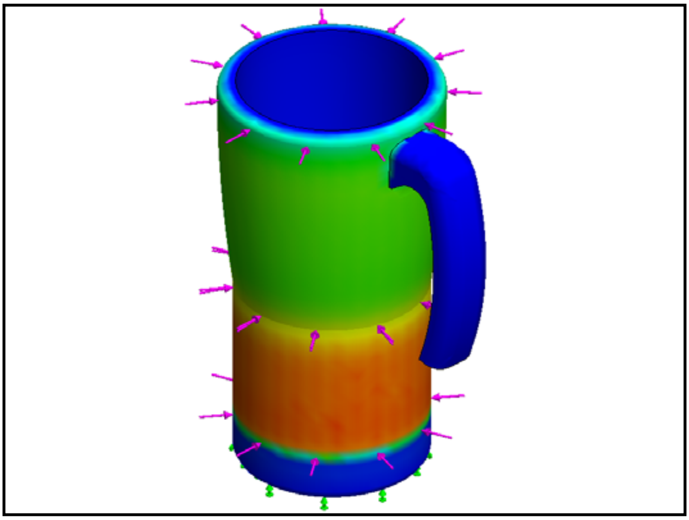
Engineering Design
and Technology Series

SolidWorks® Sustainable Design An Introduction to Material Choice and Sustainable Redesign



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Lesson 1
Introduction and Material Choice

When you complete this lesson, you will be able to:

- Describe the relationship between Sustainability and SolidWorks;
- Identify the principle components of the Sustainability Add-in;
- Explain significance of Material choice and environmental impacts.

Using This Book

SolidWorks Sustainability An Introduction to Sustainable Design helps you learn the principles of using Sustainability and Sustainability as integral parts of a creative and iterative design process.

For this project, You will “learn by doing” as you complete a structural analysis.

What is SolidWorks Software?

SolidWorks is design automation software. In SolidWorks, you sketch ideas and experiment with different designs to create 3D models using the easy to learn Windows® graphical user interface.
SolidWorks is used by students, designers, engineers and other professionals to produce single and complex parts, assemblies and drawings.

Prerequisites

Before you begin the SolidWorks Sustainability An Introduction to Sustainable Design you should complete the following online tutorials that are integrated in the SolidWorks software:

- Lesson 1 - Parts-Set 1
- Lesson 2 - Assemblies-Set 1
- Designing for Sustainability-Set 2
- Simulation - Static Analysis
- Simulation - Thermal Analysis

You can access the online tutorials by clicking **Help**, **SolidWorks Tutorials, All SolidWorks Tutorials (Set 1)** and Simulation Tutorials by clicking **Help, SolidWorks Simulation, Tutorials**. The online tutorial resizes the SolidWorks window and runs beside it.

As an alternative, you can complete the following lessons from *An Introduction to Engineering Design With SolidWorks*:

- Lesson 1: Using the Interface
- Lesson 2: Basic Functionality
- Lesson 3: The 40-Minute Running Start
- Lesson 4: Assembly Basics
- Lesson 6: Drawing Basics

Conventions Used in This Book

This manual uses the following typographical conventions:

Convention	Meaning
Bold Sans Serif	SolidWorks commands and options appear in this style. For example, **Insert, Boss** means choose the **Boss** option from the **Insert** menu.
`Typewriter`	Feature names and file names appear in this style. For example, `Sketch1`.
17 Do this step.	The steps in the lessons are numbered in sans serif bold.

Sustainability Options

Here we will go through the Sustainability interface and different menus as well as define various terms used within the SolidWorks Add-In. There are four main menus, **Material**, **Manufacturing**, **Transportation and Use**, and **Environmental Impact**.

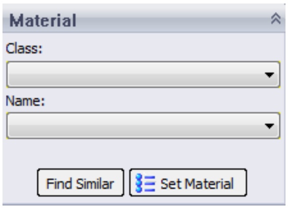
First, we will start SustainabilityXpress.

1 Start Sustainability.
Click **Tools, Add-Ins, Check Sustainability**.

Note: A Part or Assembly needs to be open to view Sustainability. When you first open the Add-In, everything should be black except for the regions.

Materials

In this option you can choose between different materials for the specific part using the drop down menus. You are also able to search for alternative materials using the **Find Similar** option. You can also assign a material of your choice to the part.



Manufacturing (Parts)

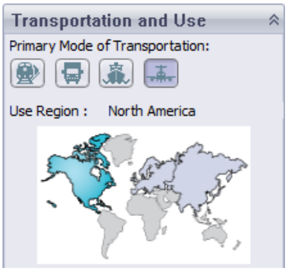
The **Manufacturing** section includes **Process** and **Use** to define world locations.

Process

In this option, there is a drop down menu labeled **Process** where the user can chose between multiple different production techniques to manufacture their part. There is also a world map. The world map is for the user to define where the part is going to be made. There are four different areas to choose from, North America, Europe, Asia, and Japan.

Use

The second world map is used in this menu. Here you are able to chose where your product will be transported to after production. The further the distance between manufacturer and user the less environmentally friendly.



Note: All the regions for Manufacturing and Use are the same.

Manufacturing and Transportation (Assemblies)

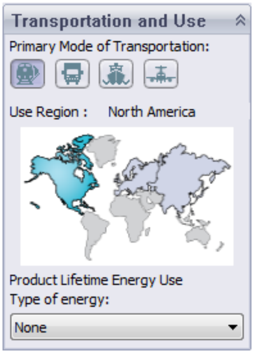
Within Assemblies the Sustainability interface changes slightly.

Manufacturing (Assemblies)

The only difference from the manufacturing menu for a part is that it does not have a process drop down menu instead the user is only able to choose the Manufacturing Region.

Transportation and Use (Assemblies)

With in this menu the user is given the ability to chose the Primary Mode of Transportation (Train, Truck, Boat, or Plane). The user is also able to choose the Type of Energy that will be used throughout the lifetime of the product. Like before in the Use menu for Parts the user is also able to choose the Region the that product will be used.

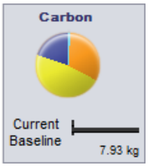


Environmental Impact

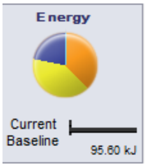
This area includes four quantities: **Carbon Footprint**, **Total Energy**, **Air Acidification**, and **Water Eutrophication**. Each graph shows the user a graphic breakdown of **Material Impact**, **Transportation and Use**, **Manufacturing**, and **End of Life**.

Carbon Footprint

A measure of carbon-dioxide and other greenhouse gas emissions such as methane (in CO2 equivalent units, CO2e) which contribute to an emissions, predominantly caused by burning fossil fuels. Global warming Potential (GWP) is also commonly referred to as a carbon footprint.

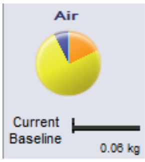


Energy Consumption



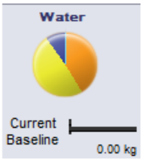
A measure of the non-renewable energy sources associated with the part's life cycle in nits of mega joules (MJ). This impact includes not only the electricity or fuels used during the product's life cycle, but also the upstream energy required to obtain and process these fuels, and the embodied energy of materials which would be released if burned. Energy Consumed is expressed as the net calorific value or energy demand from non-renewable resources (e.g. petroleum, natural gas, etc.). Efficiencies in energy conversion (e.g. power, heat, steam, etc.) are taken into account.

Air Acidification



Sulfur dioxide, nitrous oxides other acidic emissions to air cause an increase in the acidity of rain water, which in turn acidifies lakes and soil. These acids can make the land and water toxic for plants and aquatic life. Acid rain can also slowly dissolve man-made building materials such as concrete. This impact is typically measured in nits of either kg sulfur dioxide equivalent (SO2e), or moles H+ equivalent.

Water Eutrophication



When an over abundance of nutrients are added to a water ecosystem, eutrophication occurs. nitrogen and phosphorous from waste water and agricultural fertilizers causes an overabundance of algae to bloom, which then depletes the water of oxygen and results in the death of both plant and animal life. This impact is typically measured in either kg phosphate equivalent (PO4e) or kg nitrogen (N) equivalent.

Report

On the very bottom of SustainabilityXpress, there are the **Generate Report** and **Email Report** buttons. By clicking generate report, SolidWorks automatically creates a Word document about the current analysis. This analysis can be on an individual material type and environmental impacts or it can be on a comparison of two different material types. The email report opens Microsoft Outlook for the user to send the word document to an email address.

Baseline

To the right of the report buttons are the **Set Baseline** and **Import Baseline** buttons. By clicking set baseline, SustainabilityXpress automatically takes the most recent material type and sets it as the material that every other material will be compared to. Otherwise, every time the user clicks on another material, SustainabilityXpress will automatically compare them and dynamically recalculate the Environmental Impacts. Also, if there is no difference between the current and previous settings and materials then all of the Environmental Impacts will automatically turn green. Then, by clicking import baseline, the user can import a saved SustainabilityXpress baseline from another part.

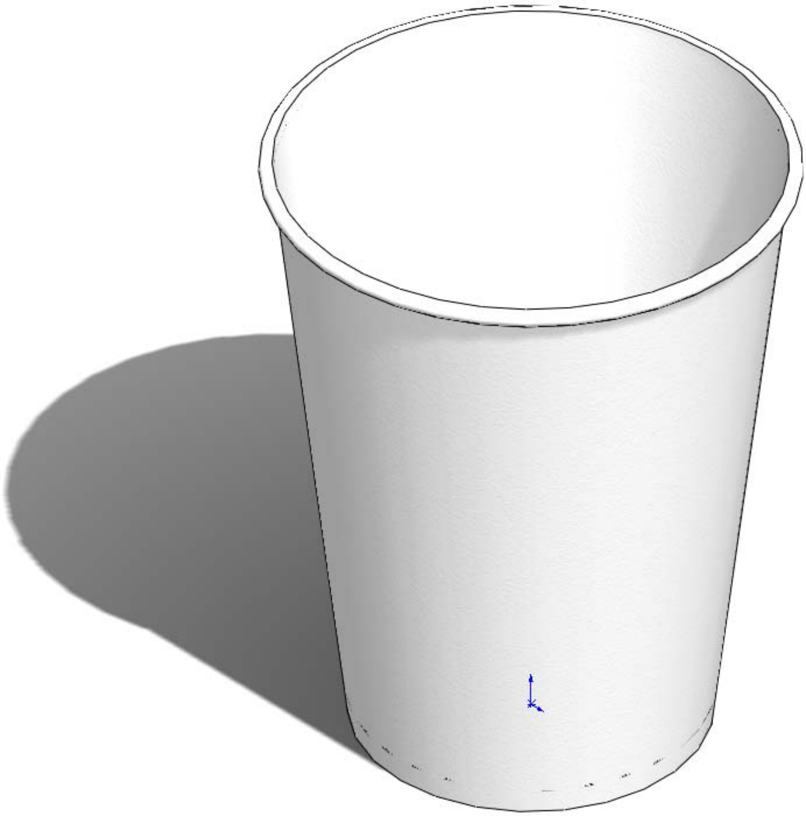
Color Coding

When Baseline is clicked, the environmental impacts turn colors to represent different states.

- *Black* represents the baseline material.
- *Green* indicates that the current material is more environmentally friendly than the baseline material.
- *Red* indicates that the current material is less environmentally friendly than the baseline material.

Material Choice in Sustainable Design

Here we will decide which material is the correct material to use depending of the materials environmental impacts over its lifetime. In this example, imagine the analysis of a cup.



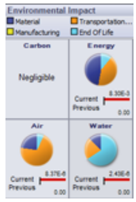
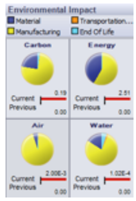
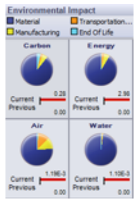
The lifetime of a cup

The material of a product significantly affects its lifetime. For example, a cup could be made from paper, plastic, or metal. Depending on what material we use will decide how many times the cup can be used. For this example we will assume if we made the cup out of paper (we will be using pine because there is no paper material choice within SolidWorks that is linked with Sustainability) it could be used only once, a cup made from plastic can be used 10 times, and a cup made from metal could be used 1000 times.

Environmental Impacts

With the SolidWorks model we have for a simple cup, we have created three different configurations, one for each material type. We activated Sustainability and kept the Manufacturing and Transportation and Use the same continents for all three configurations.

Here are the Environmental Impacts for each material:

Paper (Pine)	
Plastic (ABS)	
Metal (Chrome Stainless Steel)	

From these we will use the Total Energy as a baseline to examine which material is the most Sustainable for its lifetime. The results were as follows:

Paper: 8.30E-3 MJ

Plastic: 2.51 MJ

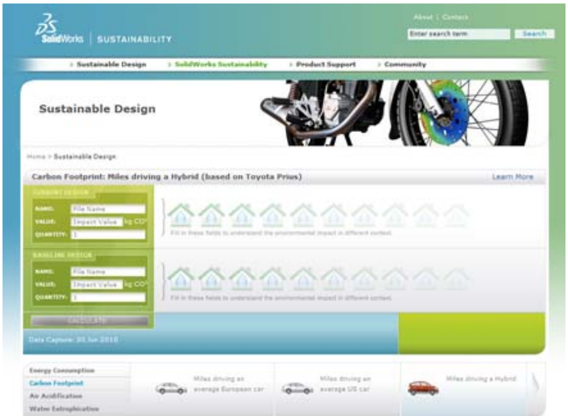
Metal: 2.98 MJ

2 Sustainability Calculator.

Now we need to see which material is the most environmentally friendly based on its lifetime. The **Sustainability Calculator** takes the values that we found for the Environmental Impacts (CO2, MJ, SO2, and PO4) and re calculates it into something that is easier for us to understand (example: miles driven in a car or hours watching tv).

To start, we will open the Sustainability Calculator.

3 Go to <u>www.solidworks.com/sustainability/products/calculator/index.htm</u>.



4 Click Energy Consumption.

5 Click hours of watching TV.

Impact vs. Lifetime

Here we will discuss whether the Lifetime of the material is more important than its Environmental Impacts.

Using the Sustainability Calculator we will use the three Energy values we got from SolidWorks Sustainability and calculate which material is the best for the environment according to its lifetime.

In order to do this we need to use the Lifetime values we discussed earlier. Instead of using the number of times each cup can be used we will use the number of cups that need to be made to equal one Metal cup. This means 1000 paper cups and 10 plastic cups need to be made to equal one Metal cup.

1 Sustainability calculator.
Now, go back to the Sustainability Calculator and find the Current Design box.

2 Enter numbers.
Enter the numbers given for the Values and Quantities and click Calculate.

Note: You are only able to enter one set two sets of values. It would be easier to open three se per ate windows and compare the outcomes.

Name: **Paper**

Value: 8.30E-3

Quantity: 1000

Name: **Plastic**

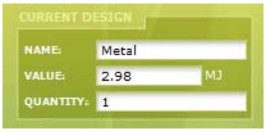
Value: 2.51

Quantity: 10

Name: **Metal**

Value: 2.98

Quantity: 1



The Sustainability Calculator will calculate how many Hours of Watching TV is equivalent to produce these cups. You should get:

Paper: 2 Hours

Plastic: 6 Hours

Metal: 1 Hour

3 Material Decision.
When comparing materials according to their lifetimes and environmental impacts it is wise to chose the material that effects the environment the least compared to how long its lifetime is. In this case, it is wise to chose the Metal Cup. The Metal Cup can be used the longest and when compared to the Paper and Plastic Cups it the least harmful to the Environment based on the cups Energy Consumption.

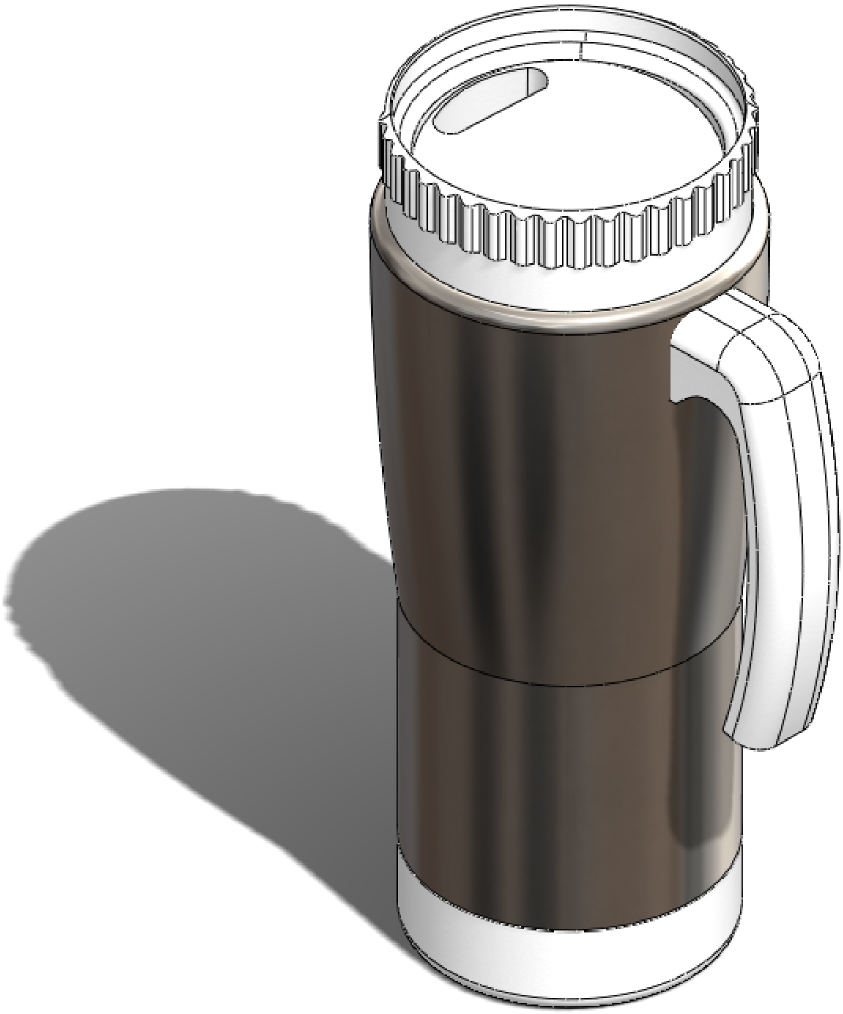
Lesson 2
Sustainability and Simulation

When you complete this lesson, you will be able to:

- Add-In Simulation;
- Activate Sustainability;
- Mate Parts in an Assembly;
- Create Static Studies;
- Create a Thermal Study;
- Edit individual Parts;
- Evaluate Sustainability throughout Redesign.

Using Simulation

SolidWorks Simulation lets you test products for defects before they're built, helping prevent errors early in the design process. It's powerful enough for trained FEA analysts, yet easy enough for product designers. SolidWorks Simulation can even help you optimize your designs for maximum performance and savings.



How to Activate Simulation with Sustainability

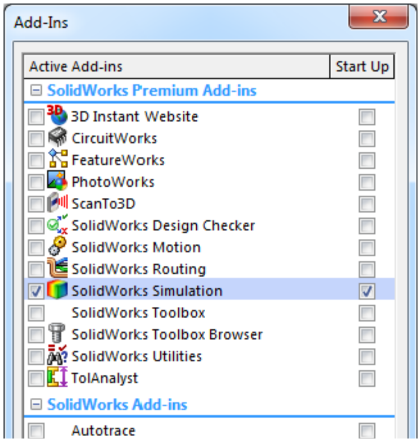
This section will show you how to activate SolidWorks Simulation and Sustainability.

Activating Simulation and Sustainability

1 Open Assembly.
Open Mug.

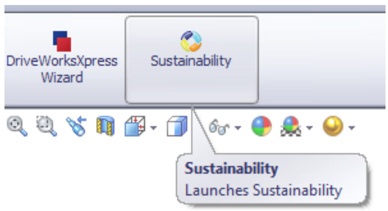
2 Activating the Add-In.
Click **Tools, Add-Ins**.

Click both checkmarks of **SolidWorks Simulation** as shown.



Note: By selecting the check mark to the right of SolidWorks Simulation you are telling SolidWorks to activate SolidWorks Simulation every time you open SolidWorks.

3 Open sustainability.
Click the **Evaluate** tab on the top left of the screen.



Select the **Sustainability** icon.

A **Life Cycle Assessment (LCA)** window will appear. Click **Continue.**

4 Pin sustainability.
Click the pin in the top right corner so it looks like 

This docks the Sustainability Task Pane so it is always on the screen.

5 Sustainability for original design.
Once Sustainability is open it will automatically calculate the Environmental Impacts for the assembly.

Note: If parts of the assembly do not have assigned materials to them, then Sustainability will prompt the user to select a material for the part or to exclude the part when calculating the Environmental Impacts.

Within the Sustainability menu under <u>Transportation and Use</u> select **Boat** under <u>Primary Mode of Transportation</u>.

Under <u>Type of Energy</u> leave it set to none.

Note: After each change made in the Sustainability Menu the Environmental Impacts will automatically update.

6 BaseLine.

Click the **Set Baseline** .

By setting the baseline, every time you make a change to the assembly and/or a change the Sustainability Menu the Environmental Impacts will update and compare the new Impacts to the Baseline we set.

Mating Assembly

Here we will mate the base and Metal Outside to the rest of the mug.

Remating Metal Outside

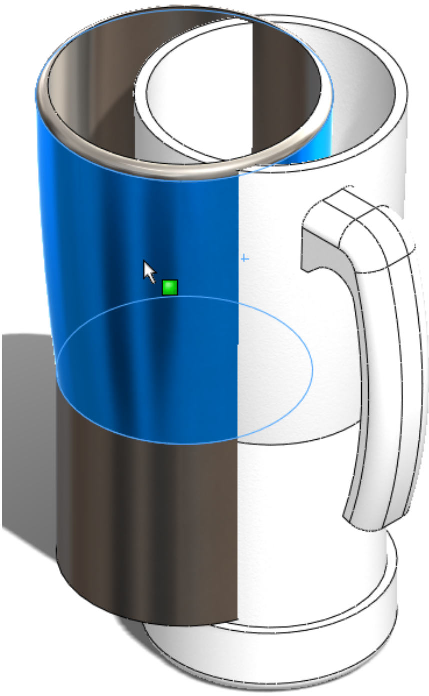
1 **Suppress Cover.**
Before starting we will suppress the Cover. Within our redesign we will not be altering the Cover.

Right-click Cover in the FeatureManager Design Tree.

Select the **Suppress**.

Note: By suppressing the part it fully takes the part out of the assembly. If any simulation is run the Cover will not be included in the mesh and if sustainability is run it will not be included in the environmental impacts.

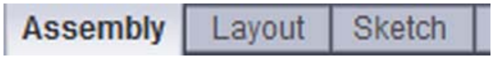
2 **Move Metal Shell.**
Click and drag Metal_Outside to the left slightly as shown.



Note: By dragging it to the left we will be making it easier for us to see the mates we are selecting.

3 Mate `Metal_Outside` and `Plastic_Inside`.

Select the **Assembly** tab at the top left of the screen.

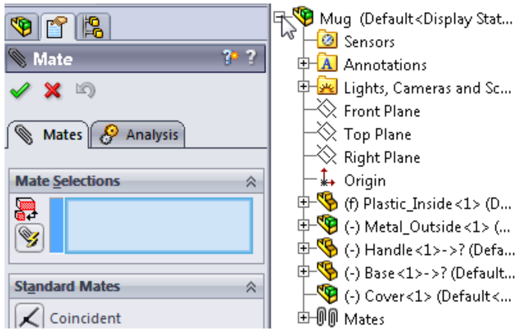


Click **Mate**.

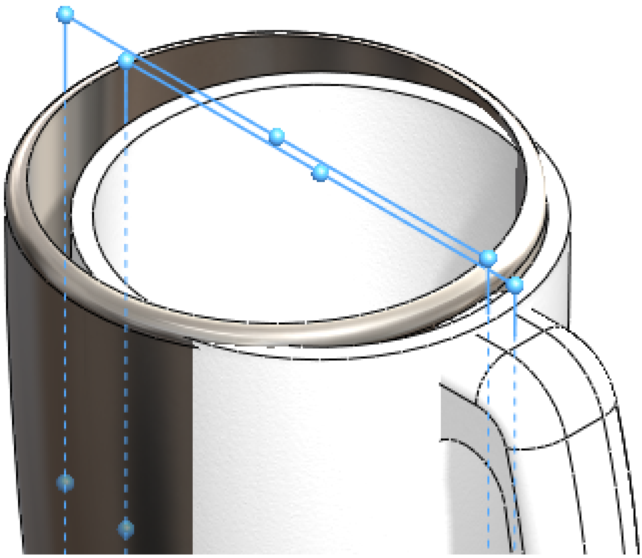
Expand the Design Tree.

Now expand both Plastic_Inside and Metal_Outside.

Click **Front Plane** in both Plastic_Inside and Metal_Outside.



Click  on the tool bar that appears.



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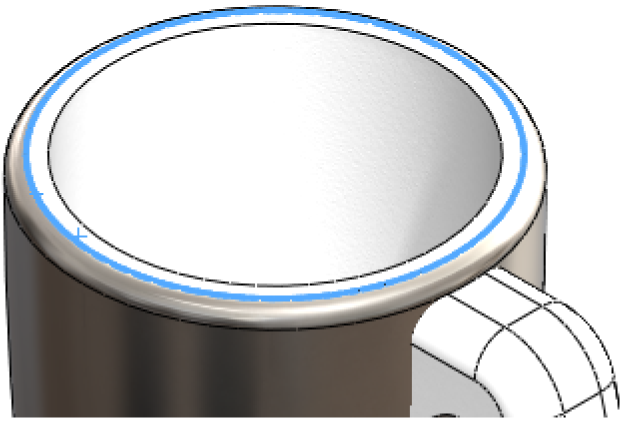
Select the **outside circle** of the Plastic_Inside and the **inside circle** of the Metal_Outside.



Click  on the tool bar that appears.

You have successfully mated the Metal_ Outside and the Inside_Plastic together.

Click **Save**.

Analysis of Insulation

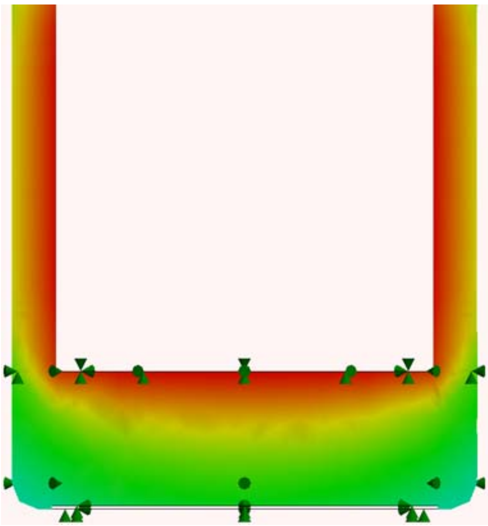
Here we will present why we chose to use a plastic inside and Metal Outside.

What makes good Thermal Insulator?

The term thermal insulation can refer to materials used to reduce the rate of heat transfer, or the methods and processes used to reduce heat transfer. Heat energy can be transferred by conduction, convection, radiation or by actual movement of material from one location to another. Thermal insulation is the method of preventing heat from escaping a container or from entering the container. Insulators are used to minimize that transfer of heat energy. In home insulation, the R-value is an indication of how well a material insulates. The flow of heat can be reduced by addressing one or more of these mechanisms and is dependent on the physical properties of the material employed to do this.

Plastic

Plastic is a good thermal insulator, it minimizes the transfer of heat very well as show.

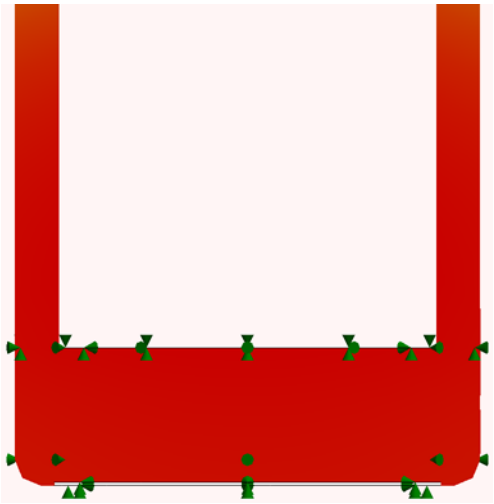


Also, plastic is significantly better for the environment than metal is.

Note: During this Tutorial we will not assume that this part will be recycled. We are basing our redesign on the calculated Environmental Impacts within SolidWorks.

Metal

Metal has some opposite qualities to plastic. Heat is transferred through metal very easily as shown.



Metal is not very environmentally friendly according to the Environmental Impacts within SolidWorks.

On the other hand, because metal has a high thermal conductivity it is good for spreading out the heat.

Also, metal is cheaper than strong plastics like ABS and has a nicer finish.

Plastic and Metal

The mug will consist of an outer layer and an inner layer. Because plastic is such a good insulator, plastic will be used as the inner layer for the mug as well as the handle. This way, the heat will be kept in with the plastic and the heat that does pass through the plastic will be distributed throughout the metal because metal has such a high thermal conductivity.

Static Simulation

Here we will start our redesigning process. We will perform two se per ate Static Simulations. One will be for the outside walls of Metal_Outside and the other will be for the Handle.

Static Study 1

1 Create Study.
Click the Simulation tab.

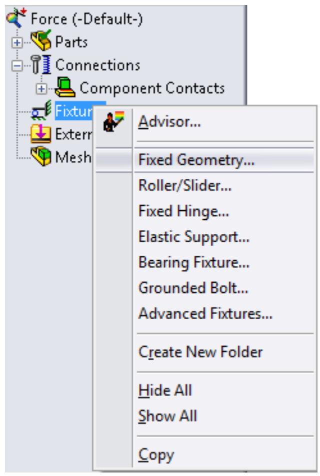
Select the down arrow under Study Advisor.

Select **New Study**.

First name the analysis **Force 1**.

Under Type select Static and Click .

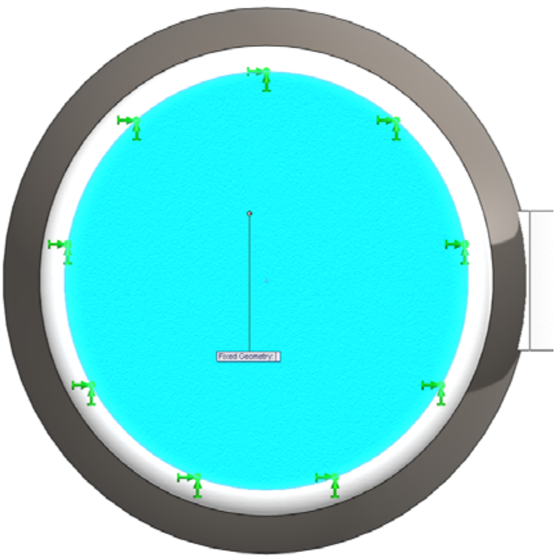
2 Fixtures.
Within the Static Study Design Tree right-click **Fixtures** and Select **Fixed Geometry**.



Change the view to Bottom by pressing the **SpaceBar**.

A dialog box will appear. double-click ***Bottom**.

Select the large inner face as shown.



Click .

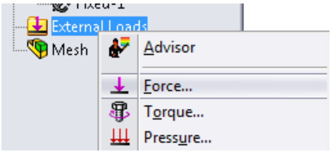
3 External Loads.

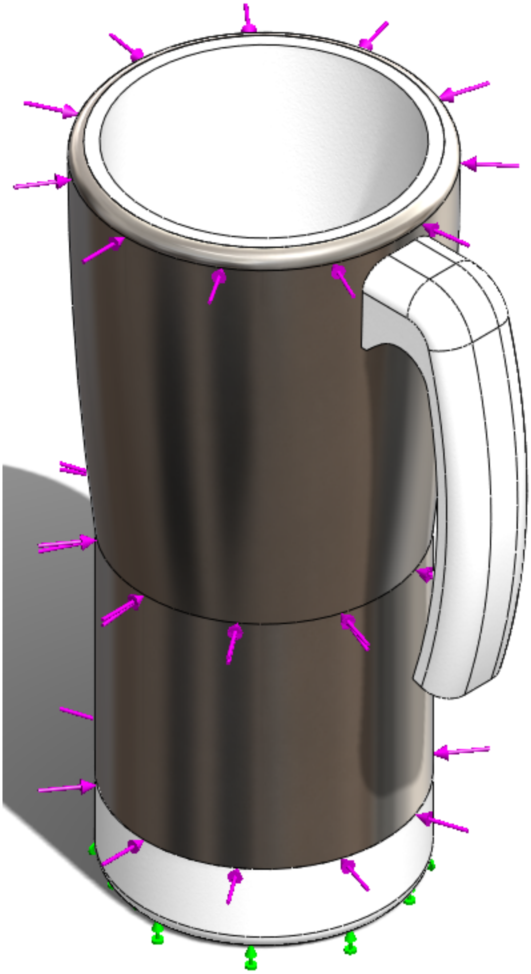
Press **space bar** and double-click **Isometric**.

Right-click **External Load** in the Static Study Design Tree.

Select **Force** as shown.

Select the 2 round faces of the Metal_Outside.





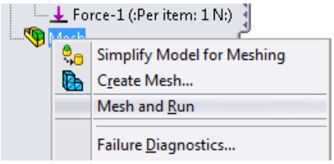
Change the Force Value to **400 N.**

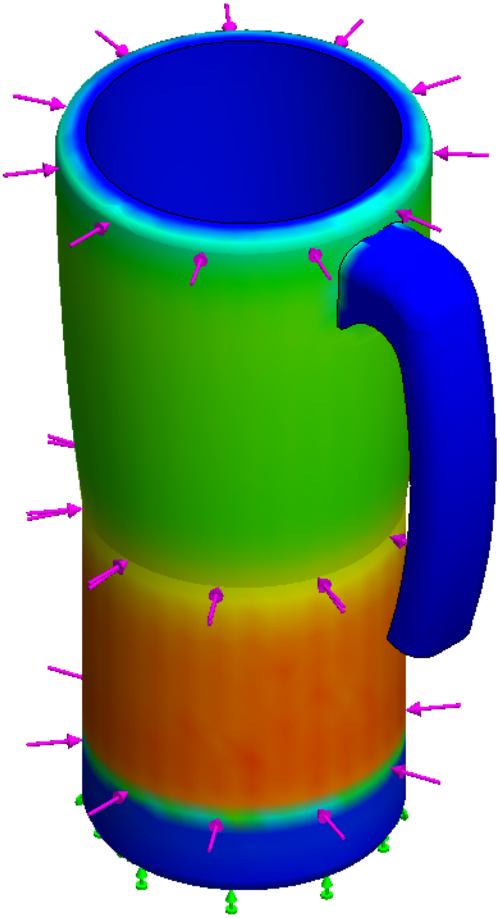
Click 

4 Running Simulation.

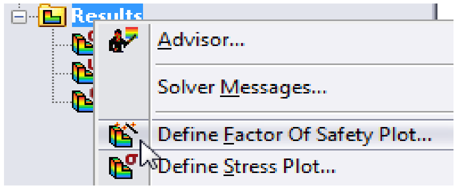
Right click **Mesh** and Select **Mesh and Run**.

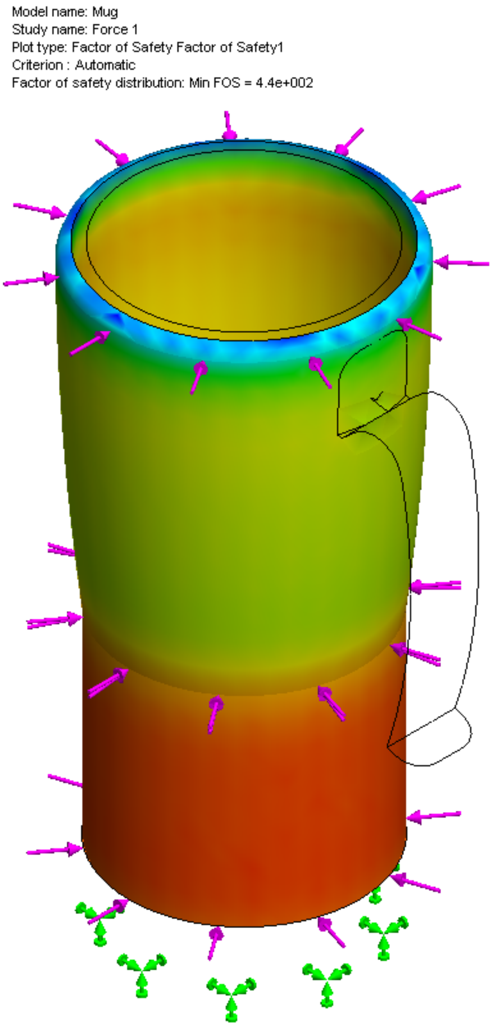
Two dialog boxes will appear. One will Mesh the assembly and the other will Run the Static Study.





5 Factor of Safety.
Right-click **Results.** Select **Define Factor Of Safety Plot.** Leave all settings the same and click . A simulation box will appear click **OK**.





In the top left corner of the screen SolidWorks tells us the **Factor of Safety** is 4.4e+002 or 440.

Click **Save**.

Static Study 2

Now we will run a similar Static Study with the same amount of Force on the `Handle` to see what the **Factor of Safety** is.

1 Create Study.
Click the `Simulation` tab if not already displaying it.

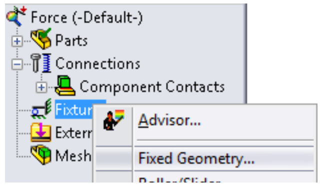
Select the down arrow under `Study Advisor`.

Select **New Study**.

First name the analysis **Force 2**.

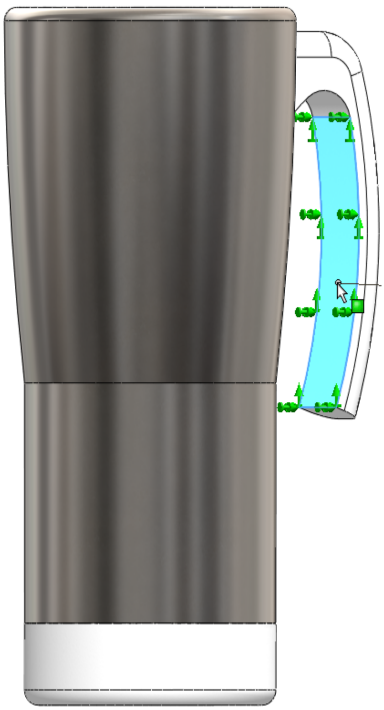
Under Type select Static and Click ✓.

2 Fixtures.
Within the Static Study Design Tree right-click **Fixtures** and Select **Fixed Geometry**.



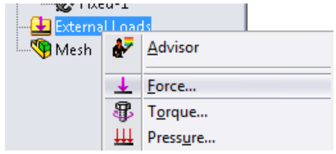
Press the **SpaceBar**. Double-click **Handle Fixture**.

Select the inner face of the handle.



Click ✓.

3 External Loads.

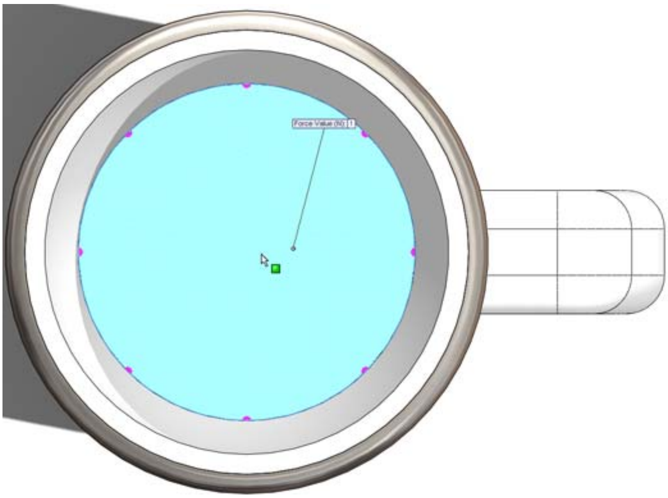


Right-click **External Load** in the Static Study Design Tree.

Select **Force**. Press **space bar**.

Double-click **Top**.

Select the inner bottom surface of the Plastic_Inside.

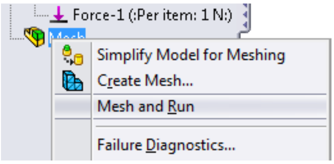


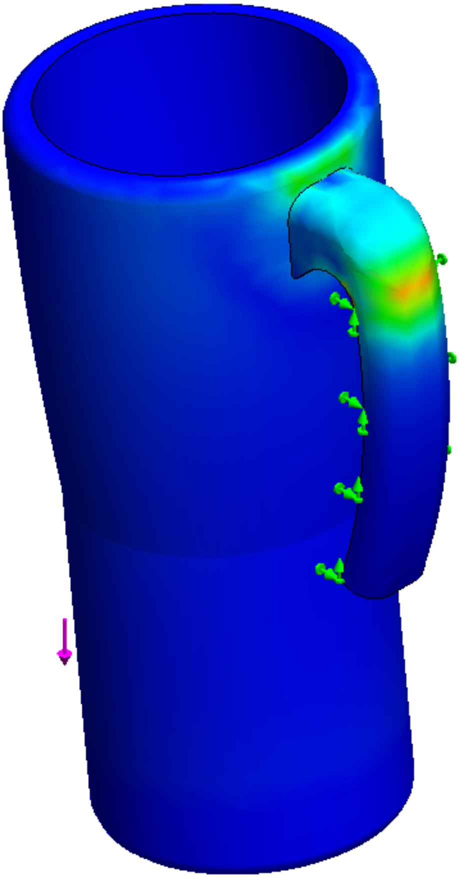
Change the Force Value to **400 N.**

Click 

4 Running Simulation.
Press the **Space Bar.**

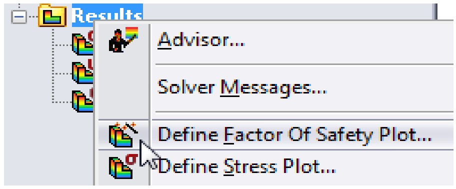
Double-click **Isometric.** Right-click **Mesh**. Select **Mesh and Run**. A Linear Static dialog box will appear click **Yes**.

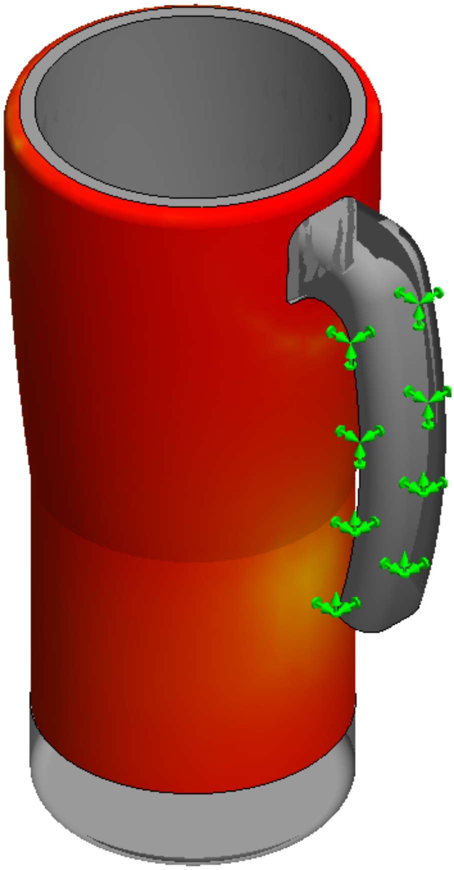




5 Factor of safety.

Right-click **Results.** Select **Define Factor Of Safety Plot.** Leave all settings the same and click . A simulation box will appear click **OK**.





In the top left corner of the screen SolidWorks tells us the **Factor of Safety** is **4**.

Click **Save**.

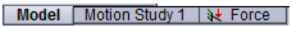
6 Design Evaluation.

After completing the both Static Studies, `Force 1` to show the concept of grasping the `Mug` and `Force 2` simulating holding the handle with a substance inside, it is obvious that the handle is much weaker.Study for `Force 1` showed a **Factor of Safety** of 430 and `Force 2` showed a **Factor of Safety** of 4. For the concept of redesign, we will delete the handle from the Assembly. This will better the Environmental Impacts because there will be less material.

7 Deleting `Handle`.

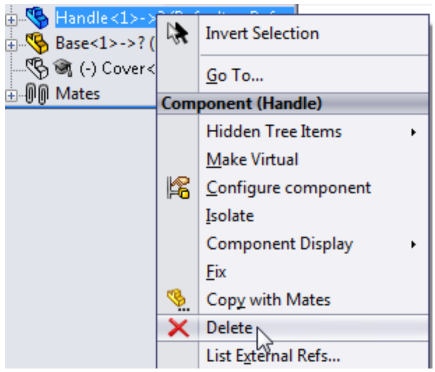
Because the Handle is a se per ate part, all we have to do is delete it from the assembly.

Select the **Model** tab in the bottom left hand corner of the screen.



Right-click (-)`Handle<1>`.

Select **Delete**.



Immediately after deleting the `Handle`, Sustainability updates the Environmental Impacts. Notice that all four of the Impacts show Green bars and all the numbers are less than the set baseline.

Thermal Study

Here we will create a thermal analysis that will show us if we are able to remove more material so that it is still suitable to hold.

1 Creating a Thermal Study.
Once again Select the `Simulation Tab` and create a `New Study`.

Name this Study **Convection** and select **Thermal**.

Click ✔.

From here we will be creating convection between the multiple surfaces to show us the outside temperature of the mug.

We will assume that the fluid inside the mug is water (example tea) at a temperature of 80 degrees celsius. We will also assume the air temperature surrounding the mug is at 25 degrees celsius.

Along with temperature, a convection coefficient is needed to calculate the convection through the materials of the mug. For the air we will use a convection coefficient of 10W/(m^2xK) and for the water we will use 500 W/(m^2xK). W stands for Watts, m stands for meters, and K stands for Kelvin.

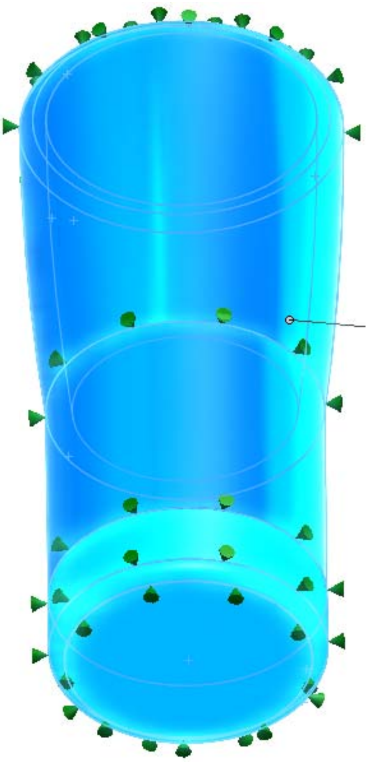
2 Thermal Loads 1.

Right-click **Thermal Loads.** Select **Convection**.

The first convection we will create will be the convection of air on the mug. This convection will be on the majority of the surfaces on the mug.

Select all the exterior surfaces except the lower inner surface and inner bottom surface.

Press **Space Bar.** Double-click **Base Surfaces** to select the Base Surface and Base Fillet.



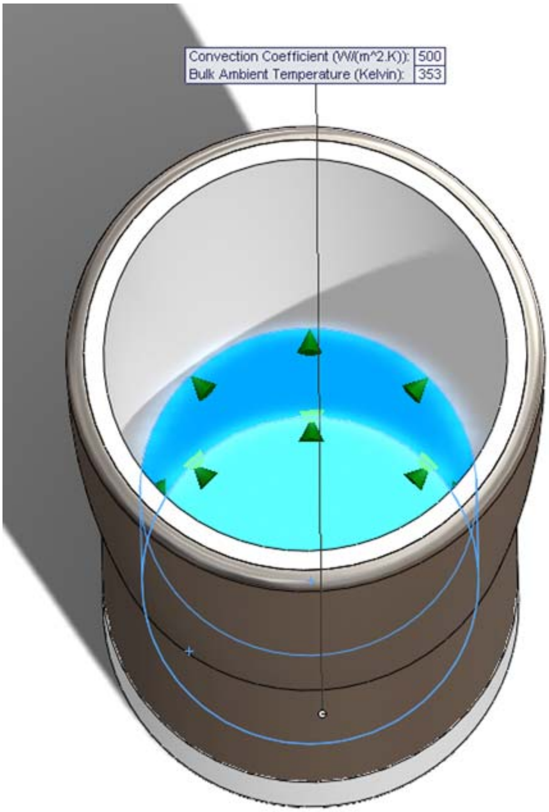
Next type **10** W/(m^2xK) for the convection coefficient and **298** (25+273) Kelvin for the ambient temperature.

Click 

3 Thermal Load 2.

Right-click **Thermal Loads.** Select **Convection**. Press **Space Bar.** Double-click **Inside Surfaces.**

This time only select the inner lower surface and the inner bottom.



Next set the Convection Coefficient to **500** W/(m^2xK) and the ambient temperature to **353** (80+273) Kelvin.

Click .

4 Running Simulation.

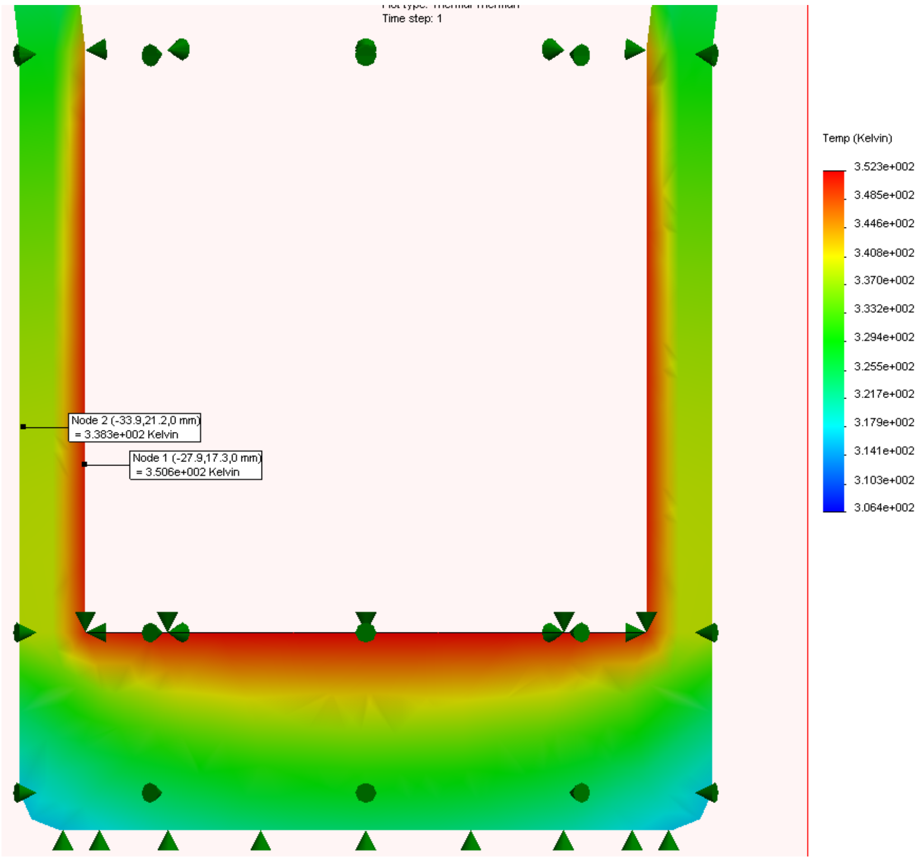
Like in the Force analysis right-click **Mesh**.

Select **Mesh and Run**.

Two dialog boxes will appear again.

5 Results.
After the analysis has run it is not easy to see the temperature difference from the inside of the mug to the outside. If we take a section clipping of the mug we will be able to better see the temperature gradient. Right-click **Thermal1** in Results. Select **Section Clipping**. Click ✓. Now we can see the temperature gradient through the walls of the mug. If we create a probe for the inner and outer surface of the mug we can see the temperature values.

6 Probe.
Press **Space Bar.** Double-click **Front.** Right-click **Thermal1.** Select **Probe**. Now click at two points relatively close to the ones shown.



7 Design Evaluation.
If we take a look at the temperature gradient we can see that it does not change much the middle of the wall to the outside surface. This tells us that we could decrease the thickness of the wall and still have a similar outside temperature.

Wall and Base Redesign

Here we will redesign the mug with thinner walls. In order to do this we will need to open the individual parts and alter the features. Once this is done we will have to remate them.

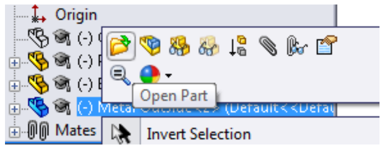
1 Edit Outside Wall.

Select the **Model** tab on the Bottom Left corner of the screen.

Right-click **Metal_Outside<1>** in the FeatureManager Design Tree.

Select **Open Part**.

SolidWorks will open a new window for this part.



2 Edit Revolve Feature.

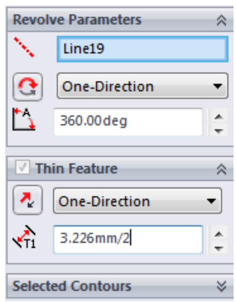
Because the Metal_Outside part is created from a Revolve feature we will decrease the thickness of the revolve.

Right Click on Revolve- Thin1 in the FeatureManager Design Tree.

Select **Edit Feature**.

Within Thin Feature add "**/2**" after **3.226mm**.

Click  .



By adding "/2" SolidWorks divides the thickness by 2.

Save and **Close** the part.

A dialogue box will appear asking if you would like to rebuild the assembly.

Click **Yes.**

Redesign of Base

Here we will be redesigning the base of the Mug. After redesigning the `Metal Outside` we will have to also redesign the plastic Base. Because we made the `Metal Outside` thinner we will have to make the Base smaller as well.

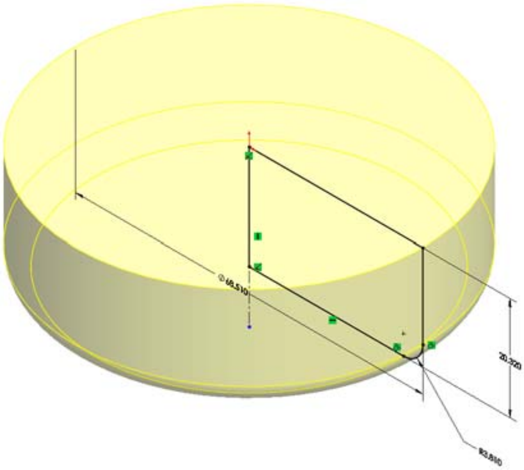
1 New Dimensions.

Right-click **Base<1>** in the FeatureManager Design Tree**.**

Select **Open Part**.

Expand `Revolve1`.

Double-click `Sketch1`.



Double-click the **68.510mm** Dimension. Change the Diameter to **65.28mm**. Click .

2 Exit sketch.

Exit the sketch by clicking . Save and close the part. A dialogue box will appear asking if you would like to rebuild the assembly. Click **Yes.**

Simulation of Redesign

Here we will re run the two simulations created before to see if the design is still acceptable.

1 **Rerun Static Study.**
Select the **Force 1.**

Press **Space Bar.**

Double-click **Isometric.**

SolidWorks will update the Fixtures and External Loads of the redesign.

Right-click **Mesh** and Select **Mesh and Run**.

Right-click **Results.**

Select **Define Factor Of Safety Plot...**

Click .

The **Factor Of Safety Plot** shows that the assembly is still very safe with a **Factor Of Safety** of 2.1e+002 or 210.

2 **Rerun Thermal Study.**
Even though the Static Study showed good results we still need to run the Thermal Study to make sure that the outside surface it not too hot.

Click the **Convection** tab at the bottom of the screen.

SolidWorks once again will update the Simulation for the redesign.

Right-click **Mesh** and Select **Mesh and Run**.

Press **SpaceBar**.

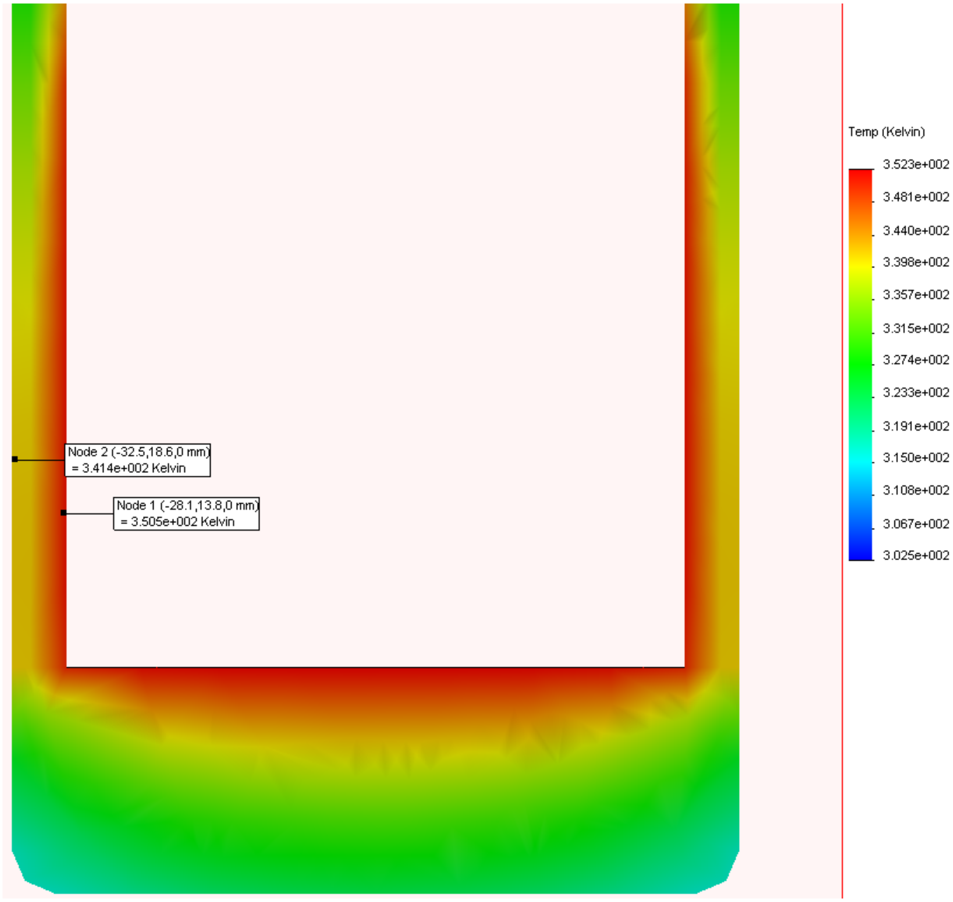
Double-click **Front**.

3 Probe.

Right-click `Thermal1` in Results.

Select **Probe**.

Select two points as shown.



Previously the outside temperature was at about 338 Kelvin or 65 degrees Celsius. Now the outside temperature is at about 341 Kelvin or 68 degrees celsius.

The temperature only increased about three degrees Celsius after decreasing the `Metal Outside` thickness by half.

This is a very minimal increase for the amount of material we saved. This will be our Final Design.

Looking at Sustainability

Here we will compare the sustainability of the original design with the that of the final design, along with two alternative designs.

Throughout the entire tutorial the environmental impacts have been decreasing.

Original Environmental Impacts	Redesign Environmental Impacts
Carbon: 7.11g of CO2	Carbon: 3.78g of CO2
Energy: 87.03MJ	Energy: 47.72MJ
Air: 0.04g of SO2	Air: 0.02g of SO2
Water: 0.02g of PO4	Water: 0.00962g of PO4

Overall all the Environmental Impacts have decreased by about 50%.

Conclusion

In the begining of the design we had a bulky mug that was far from being environmentally friendly. SolidWorks allowed us to conduct Force and Thermal Simulation which allowed us to rid the mug of excess features as well decrease material use while keeping relatively similar insulation.