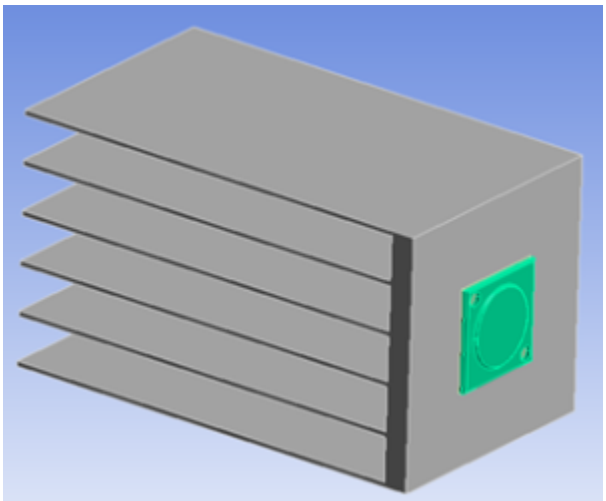
Ansys Heat Sink Analysis

Problem Specification

Consider the heat sink shown in gray in the image below. It is used to dissipate the heat from an LED and a thin interface layer shown in green below. The CAD file containing the LED + interface + heat sink can be downloaded here.



- LED and heat sink are made of Aluminum: k=186 W/(m C)
- For the interface: k=48 W/(m C)
- Heat generated by LED: 3.3×10^7 W/m^3
- Ambient temperature: 40 C
- Convection coefficient for heat sink surfaces exposed to air: 14 W/(m^2 C)

Find temperature and heat flux distribution in the model. Verify results by checking overall energy conservation and refining the mesh.

Pre-Analysis

Model Setup

Start Ansys Workbench by typing Workbench in the Start menu or click Workbench app if using Apps on Demand (aka Amazon Appstream).

Numerical Results

Energy Conservation Check and Mesh Refinement

If you are using the free Ansys student product, you won't be able to get the "convergence" object covered in the video below to solve since the number of nodes exceeds the maximum allowed in this version. A workaround is to generate a second mesh using a body sizing of 0.005m and compare the maximum temperature and heat flux on this mesh and the original mesh.

Geometry Modification

Go to all ANSYS Learning Modules