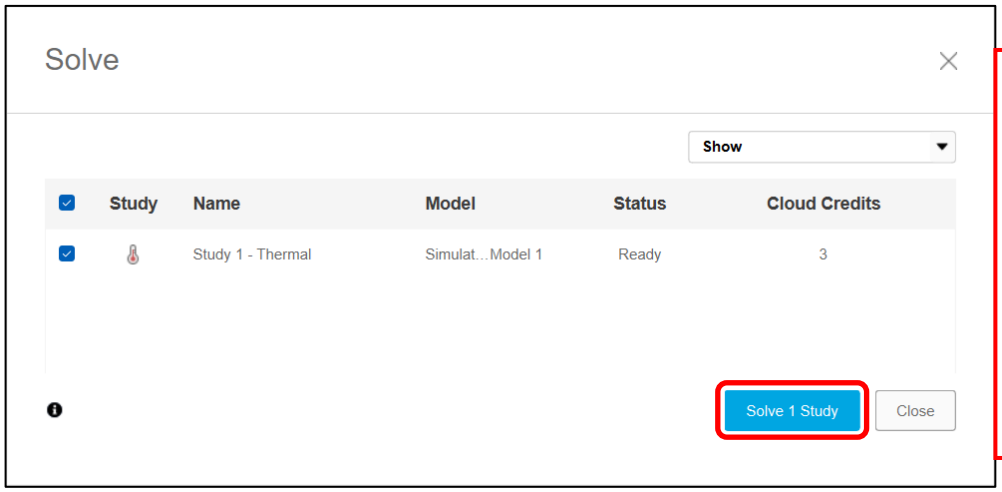


Solving the Simulation and Waiting, Waiting, Waiting…

- from the top **SOLVE** menu, select **Solve**



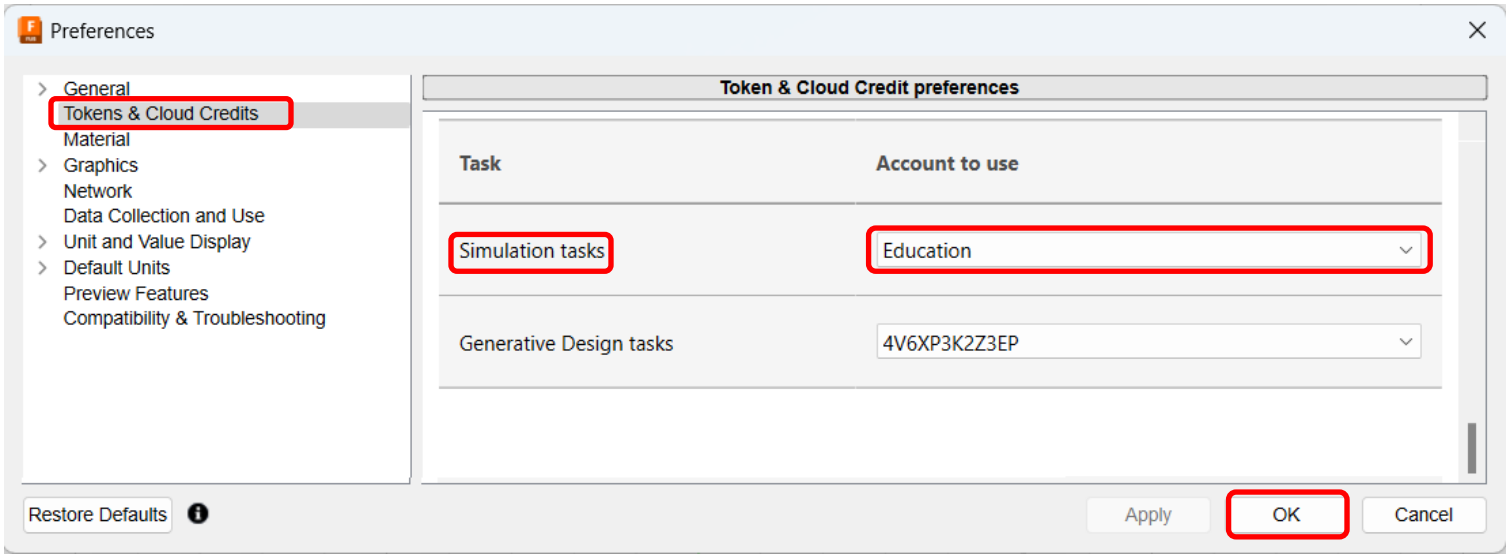
- click on the bottom **Solve 1 Study**. A simulation like this typically completes in about 2 minutes.



If a message about **Cloud Credits** opens, there are two options.

1) Transfer **V-Bucks** or **Robox** funds to Autodesk

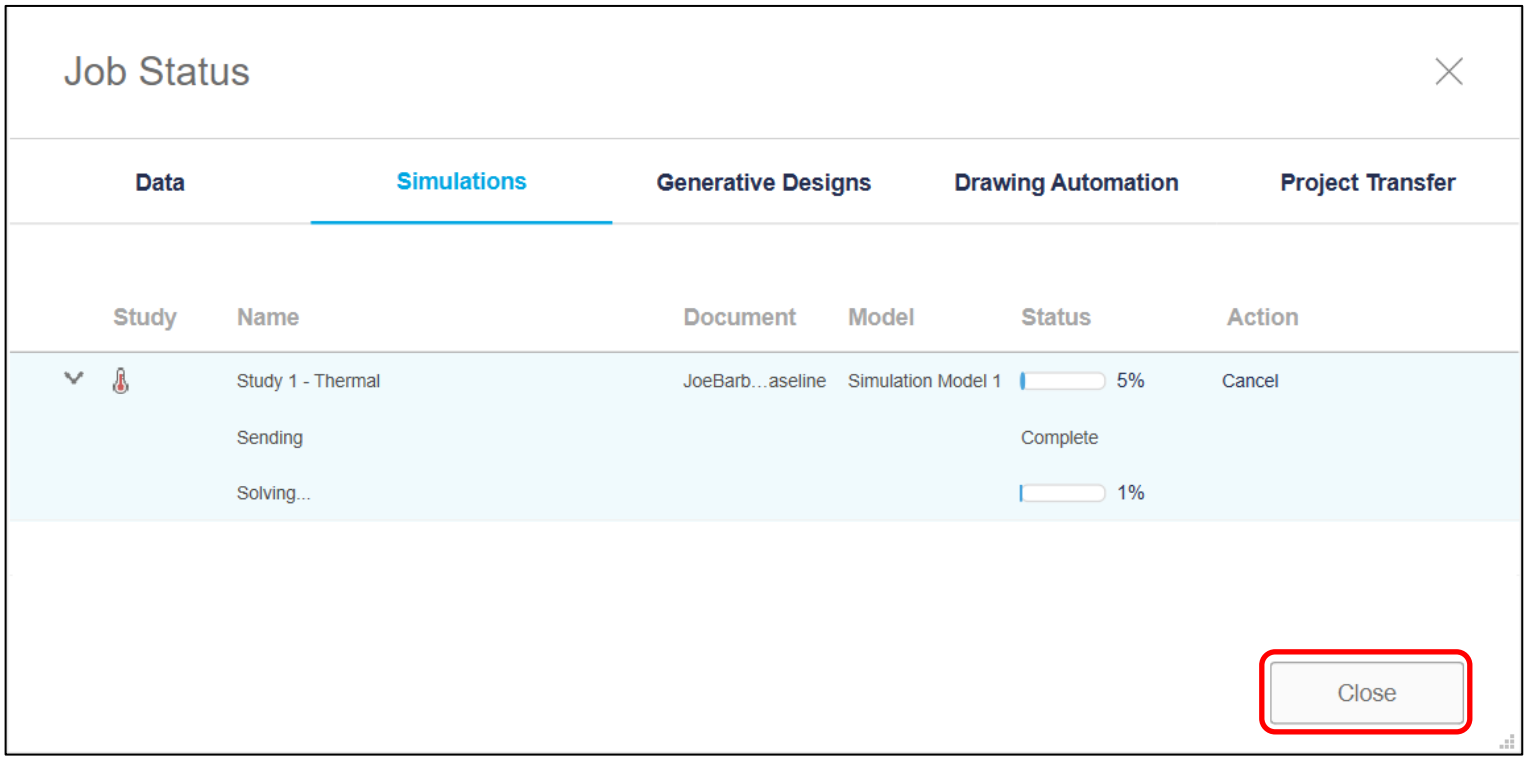
2) from the **top-right person icon of the Fusion screen**, select **Preferences**. Select **Tokens & Cloud Credits** on the left and for **Simulation tasks** select **Education**.



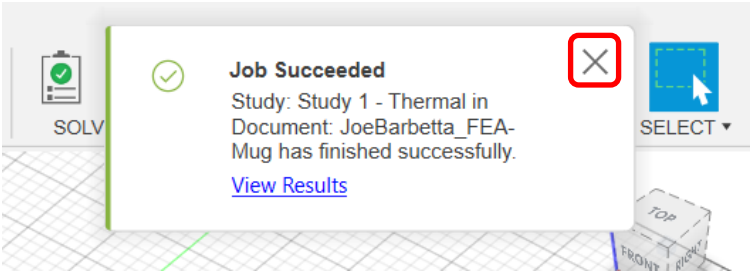
This screen will open to report the progress of the solve.

FEA simulations are computationally intensive and your project is sent to Autodesk servers in the “cloud”. The number crunching is performed on their servers.

- when the simulation is finished, click **Close**

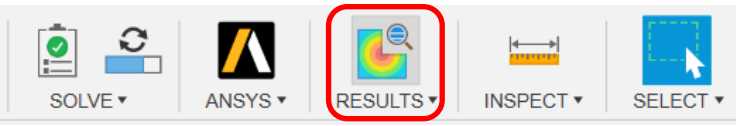


- when this message shows, click on the **X** to close it. Sometimes clicking on View Results does nothing.

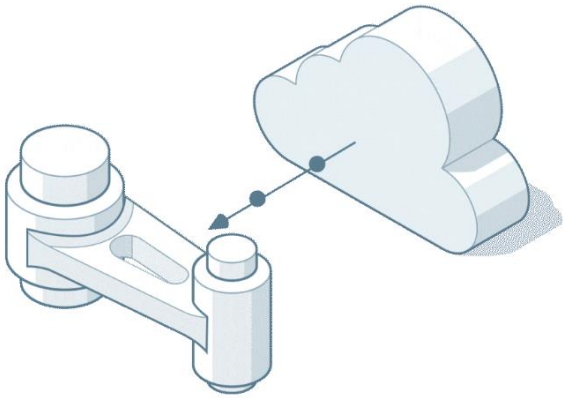


Viewing Simulation Results

- click on the **RESULTS** icon and wait for the animation, shown on the right, to complete



- enjoy the neat “data from the cloud” animation



Fetching your results...

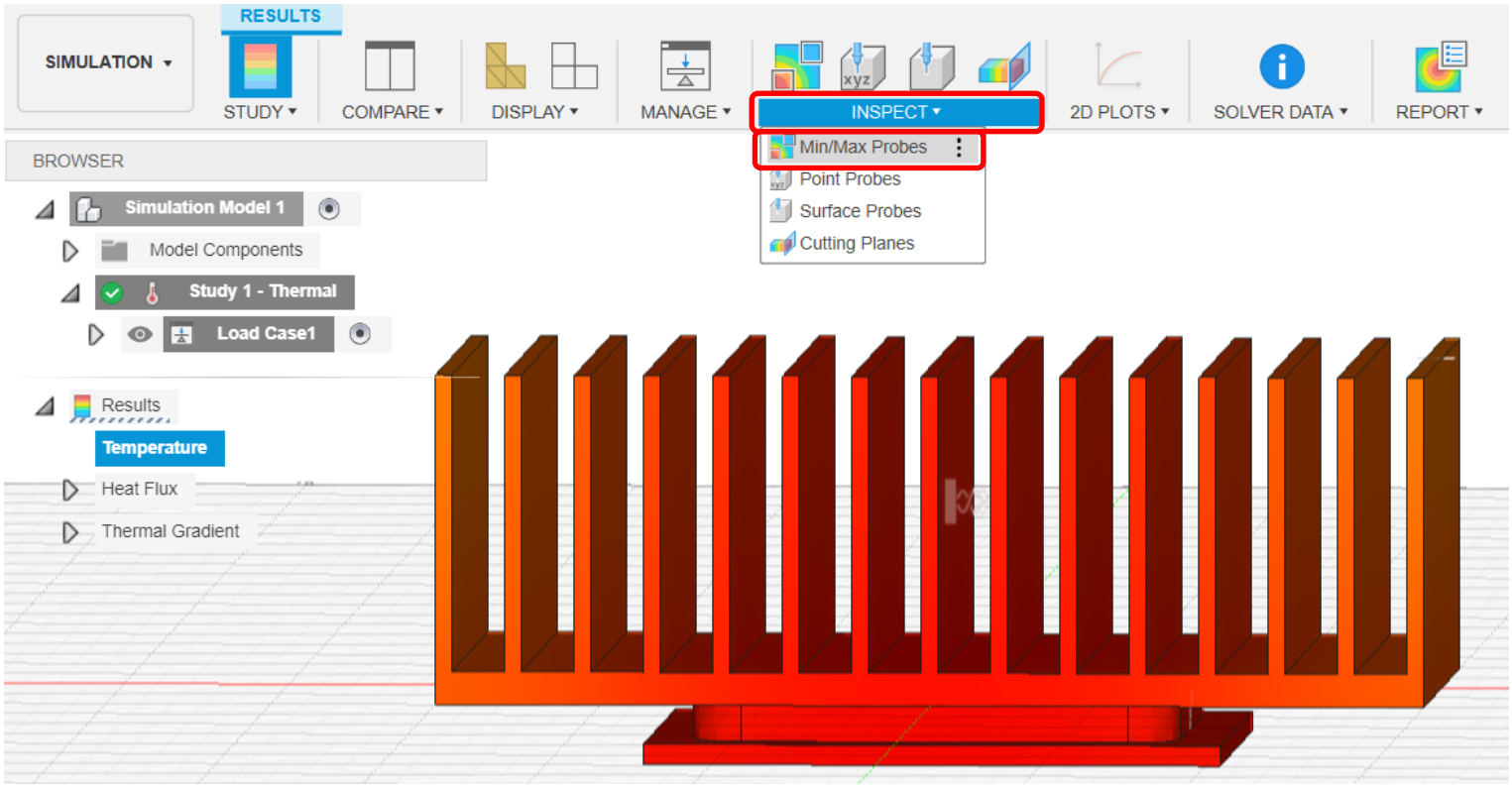
This may take a few moments.

Autodesk uses Amazon Web Services, which has a major data center in Virginia. Your simulation likely ran on a computer in this building and the results are sent in pulses of light over fiber optics to our school.

Amazon AWS Data Center in Ashburn Virginia

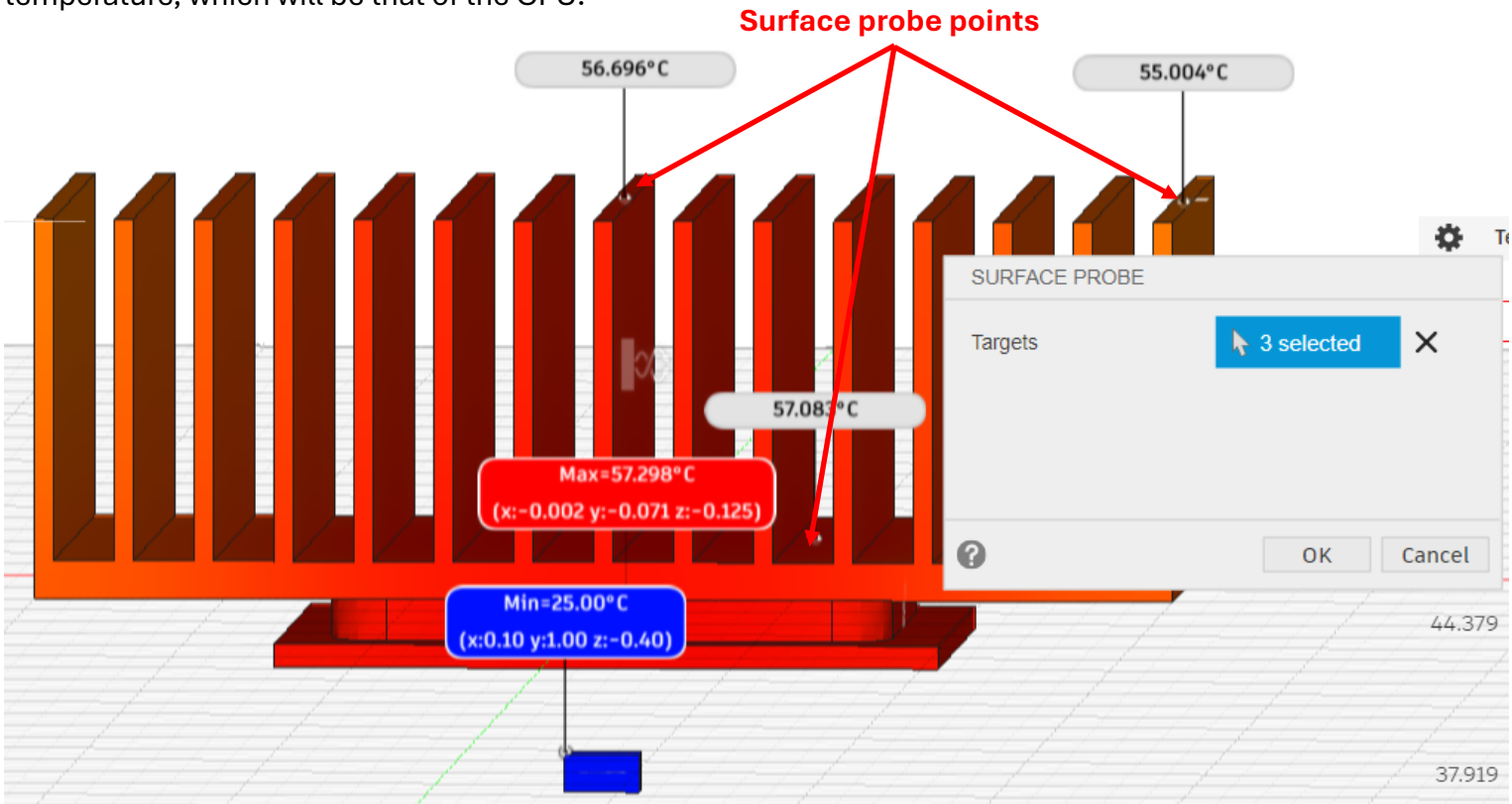
Visit ›

- from the **INSPECT** menu, select **Min/Max Probes**

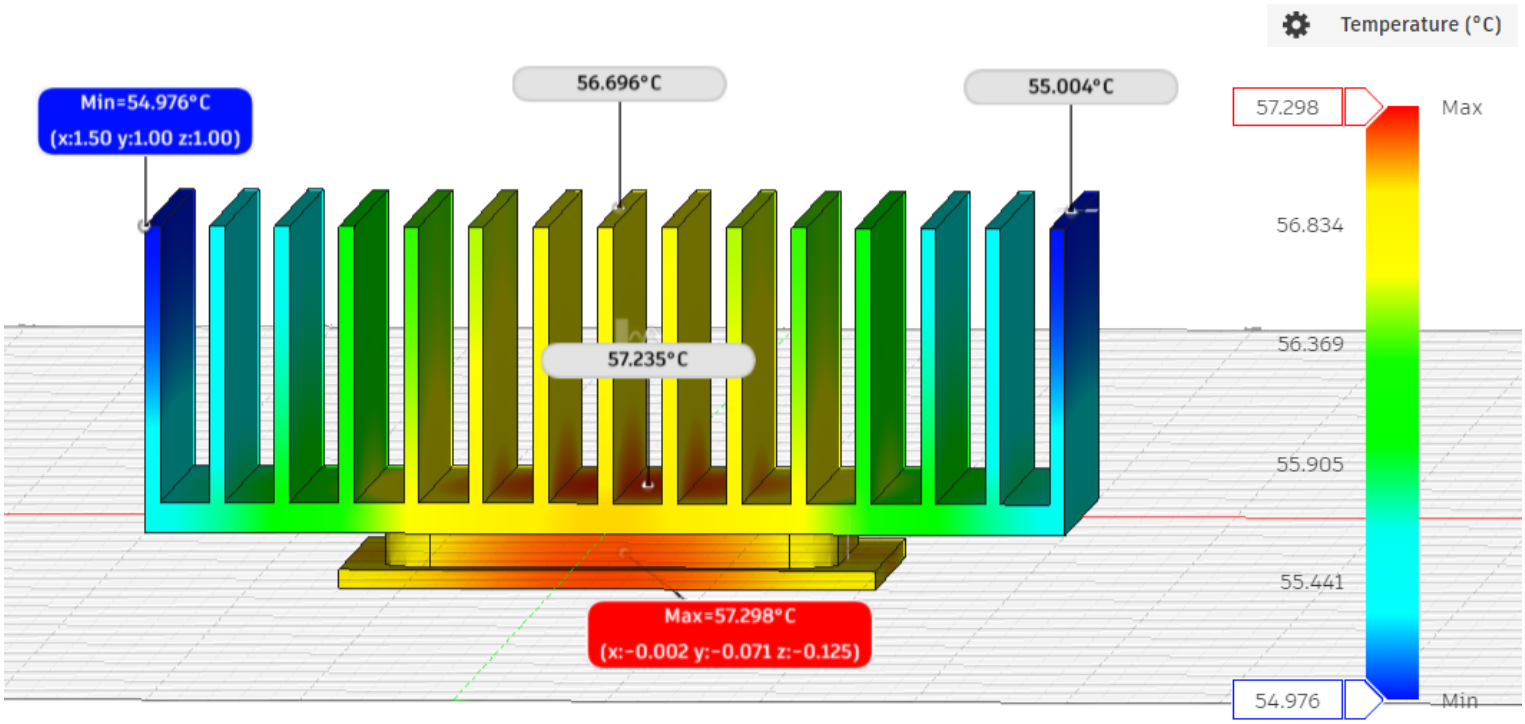


- from the **INSPECT** menu, select **Surface Probes**
- click on the **points** indicated and click **OK**. These points will provide the temperature of the **base above the CPU**, at the **top of the center fin**, and at the **top of a side fin**.

Note the Min and Max values. The Min will just be the ambient temperature. The Max will be the maximum temperature, which will be that of the CPU.



This is the result of the simulation without the ambient temperature block. This looks much more worthy of an art credit. However, it may lead one to think that the exterior fins are cold, whereas they only about 2 degrees cooler than the hotest temperature and would be very hot to the touch.

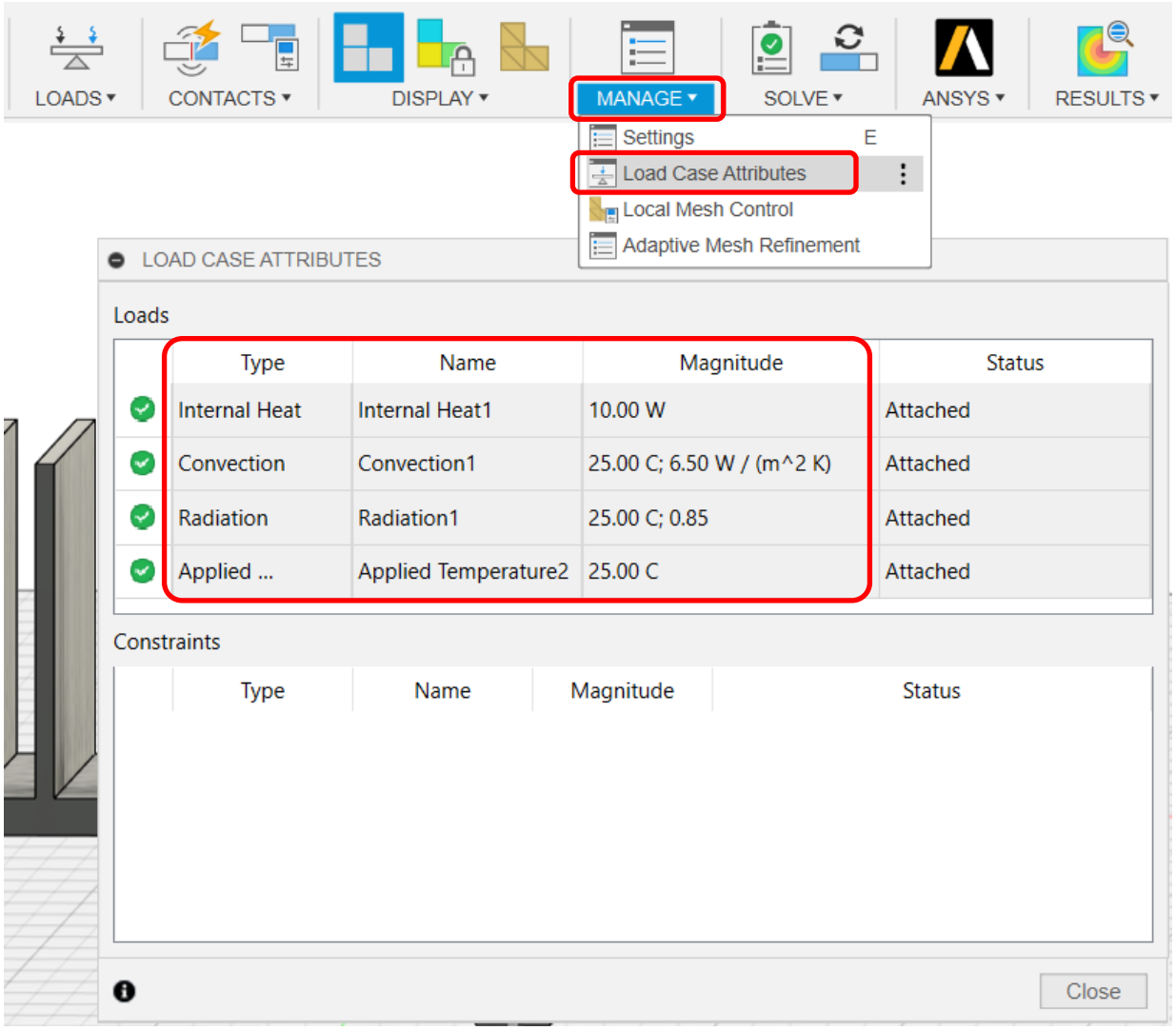


- click **Finish Results** in the upper right corner of the Fusion screen

Viewing Load Case Attributes

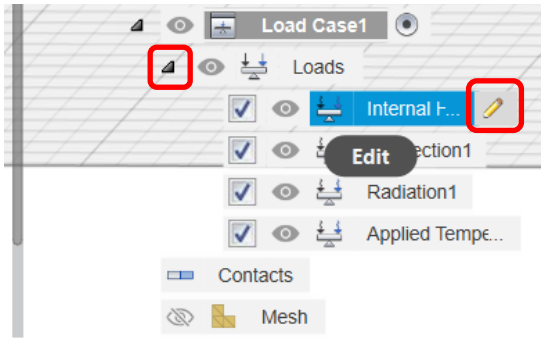
- from the **MANAGE** menu, select **Load Case Attributes**

This a convenient way to view the various Thermal Loads and including a screenshot in a report will document the termal loads used for a simulation.



- click on the **arrow** for **Loads**

One can uncheck boxes to suppress loads. However, only either Convection or Radiation can be suppresed without causing an error. One can also click the Edit icon (pencil) to change the settings.



Deliverables

1) screenshot of your simulation with both Convection and Radiational cooling

2) screenshot of your simulation with only Convection cooling

3) screenshot of your Load Case Attributes