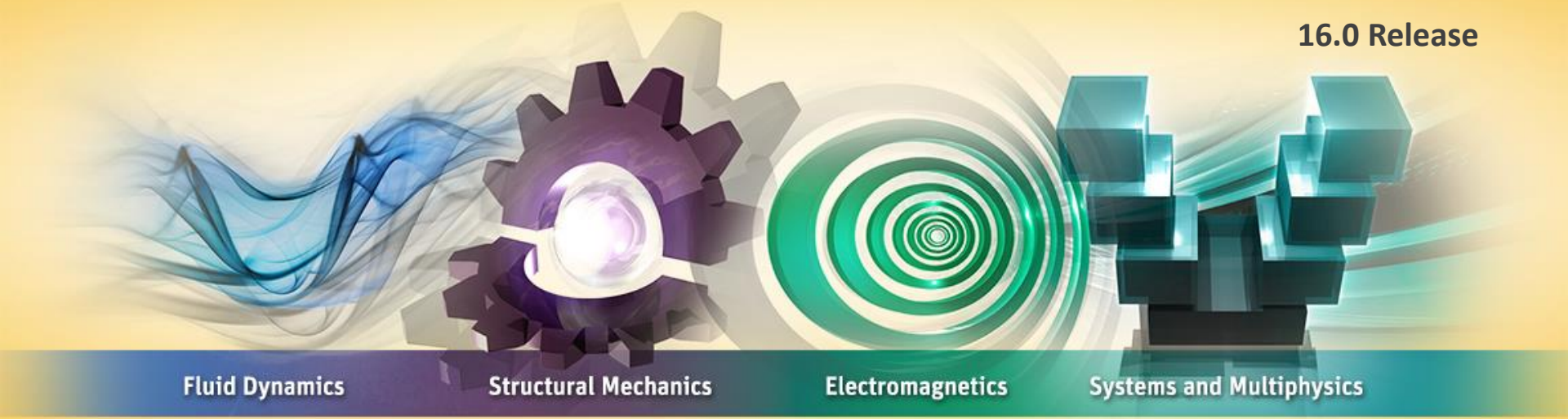
Workshop 9.1
Steady State Thermal Analysis

16.0 Release



Introduction to ANSYS Mechanical

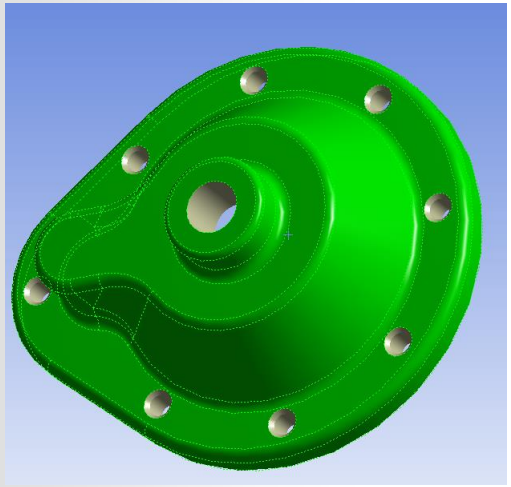
Realize Your Product Promise®

Goals

In this workshop we will analyze the pump housing shown below for its heat transfer characteristics.

Specifically a plastic and an aluminum version of the housing will be analyzed using the same boundary conditions.

Our goal is to compare the thermal results for each configuration.



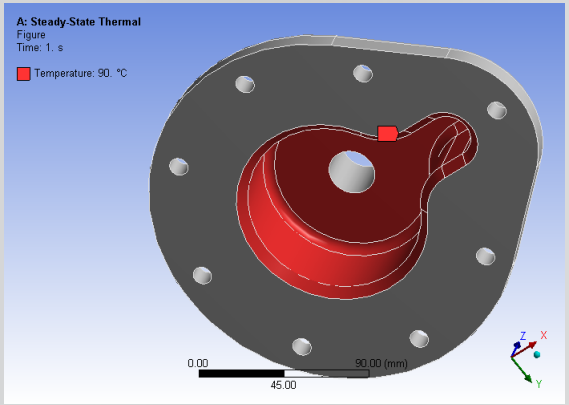
Assumptions

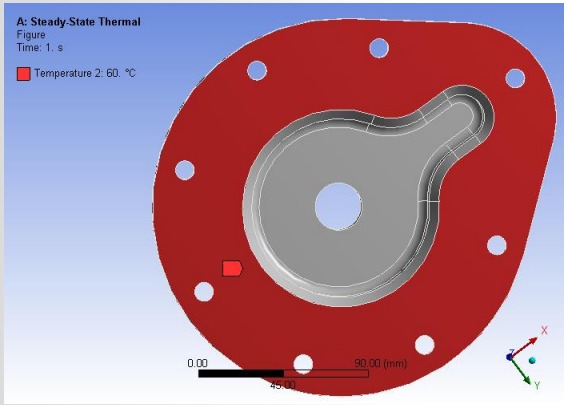
Assumptions:

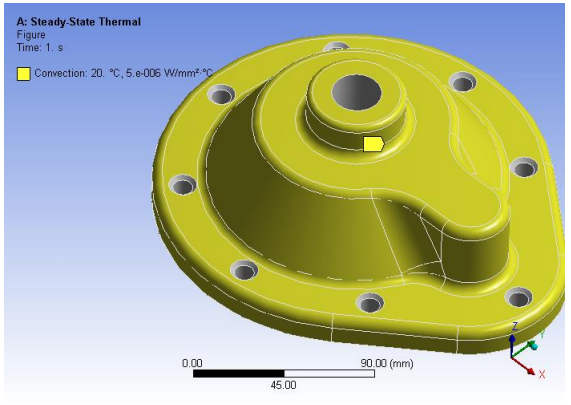
The pump housing is mounted to a pump which is held at a constant 60 °C. We assume the mating face on the pump is also held at this temperature.

The interior surfaces of the pump are held at a constant temperature of 90 °C by the fluid.

The exterior surfaces are modeled using a simplified convection correlation for stagnant air at 20 °C.



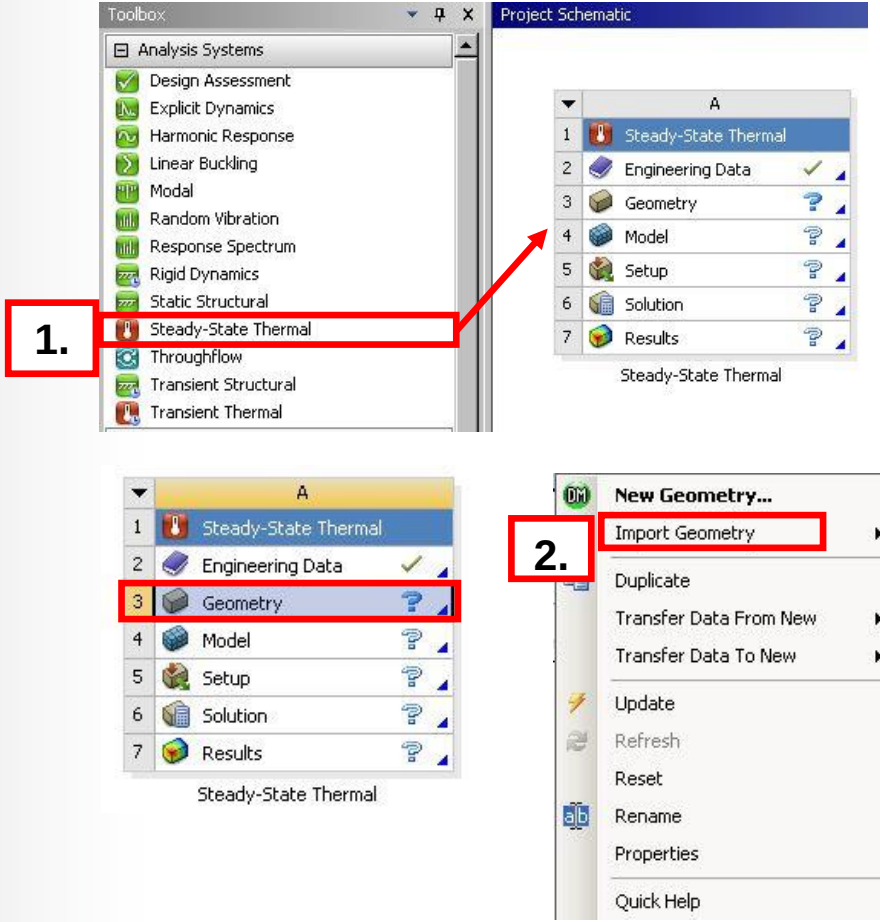




Project Schematic

1. From the Toolbox, double click “Steady-State Thermal” to create a new Steady State Thermal system.

2. RMB the Geometry cell and “Import Geometry” – browse to the file: “Pump_housing.stp”.

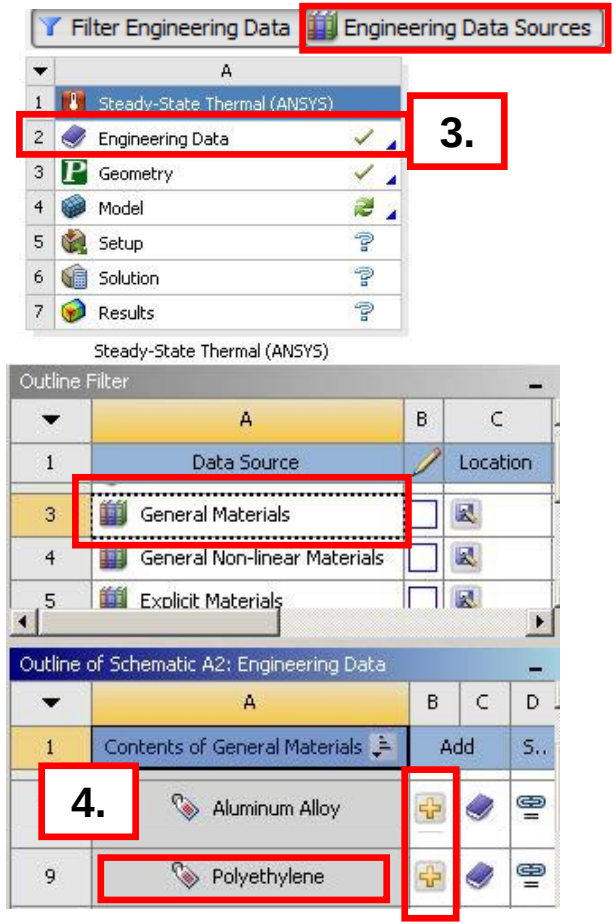


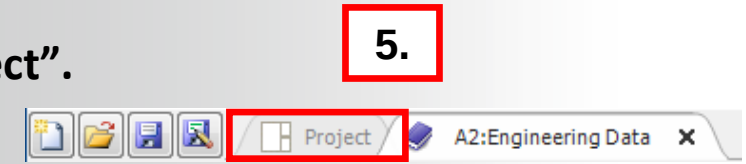
. . . Project Schematic

3. Double click “Engineering Data” and activate the Data Source filter.

4. With “General Materials” highlighted click the ‘+’ next to “Aluminum Alloy” and “Polyethylene” properties to add them to the project.

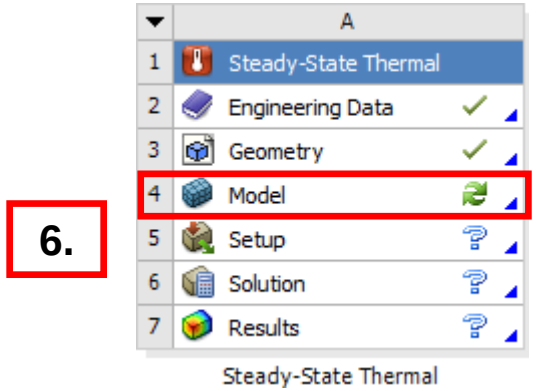
5. “Return to Project”.



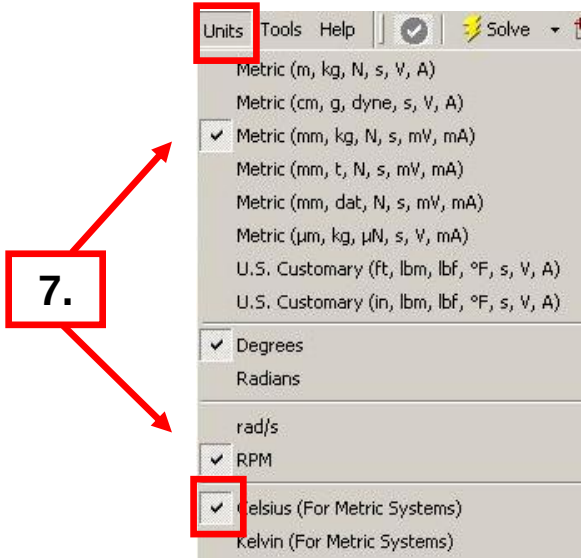


. . . Project Schematic

6. Double click the “Model” cell to open the Mechanical application.

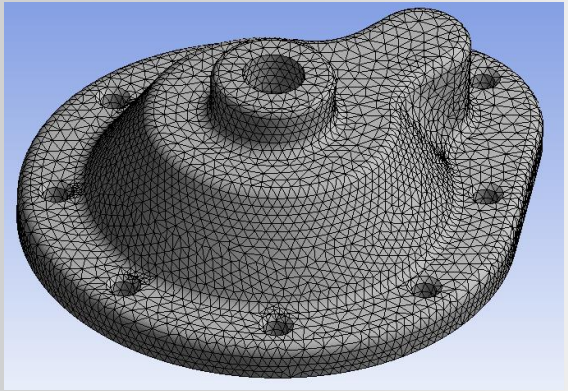


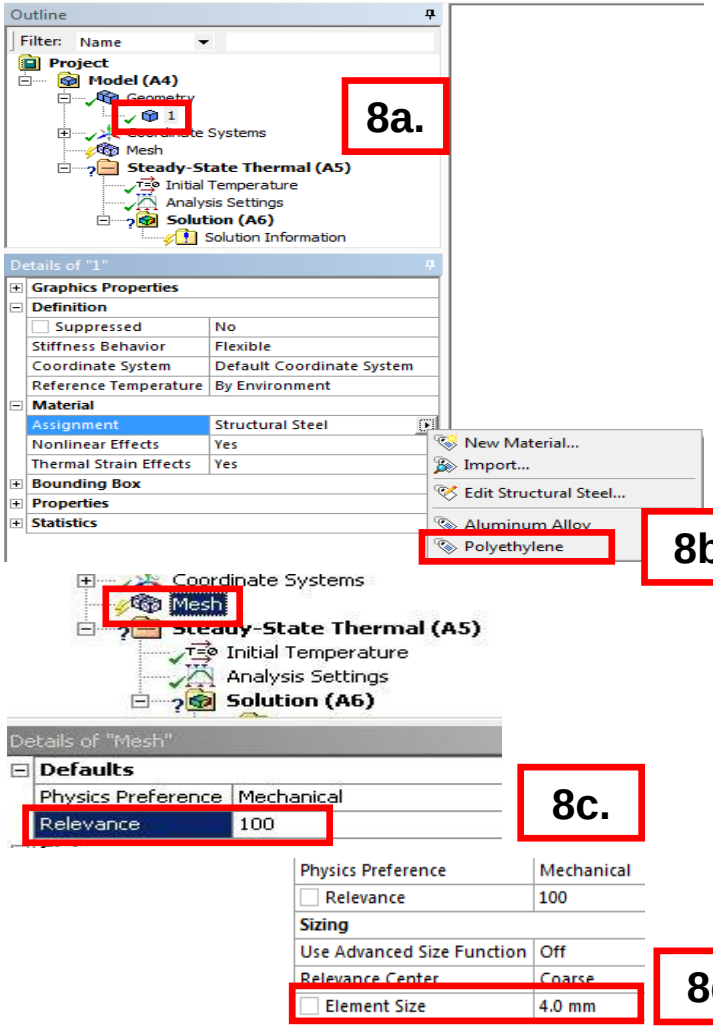
7. From the Units menu choose/verify:
   - “Metric (mm, kg, N, s, mV, mA)”
   - “Celsius (For Metric Systems)”



Preprocessing

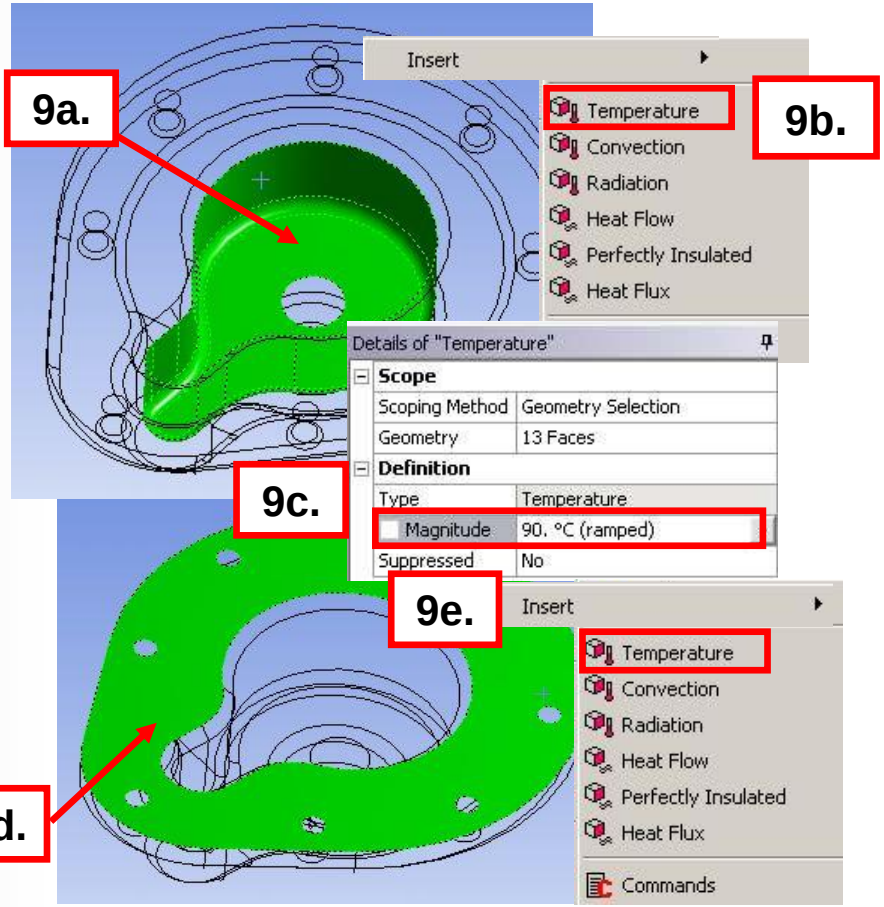
8. Change the material and mesh on the pump housing (part ‘1’):
   a. Highlight “1” under geometry.
   b. From details assign the material “polyethylene”.
   c. Highlight the Mesh branch and set the mesh relevance = 100.
   d. Use Element Size = 4 mm





Environment

9. **Apply temperatures (highlight the Steady State Thermal branch):**
   a. **Select the interior surfaces (13 faces) of the pump housing *(hint: use "Extend To Limits" selection feature).***
   b. **RMB > Insert > Temperature.**
   c. **Set "Magnitude" field to 90 °C.**
   d. **Select the mating surface of the pump housing.**
   e. **"RMB > Insert > Temperature".**
   f. **Set "Magnitude" field to 60 °C.**



9f.

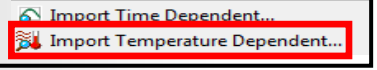
Details of "Temperature 2"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Temperature
Magnitude	60. °C (ramped)
Suppressed	No

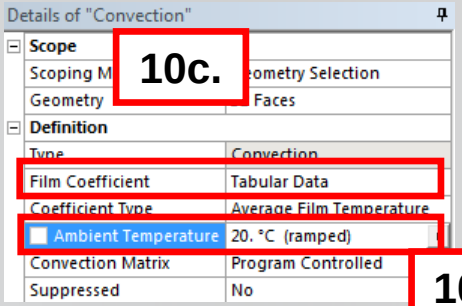
. . . Environment

10. Apply Convection:
    a. Select the exterior (32) surfaces of the pump housing (hint: use extend to limits).
    b. "RMB > Insert > Convection".
    c. In the "Details of Convection" click in the "Film Coefficient" field and choose "Import . . . ".

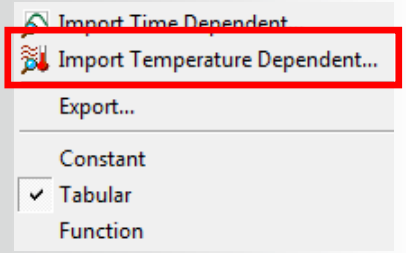
Be sure to choose import for convections.

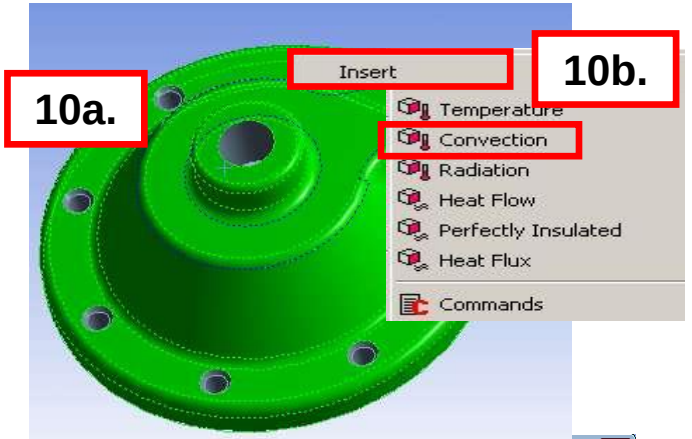


    d. "Import" the correlation "Stagnant Air – Simplified Case".
    e. Set the "Ambient Temperature" field to 20 °C.



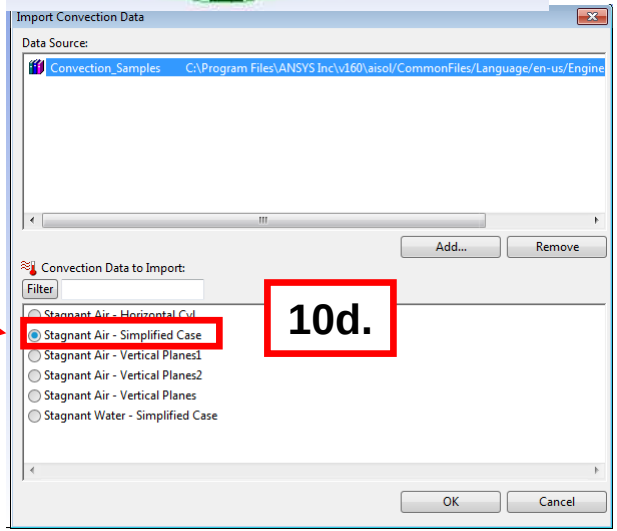
10c.





10a.

10b.



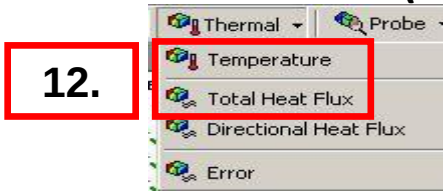
10d.

10e.

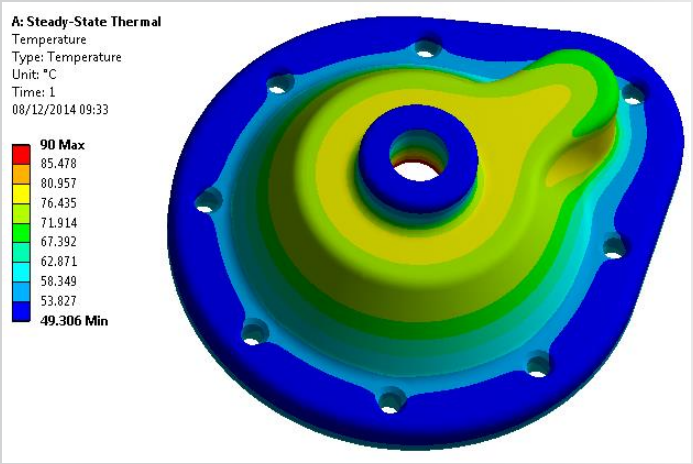
Solution – Model A

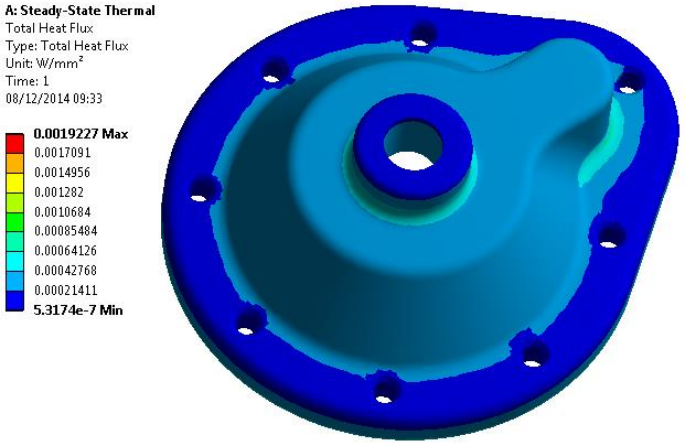
11. Solve the model.

12. When the solution is complete insert Temperature and Total Heat Flux results (solve to evaluate results).



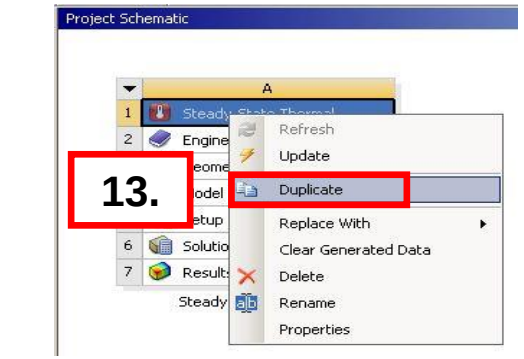
Results for polyethylene model.

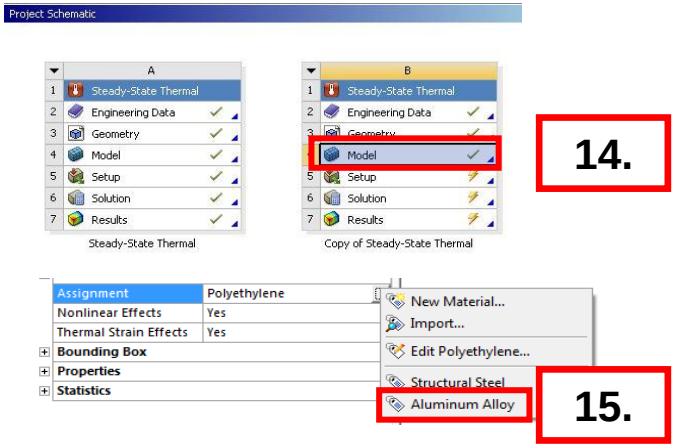




Model B Setup

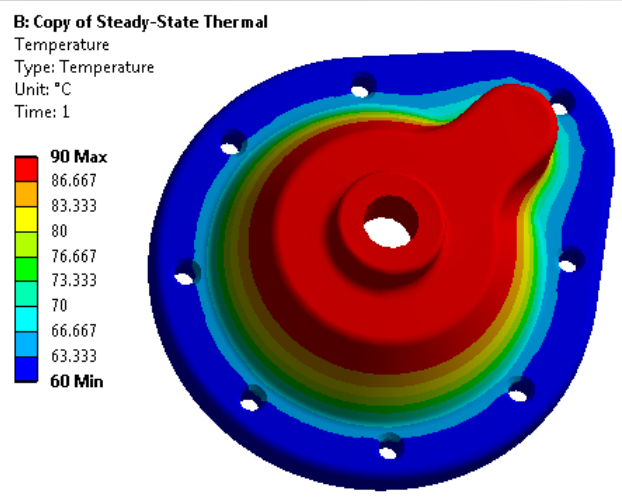
13. From the project schematic RMB in the A1 cell and “Duplicate”.
14. Double click the Model branch in the second (B) system
15. When the new model opens change the material to “Aluminum Alloy” as in step 8.
16. Solve the model.

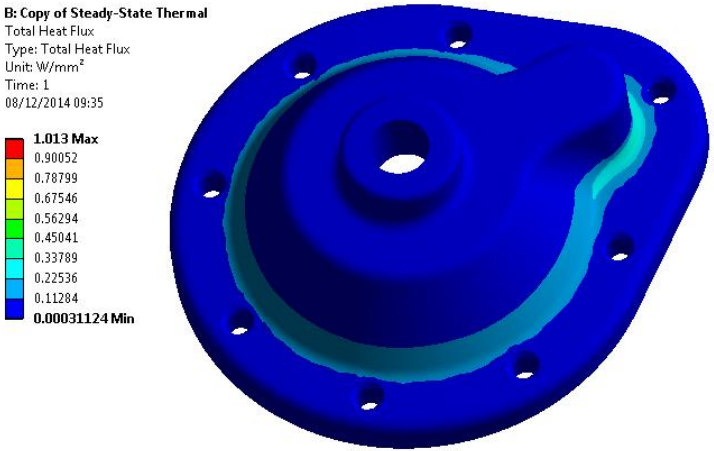




Solution – Model B

Results for aluminum alloy model.

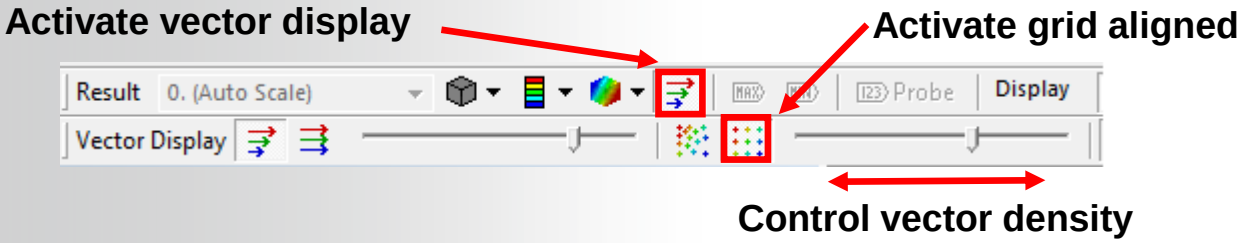


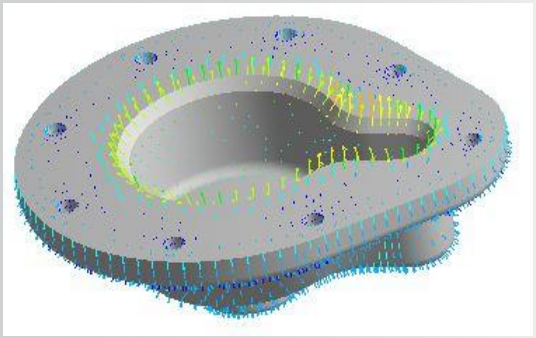


Postprocessing

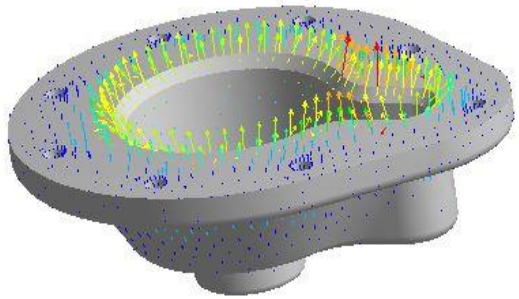
Compare Heat Flux:

- Highlight the “Total Heat Flux” results from each model and switch to vector display mode.





Polyethylene

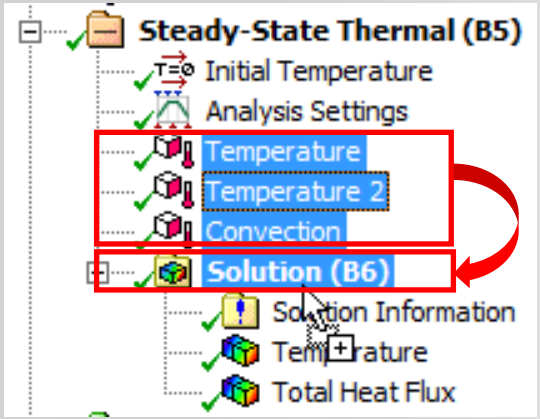


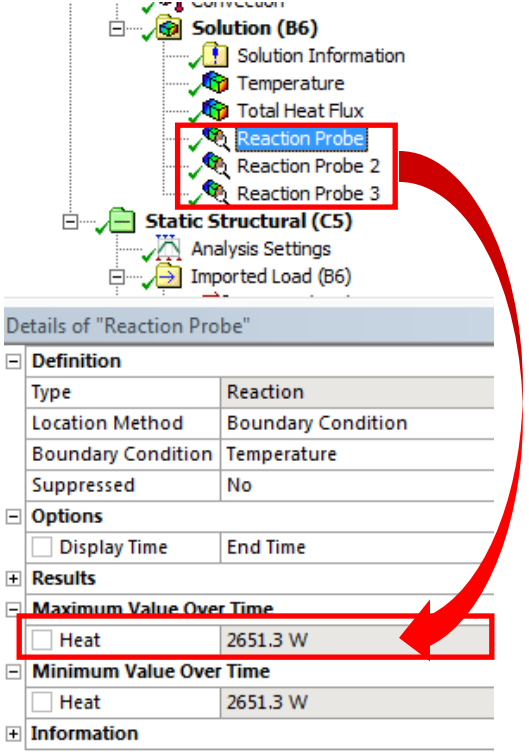
Aluminum

. . . Postprocessing

Heat balance

- Drag and drop the three boundary conditions to the solution and verify the heat balance.





Go further!

At the end of steady state thermal analysis, we know the temperature fields in the housing. Now, create a thermal-stress analysis in order to evaluate the stress level and thermal strains in the part.