AUTODESK UNIVERSITY 2013

Getting Started with Autodesk® Sim 360

James Herzing – Autodesk

SM3410-L Getting Started with Autodesk Sim® 360

Learning Objectives

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

- Open models in Sim 360
- Simplify models for simulation
- Set up structural thermal and fatigue calculations
- Compare results from multiple designs

About the Speaker

James Herzing has been using the Autodesk Simulation software for 10 years, working in various positions that deal with customer issues, ensuring their success, and creating highly viewed marketing material. Currently, he works as part of the Industry Strategy and Marketing division as a member of the Go-to-Market team, focusing on Simulation TV and customer nurturing and enablement. He graduated from the Pennsylvania State University in 2004 with a BS in Mechanical Engineering and a minor in Engineering Mechanics. He has presented and assisted in over 12 courses at Autodesk University in the past, receiving multiple top rated speaker scores in these classes.

Email Address: *james.herzing@autodesk.com*

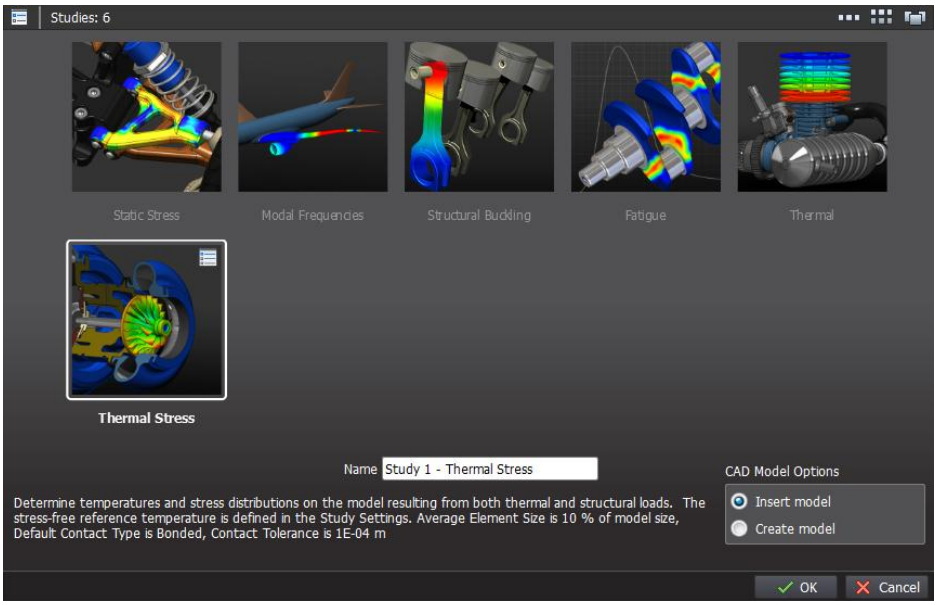
Opening Models in Autodesk Sim 360

Launch Autodesk Sim 360

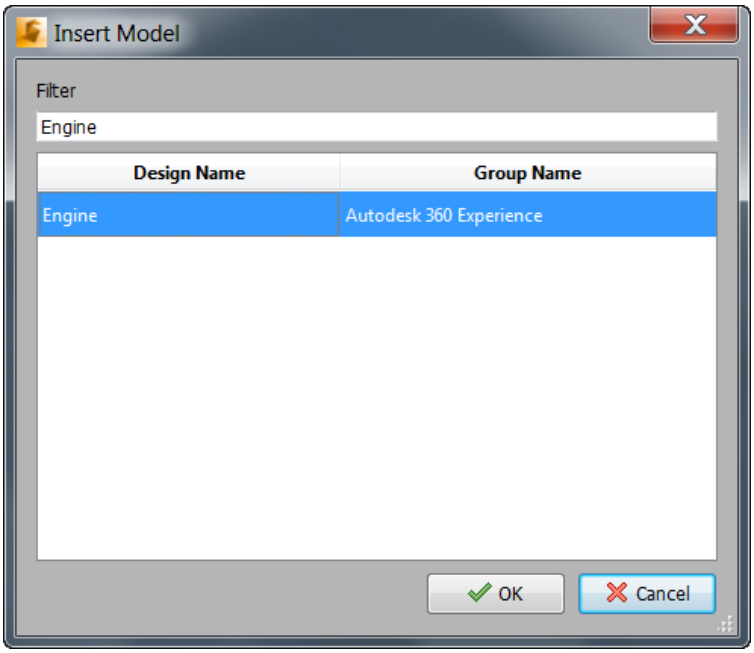
1. Launch Sim 360
   - Start > All Programs > Autodesk > Autodesk Sim 360

2. Create a new analysis named “Thermal Stress Analysis”
   - Simulation Tab > Type name “Thermal Stress Analysis > Create

3. Launch a thermal stress analysis
   - Click Thermal Stress > Insert model > OK



4. Choose to insert the Engine model
   - Filter > Engine > OK

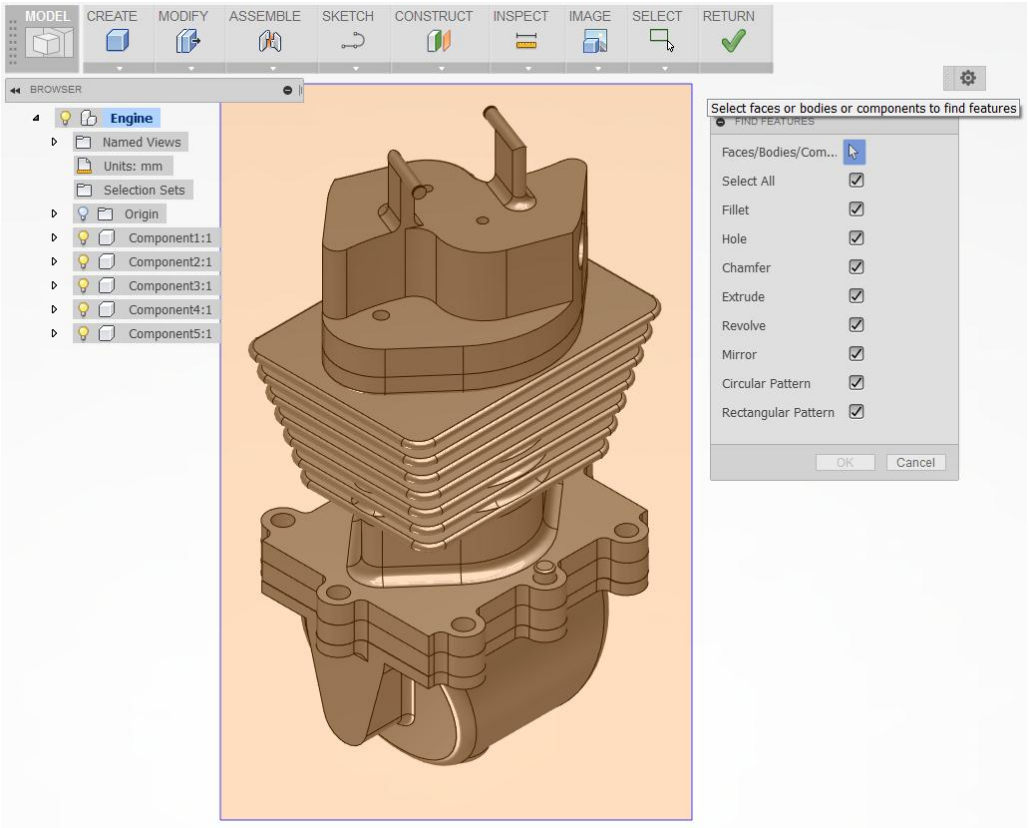


Simplify Geometry before Analysis

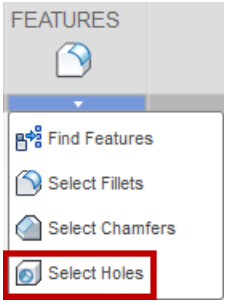
Simplify Geometry

1. Go to the Simplify workspace
   - Setup button > Simplify

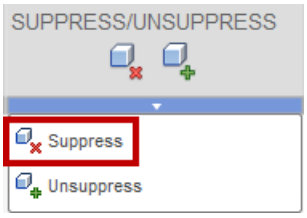
2. Find all features of the model
   - Features > Find Features
   - Box Select the entire model > OK
   - Click Return on the ribbon bar



3. Automatically select holes to suppress
   - Features > Select Holes > Drag Diameter handle all the way to the left > Enter



4. Choose to suppress the selected holes
   - Suppress/Unsuppress Panel > Suppress

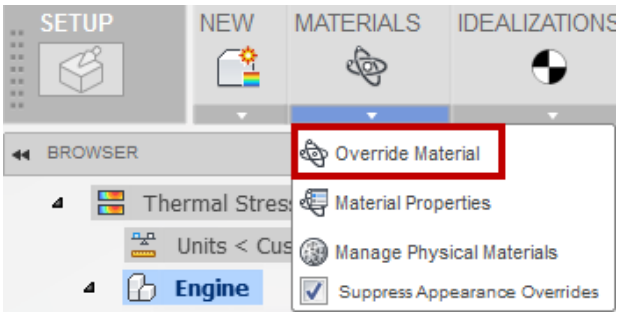


5. Go back to the FEA Setup screen
   - Click Edit Setup

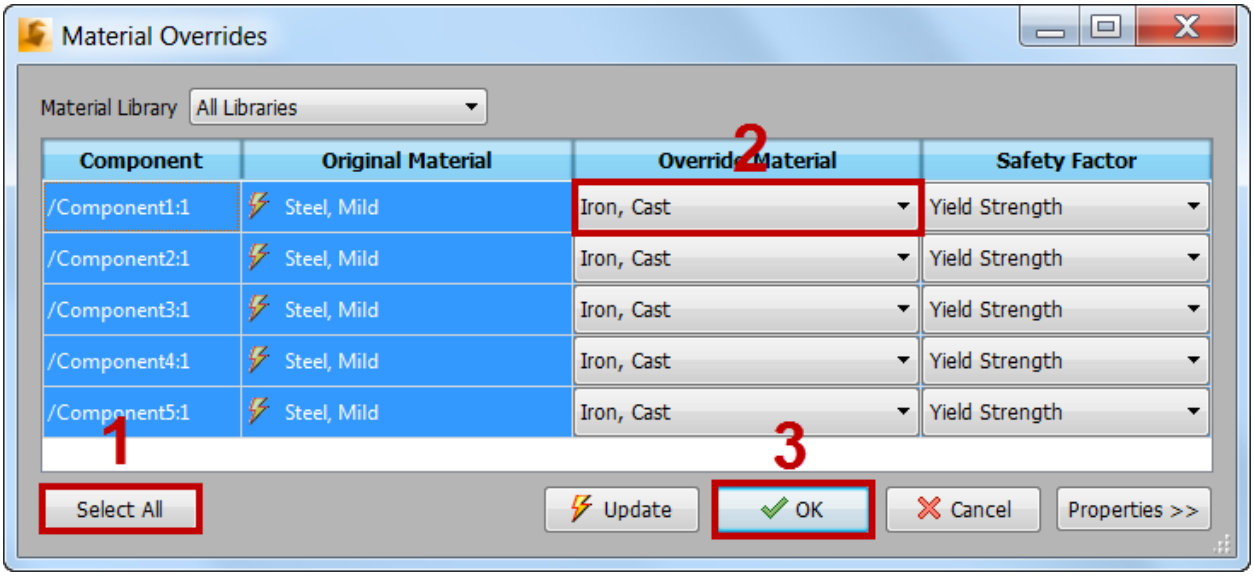
Setup a Thermal Stress and Fatigue Analysis

Define Materials

1. Choose to override the default material definitions
   - Materials > Override Material



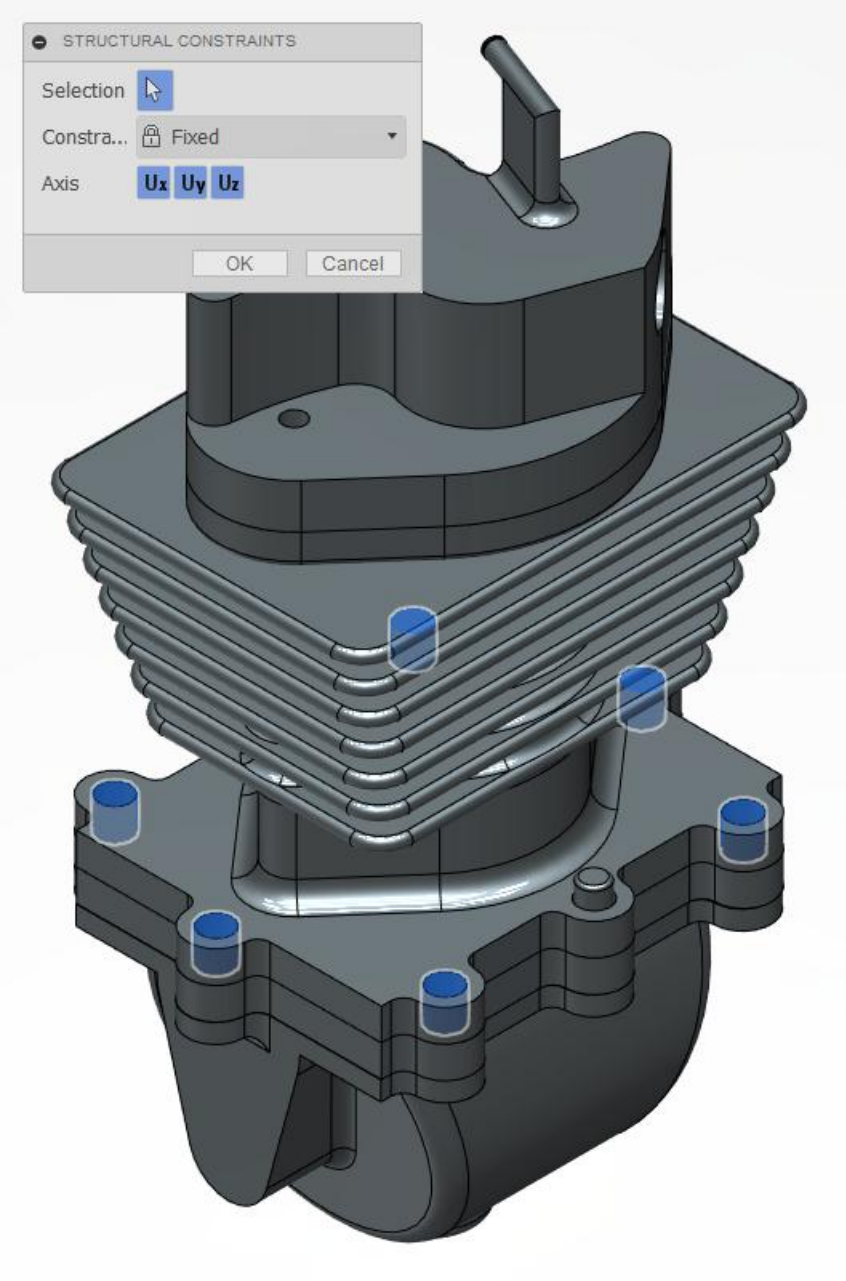
2. Select all of the materials and choose to make them cast iron
   - Select All
   - Override Material > Iron, Cast
   - Click OK



Define Boundary Conditions

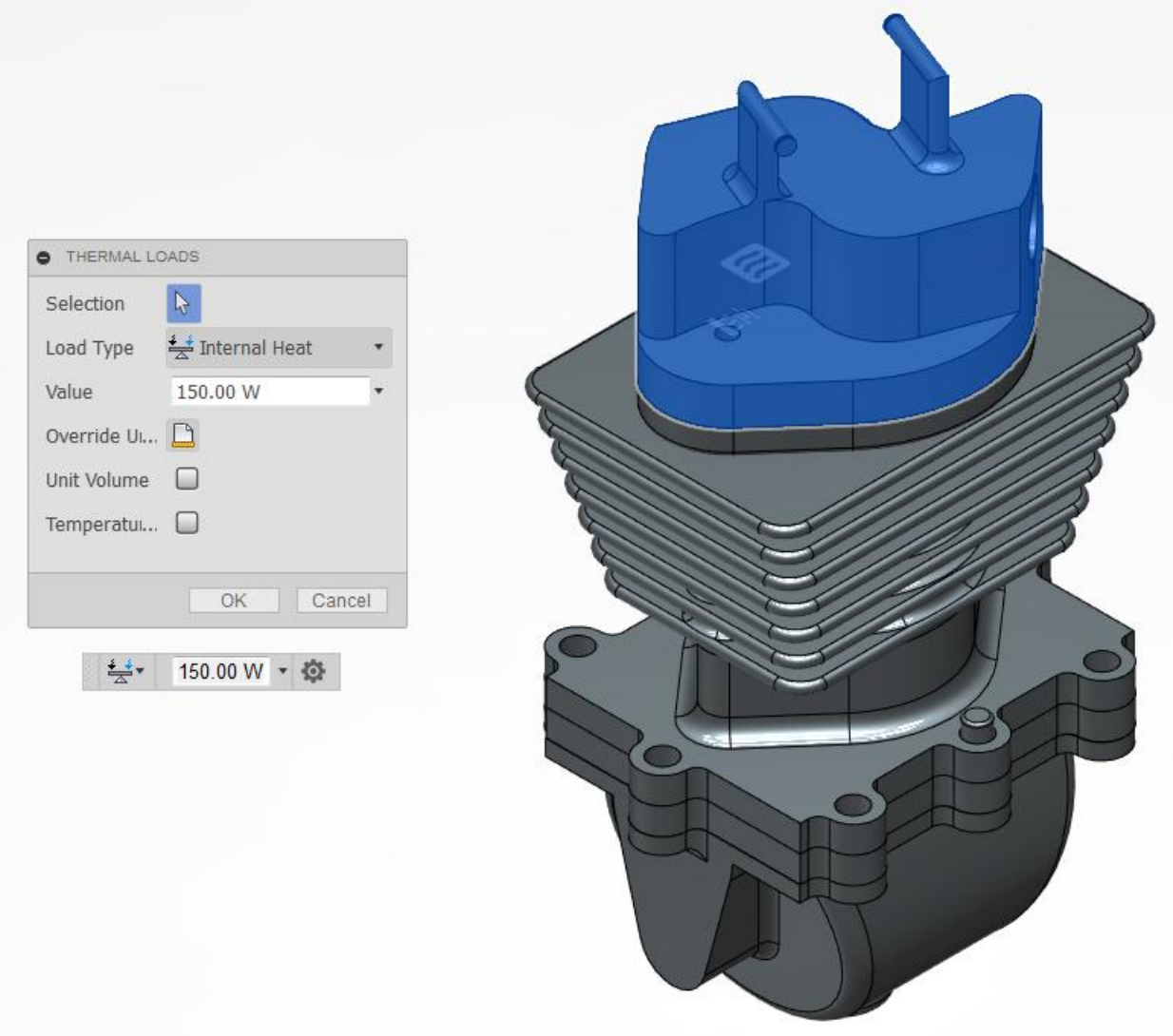
1. Select surfaces where boundary conditions will be placed
   - Select the surfaces shown below

2. Apply fixed boundary conditions
   - Constraints > Fixed > OK



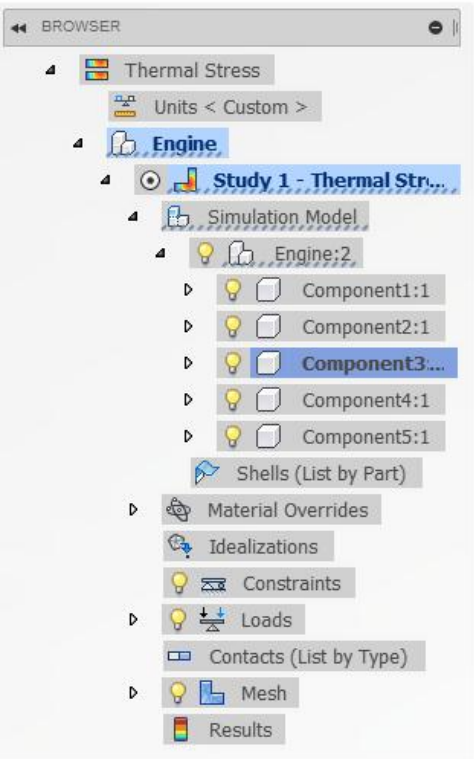
Apply Loads

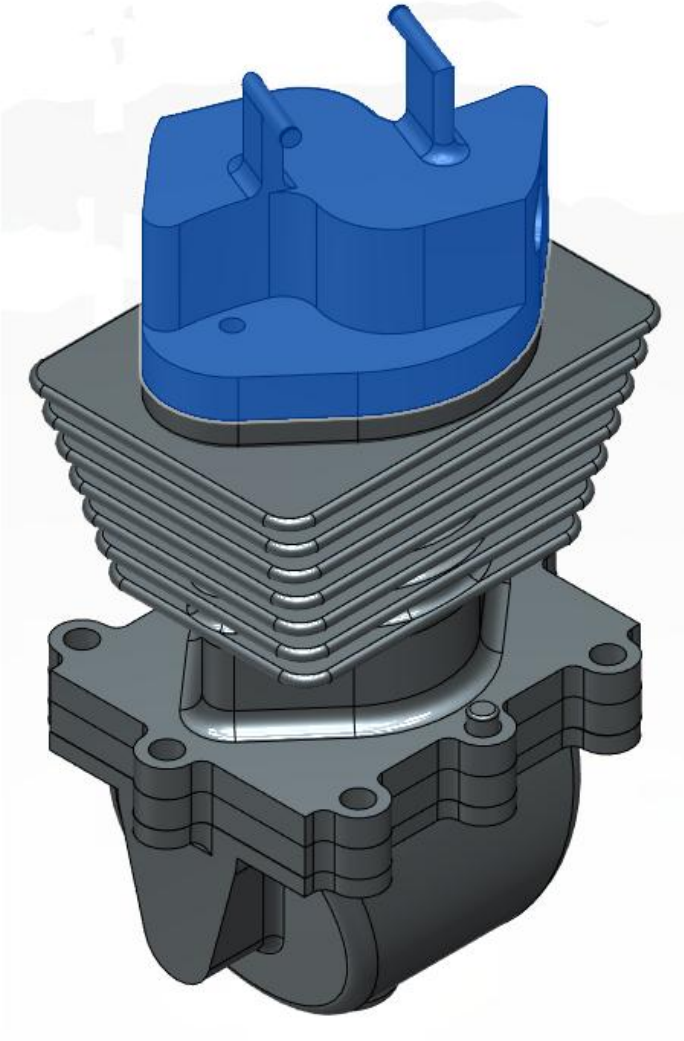
1. Apply an internal heat load to the cylinder head
   - Loads > Thermal Loads > Internal Heat
   - Select the cylinder head part
   - Value = 150 W > OK



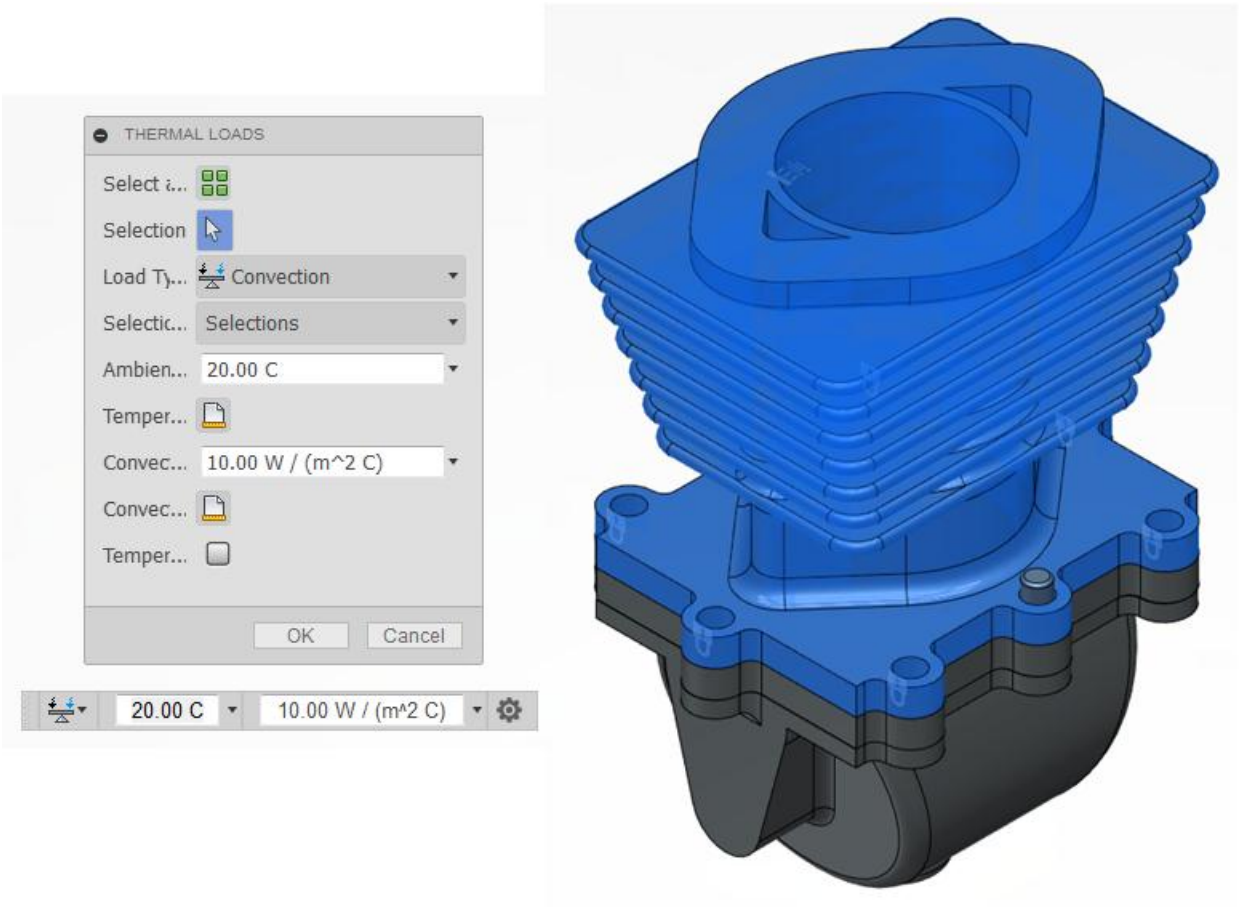
2. Hide the cylinder head to more easily select other parts/surfaces
   - Double click any surface on cylinder head (below) > Right Click > Hide

Note: *You could also expand the model tree to the left and click the light bulb next to Component3 to hide this part.*

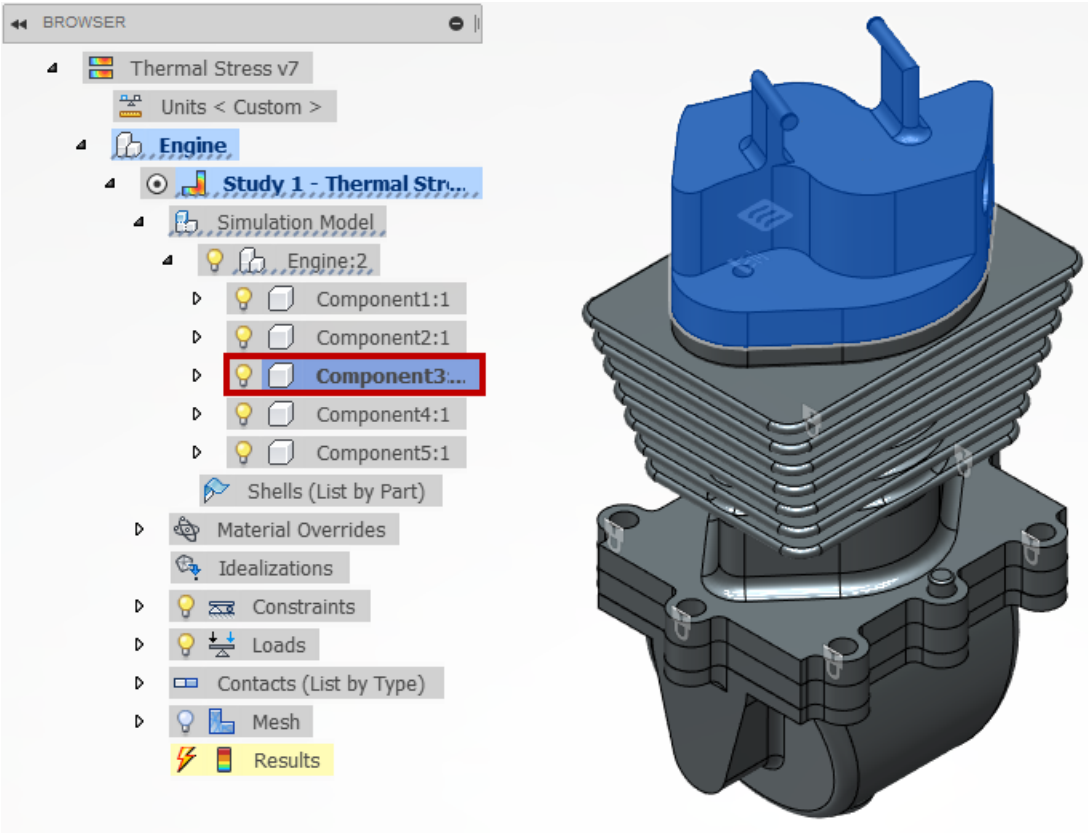




3. Apply a convection load of 10 W/m^2 C to the surfaces of the cylinder barrel
   - Loads > Thermal Loads > Convection
   - Click the “Select all faces” button
   - Pick a surface on the cylinder barrel
   - Convection = 10 W / (m^2 C)
   - Click OK

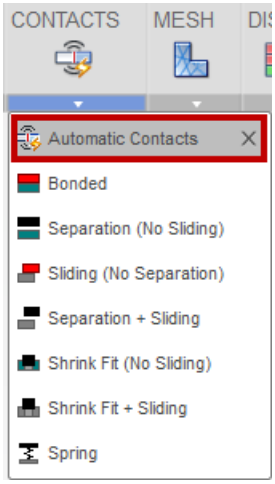


4. Activate visibility to the parts that were hidden
   - Click the arrow next to Simulation Model > Engine:2
   - Turn on the light bulb next to Component3:1

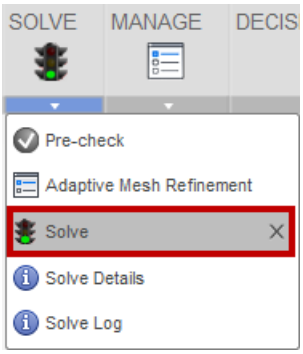


Define Contact between Parts and Analyze

1. Automatically detect contact between the parts in your model
   - Contacts > Automatic Contacts



2. Analyze your model
   - Solve > Solve

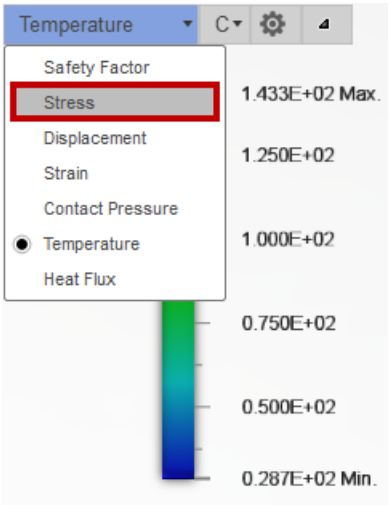


Note: If your model is sufficiently set up, your light will be green, indicating you are ready to analyze the model.

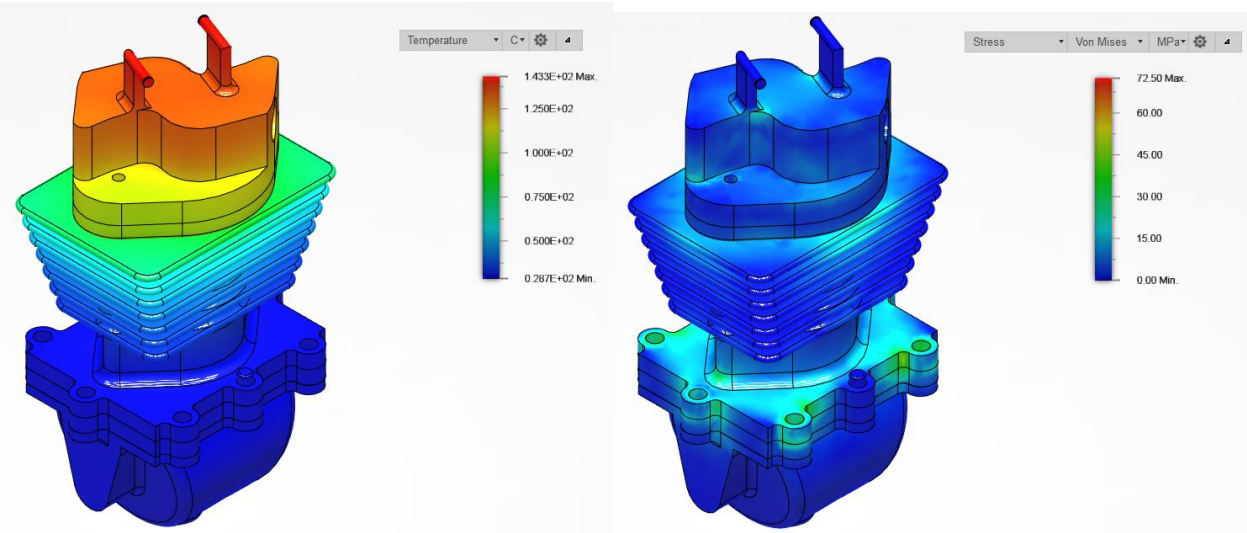
Review Your Results

Below are just a few of the steps that you can do to review your results in Autodesk Sim 360. After performing these steps, feel free to take a few minutes to explore some of the other features in the results environment. We will take a look at a few of the other options, such as comparing multiple designs in the same screen.

1. After reviewing your factor of safety, change the result type to stress
   - Legend  > Stress



2. After reviewing your stress, review your temperature results
   - Legend > Temperature

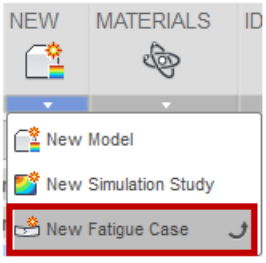


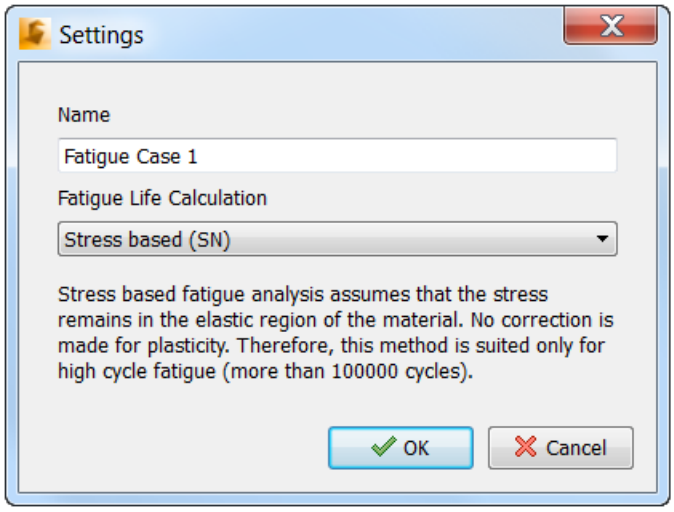
Create a Fatigue Analysis

1. Return to the Setup Environment
   - On the ribbon bar, click on Results and select Setup

2. Create a fatigue case based on the thermal results
   - New > New Fatigue Case

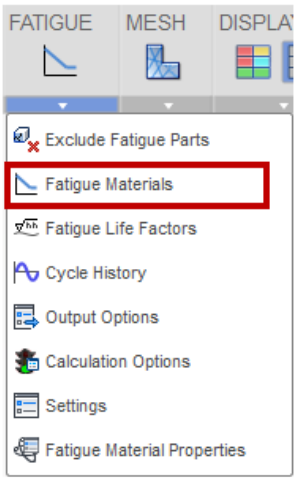
3. Choose a stress based method for fatigue calculations
   - Fatigue Life Calculation > Stress based
   - Click OK

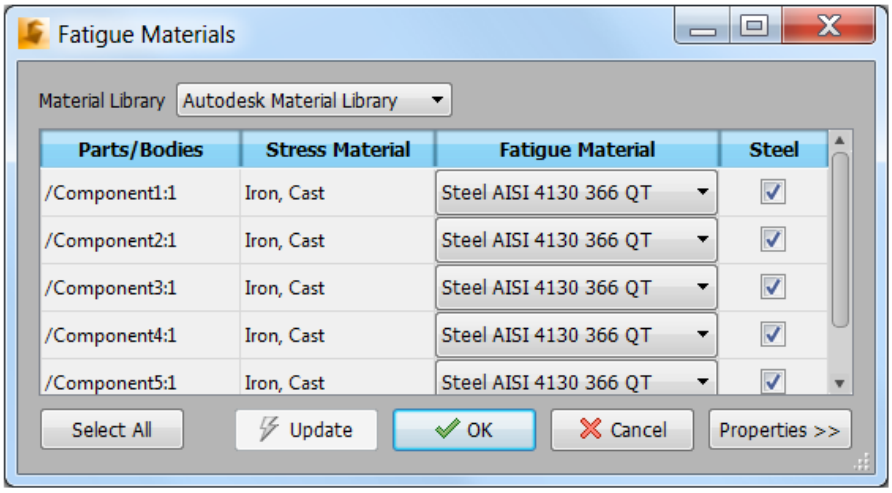




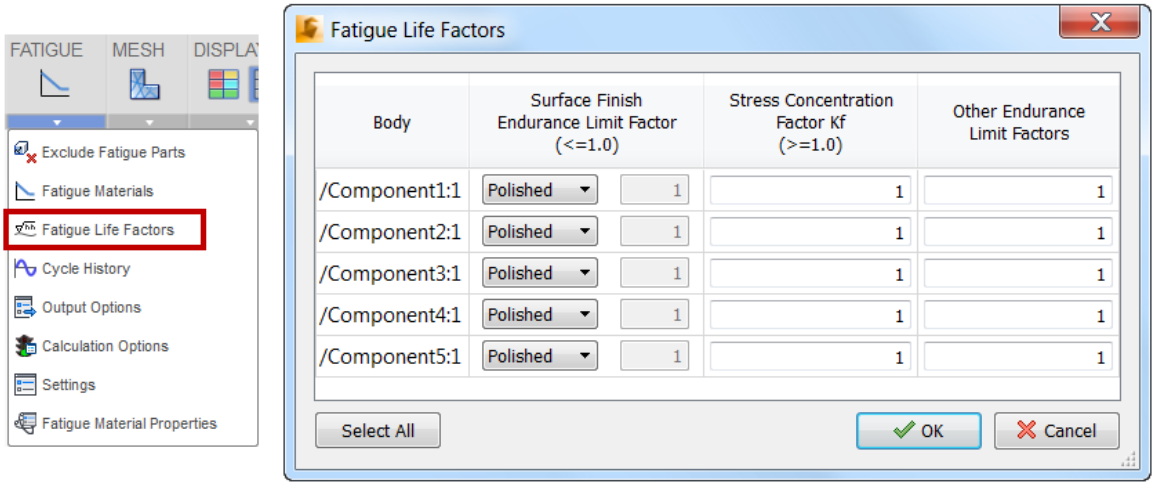
Define Fatigue Parameters

1. Define your materials the fatigue analysis
   - Fatigue > Fatigue Materials
   - Select All > Steel AISI 4130 366 QT > OK

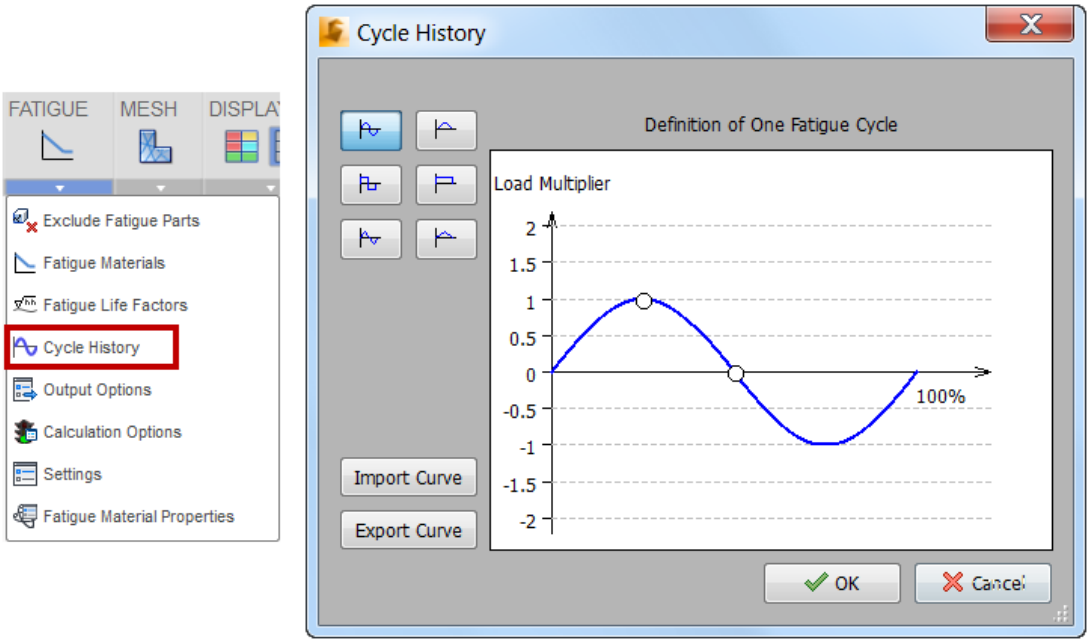




2. Define the fatigue life factors
   - Fatigue > Fatigue Life Factors
   - Select All > Polished > OK



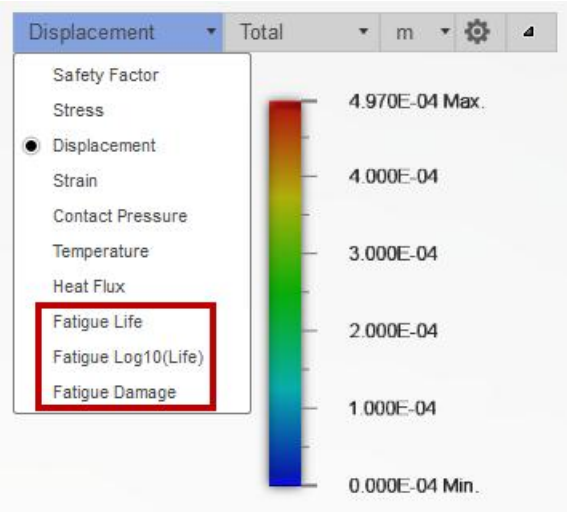
3. Define the cycle history
   - Fatigue > Cycle History
   - Choose a curve > OK



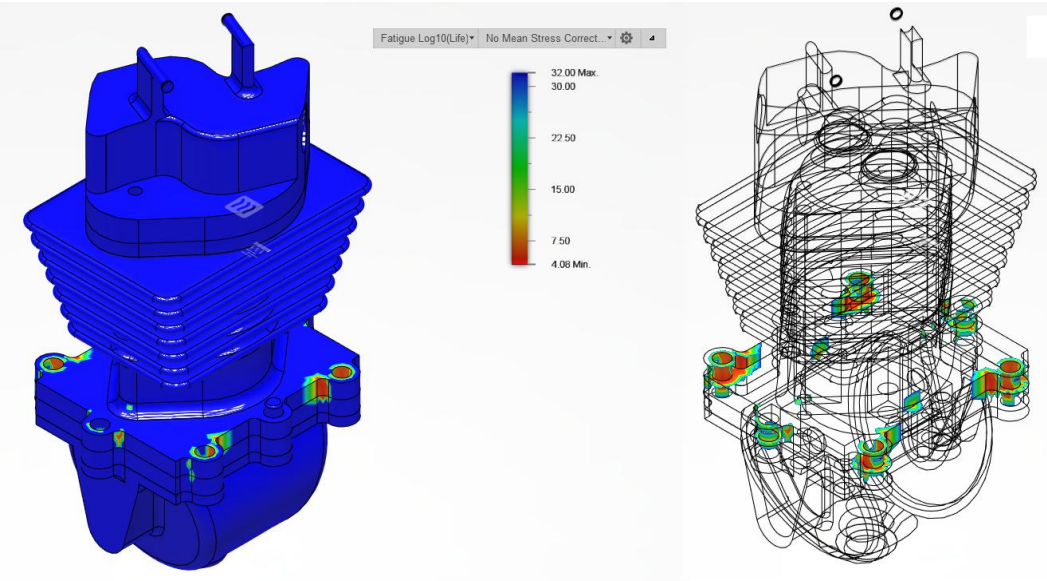
4. Perform the analysis
   - Solve > Solve

Review Fatigue Results

1. Review the fatigue life of the engine to determine cycles to failure
   - Click on Safety Factor display > Fatigue Life



2. Review the other fatigue options
   - Fatigue Life > Fatigue Log10 (Life)
   - Fatigue Log10(Life) > Fatigue Damage

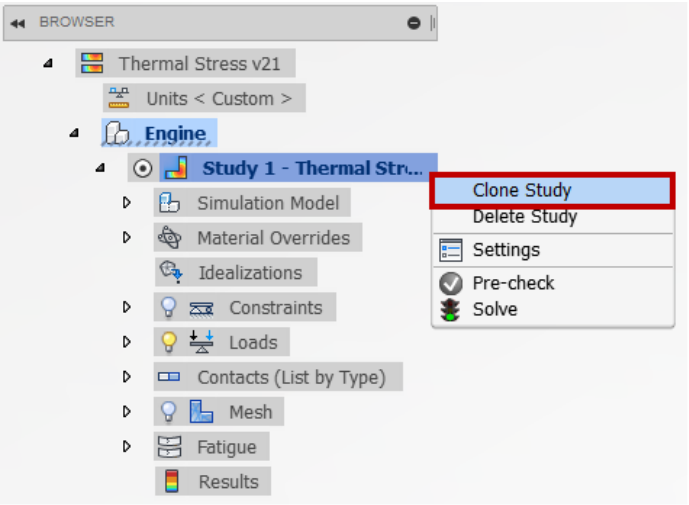


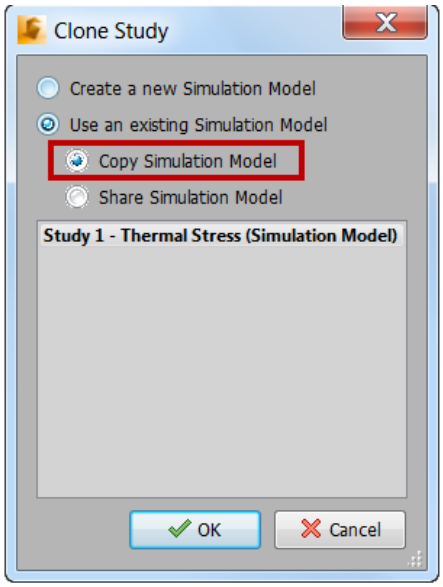
Create a New Study to Compare Results

Create a New Study

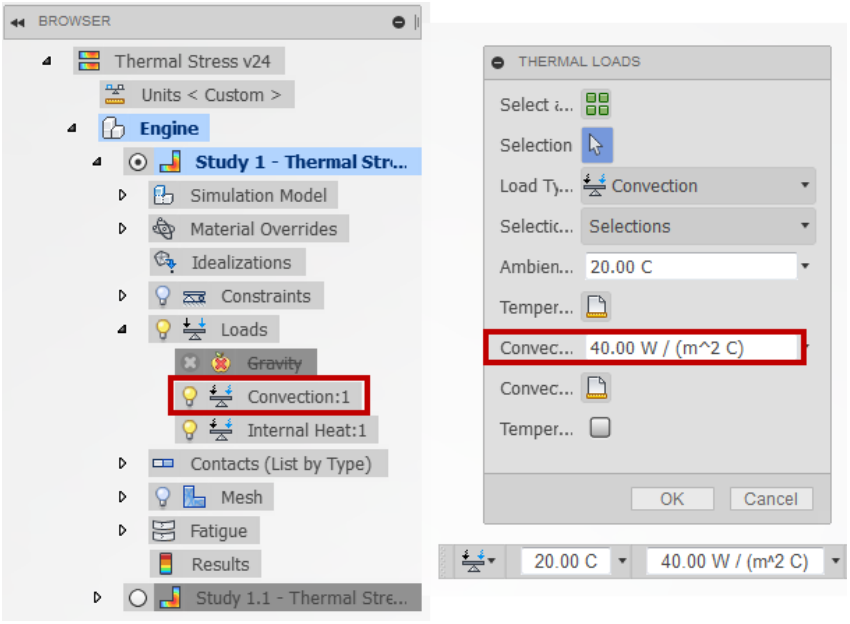
1. Return to the Setup environment
   - From the ribbon, click on Results > Setup

2. Clone the first thermal stress study
   - Right click on Study 1 – Thermal Stress > Clone Study
   - Choose Use an existing Simulation Model > Copy Simulation Model
   - Click OK





3. Modify the convection coefficient to help reduce the engine temperature
   - Right click Convection:1 > Convection Coefficient = 40 W / (m^2 C)
   - Click OK

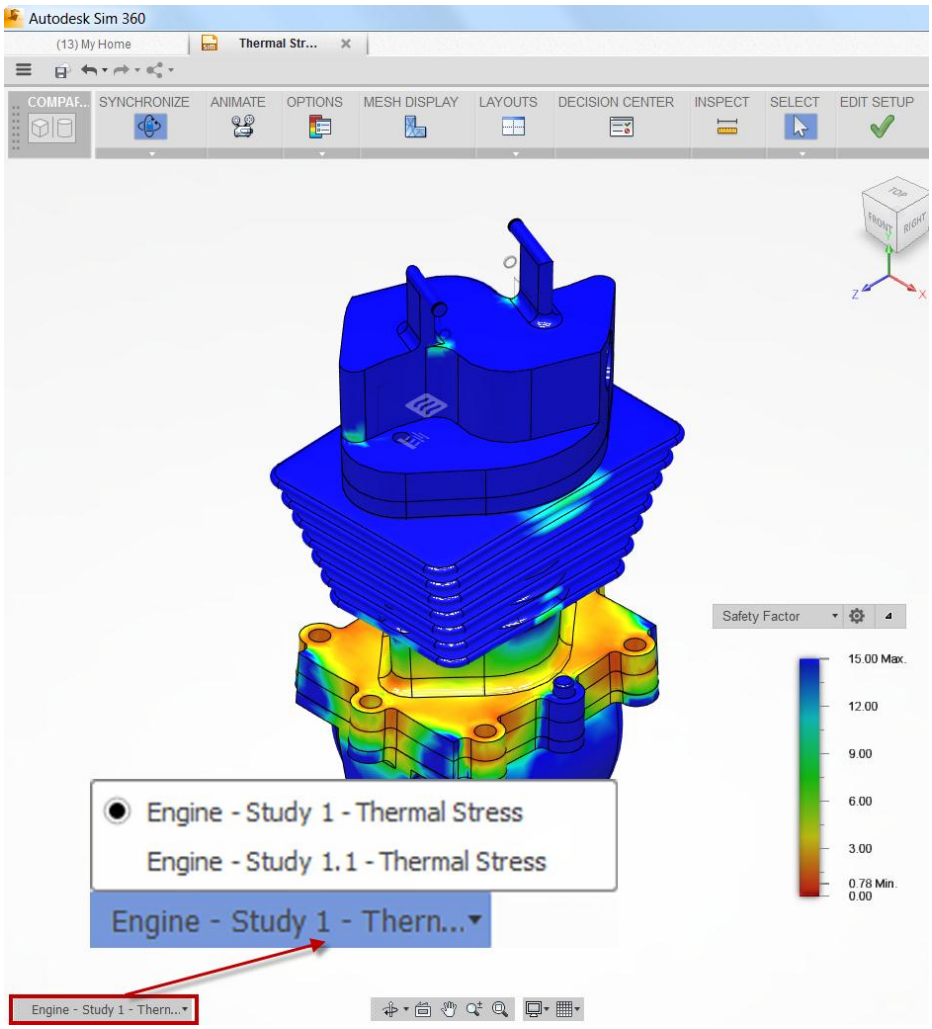


4. Analyze the new thermal stress study
   - Click on Solve > Solve

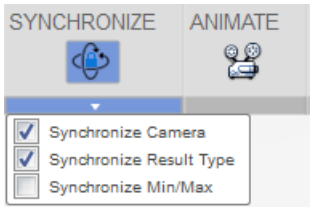
Create a New Study

1. In the results environment, compare the two studies
   - From the ribbon, click on Results > Compare

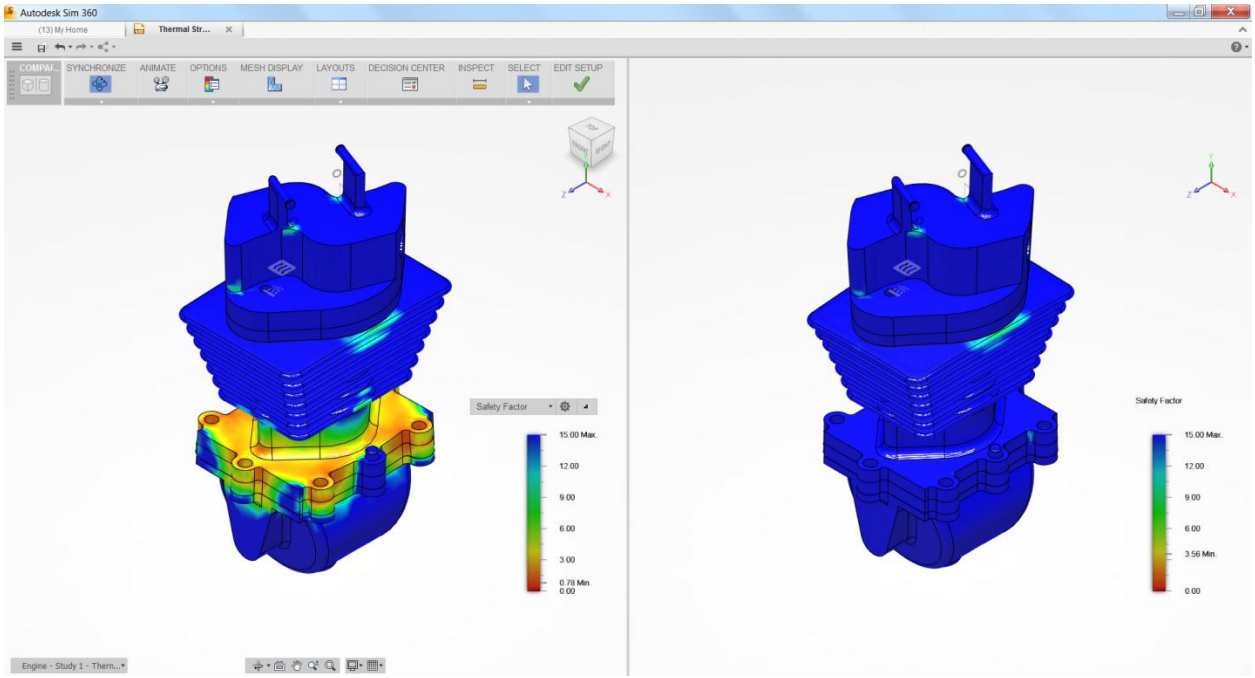
2. Verify that both of your studies are being displayed
   - On the left display > Engine – Study 1 – Thermal Stress
   - On the right display > Engine – Study 1.1 – Thermal Stress



3. Link the two design studies for easier comparison
   - From the ribbon, Synchronize > Check Synchronize Result Type



4. Review the various results to determine the improvement
   - Safety Factor
   - Stress
   - Temperature



Boom! That's it! Thank you for coming, please come again.