Ansys Fluent Getting Started
(New Fluent Experience)

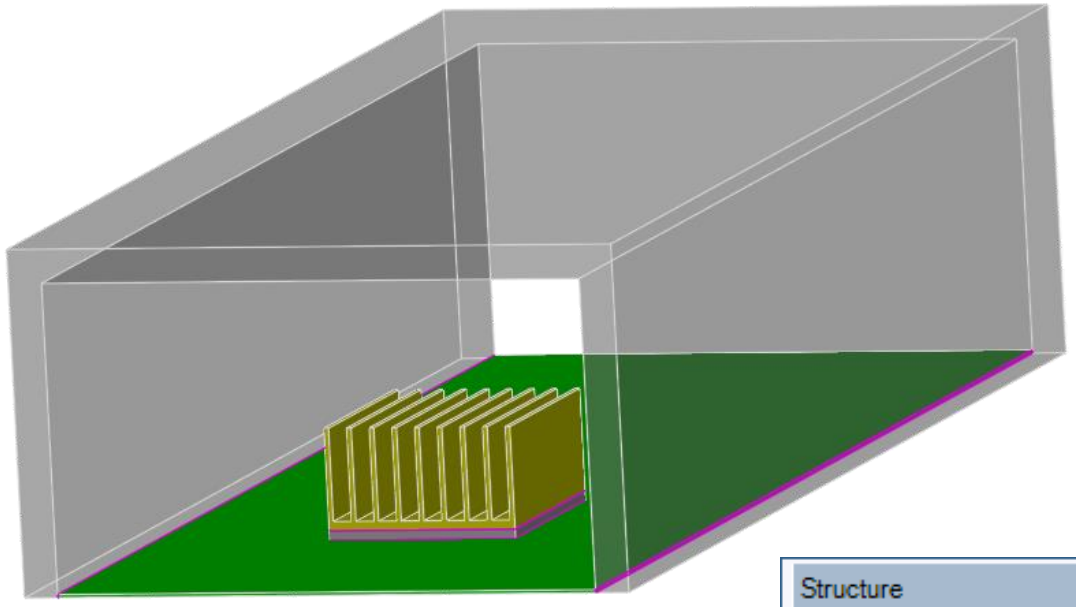
Workshop: Heat Sink

Release 2021 R1

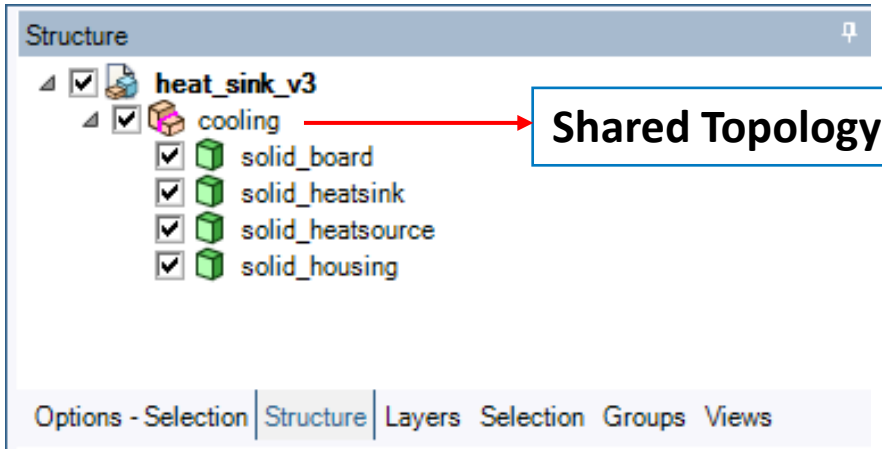
Capability Level

- This tutorial is supported by all licensing capability levels

Geometry

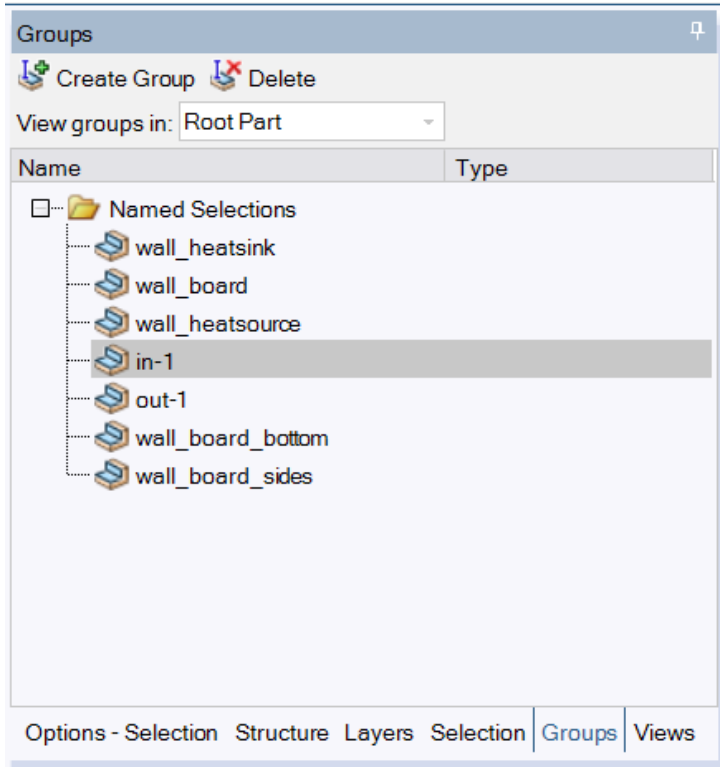


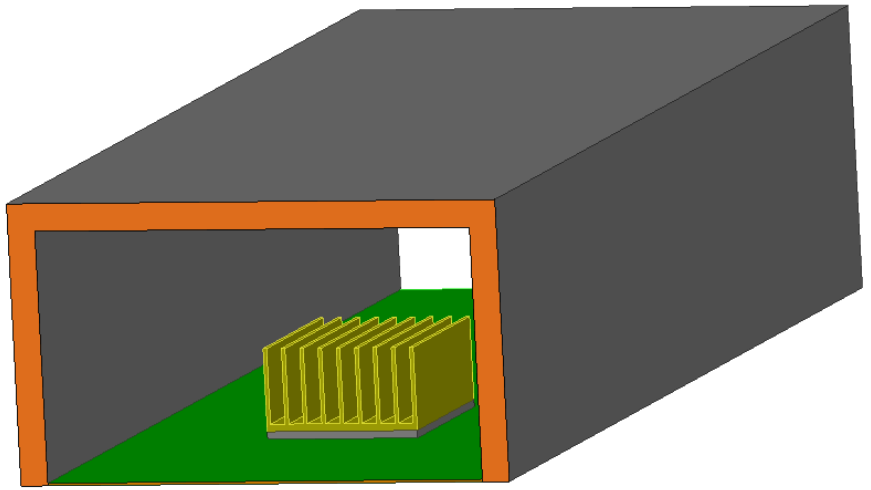
File: heat_sink_v3.scdoc, located in FLU_GSN_2021R1_EN_WI04.1_Workshop_Input_Files.zip



Shared Topology – shared items highlighted in graphics

Named Selections (NS)

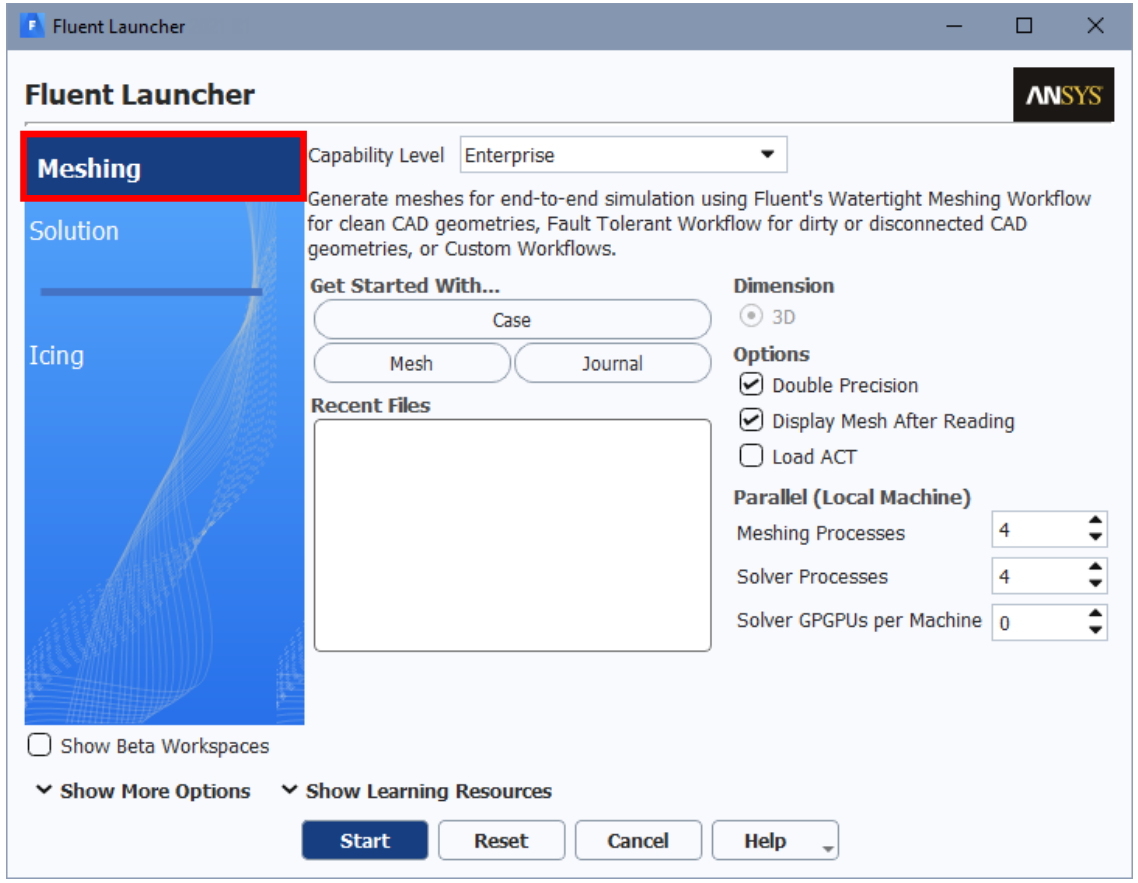




Important surfaces have been defined as named selections in the Groups panel for easier identification when the model is imported to Fluent.

Starting Fluent

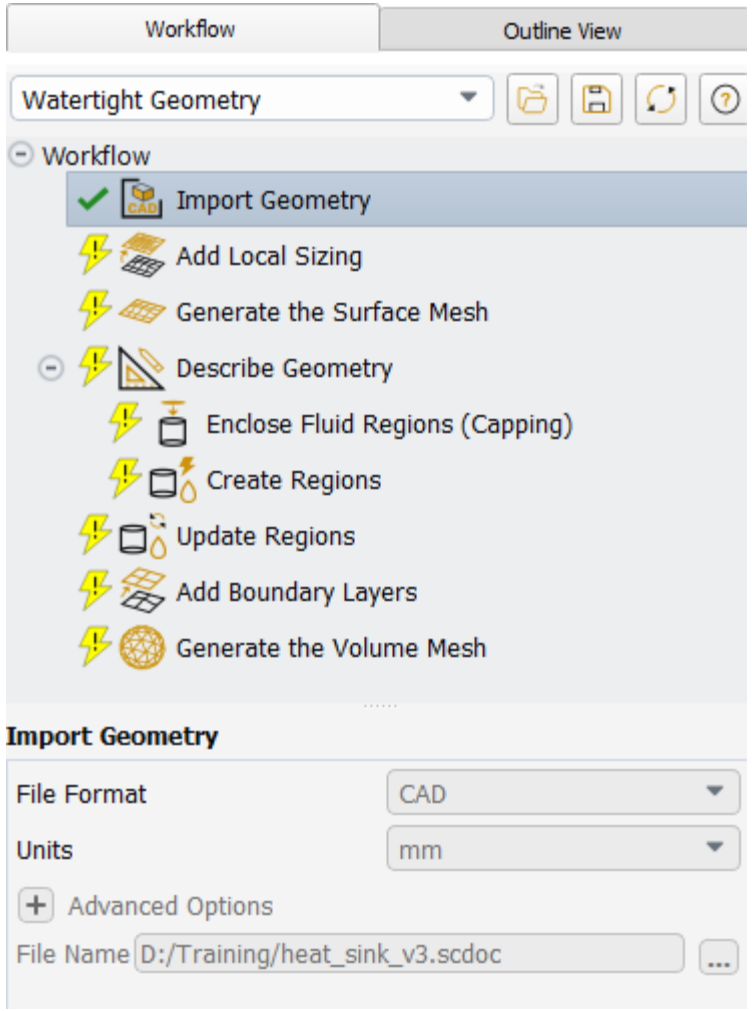
- Open the Fluent Launcher Window and ensure Meshing Mode is selected



4 Solver Processes and 4 Meshing Processes recommended for this workshop but you may need to use fewer based on availability and license status.

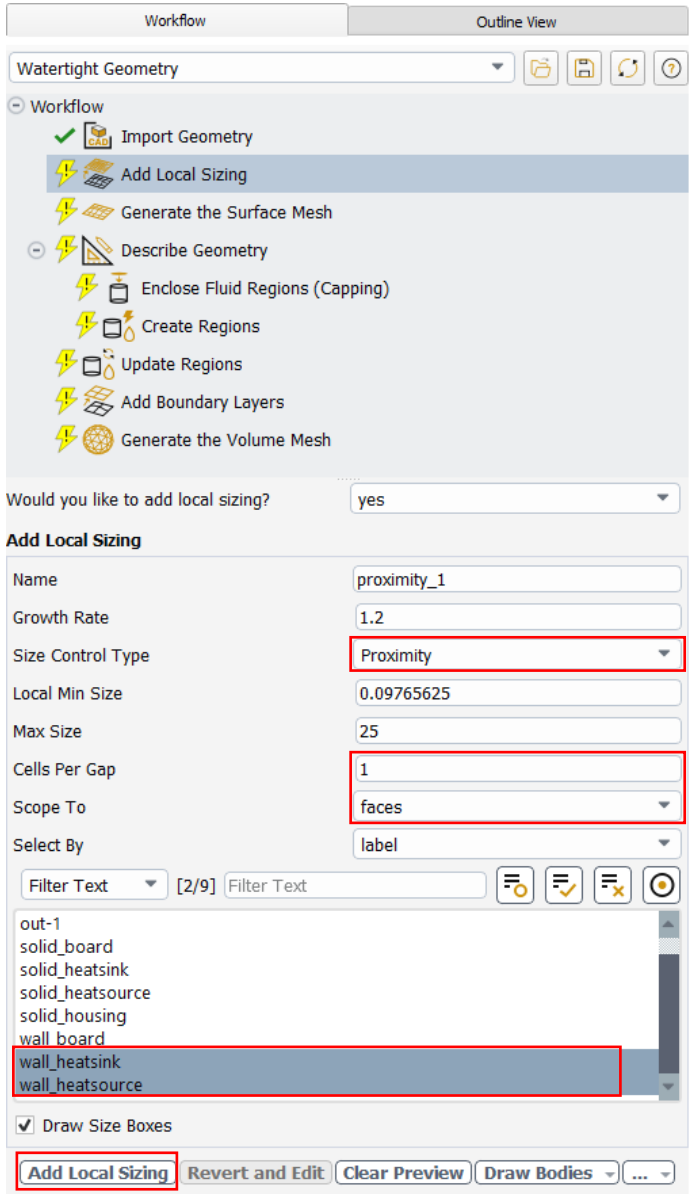
If necessary, expand "Show More Options" to change the working directory.

Import Geometry



- **Use mm for Units**
- **The name of the workshop input file is heat_sink_v3.scdoc**

Add Local Sizing

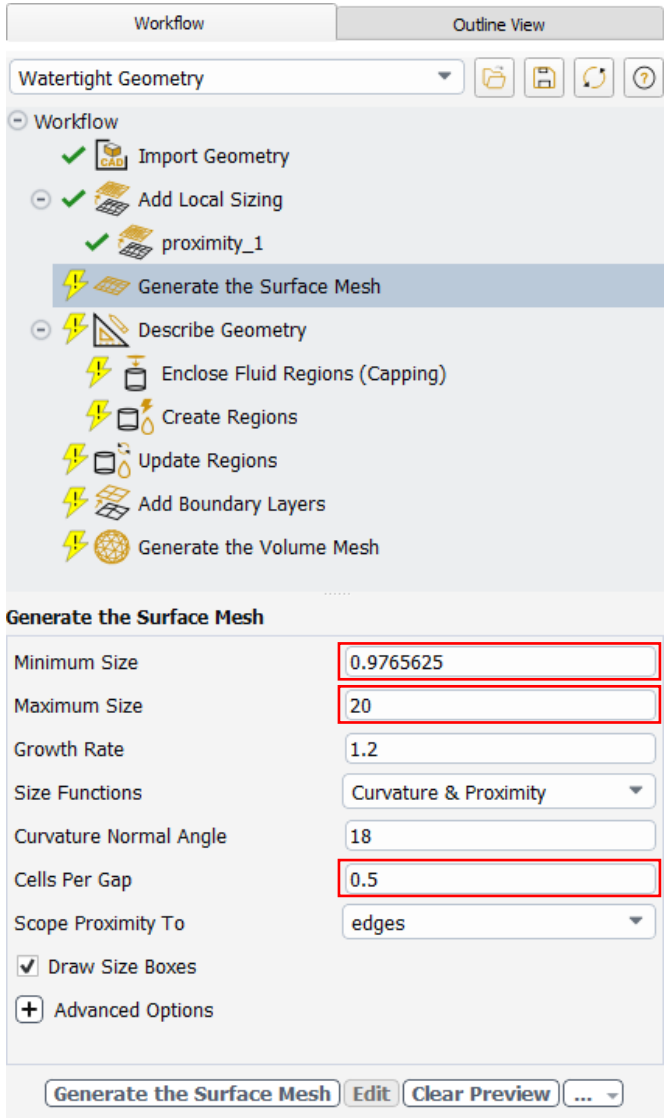


The mesh created in this workshop will be used later in the course in the workshop on heat transfer. Ideally there would be more than one computational cell across the thickness of the fins (e.g. cells per gap) and other solid parts, but that would result in a mesh with a very high cell count, such that, in a training environment, it would take too long to solve the problem. Using one cell per gap is a compromise which allows a coarser mesh with a lower cell count, which can be solved in a reasonable amount of time within the context of a training workshop.

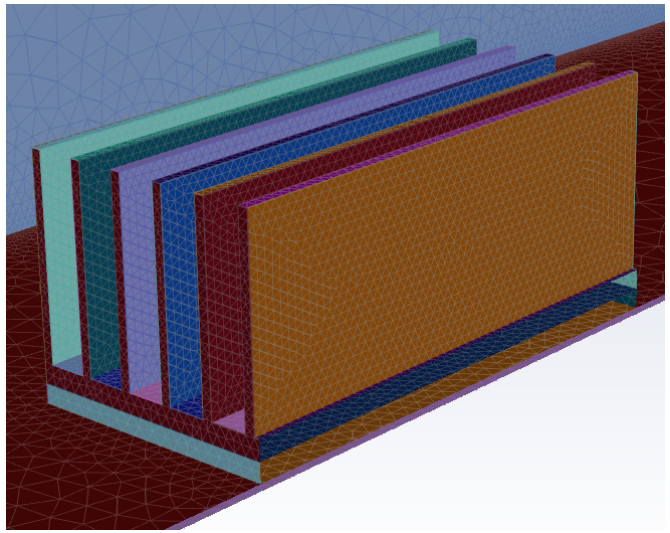
- **Size Control Type: Proximity**
- **Cells Per Gap = 1 (see discussion in blue text)**
- **Scope To = faces**
- **Face Zone Labels: wall_heatsink & wall_heatsource**

Because there is 1 cell per gap, the surface mesh size on the selected labels will be approximately the same as the fin thickness, which is 3 mm. Thus the default values for local min size and max size will not have any influence on the mesh and they can simply be left as default.

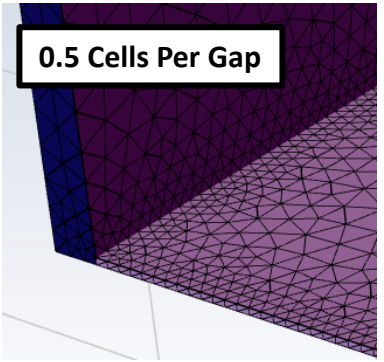
Generate the Surface Mesh

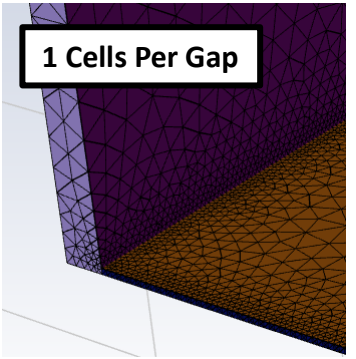


- **The minimum size is acceptable**
  - **There are no geometrical features which would cause the mesh to need to be smaller, so it is unlikely to have any influence on the mesh in this model**
- **Reduce the maximum size to 20 mm**
  - **(the enclosure is ≈ 200 mm across, 20 mm is 10% of that)**
- **Set the cells per gap to 0.5**
  - **This results in a slightly coarser mesh in the corners of the enclosure which will reduce the cell count for this training exercise**
  - **The figure on the right shows the mesh that would result from 1 cell per gap – such a refined mesh is not required for this problem**
- **Generate the surface mesh**
  - **Note the maximum skewness value reported in the console window … values below 0.7 are good**

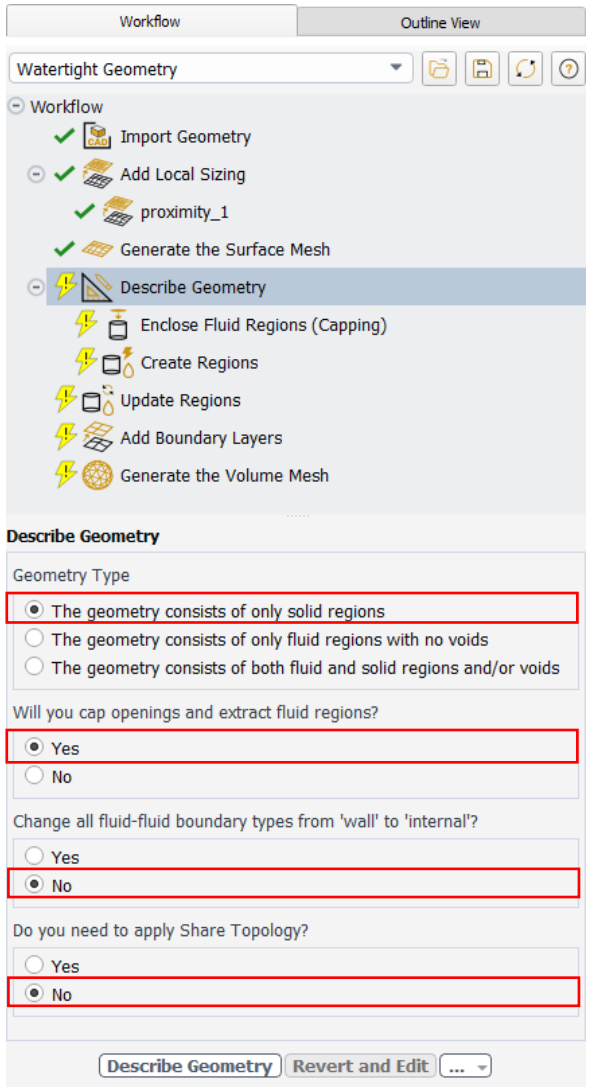


Use a clipping plane to examine mesh near the heat sink.



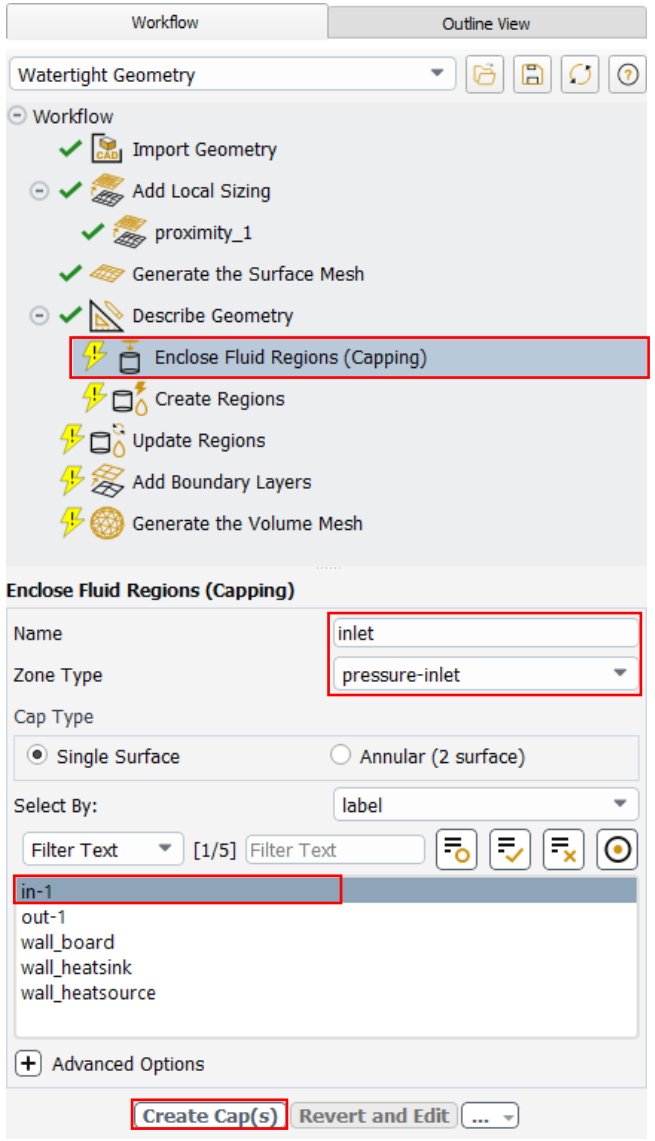


Describe Geometry

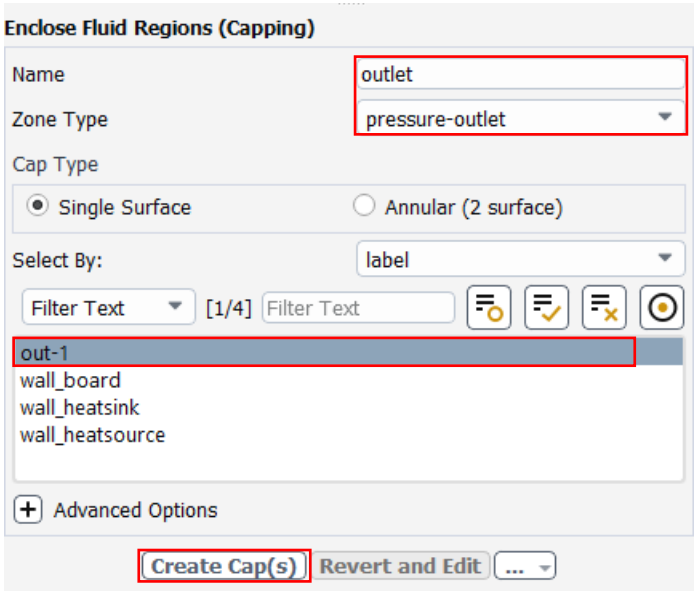


- **The geometry consists only of solid regions.**
- **Capping surfaces will be defined to allow flow volume extraction.**
- **There will be no fluid-fluid boundary types so the setting will have no effect in this model.**
- **Share Topology was applied in SpaceClaim**

Enclose Fluid Regions

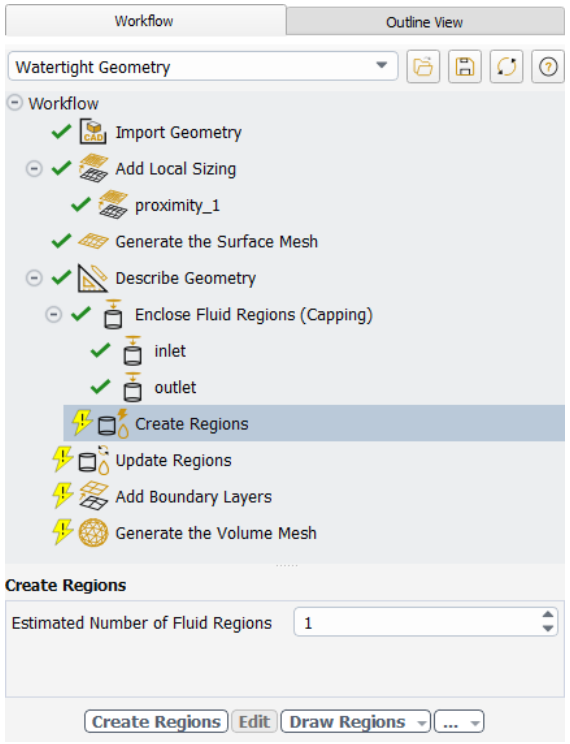


- **Leave the Select By: filter setting as "label"**
- **Select "in-1"**
- **Enter name of "inlet" and set Zone Type to "pressure-inlet"**
- **Click Create Cap(s)**

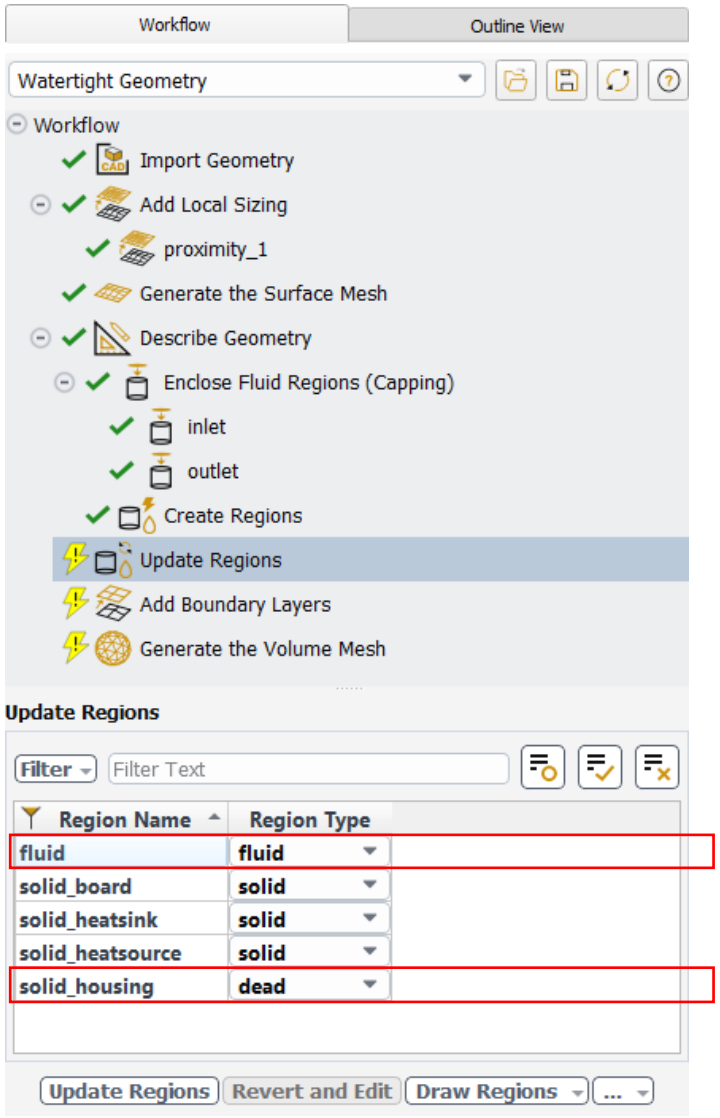


- **Similarly, select "out-1" and define a pressure-outlet named "outlet"**

Regions



- **The estimated number of fluid regions is correct so click Create Regions**

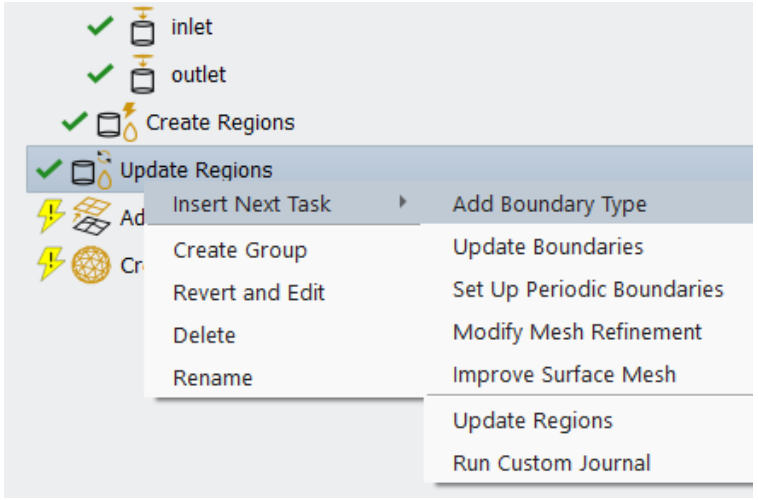


1.

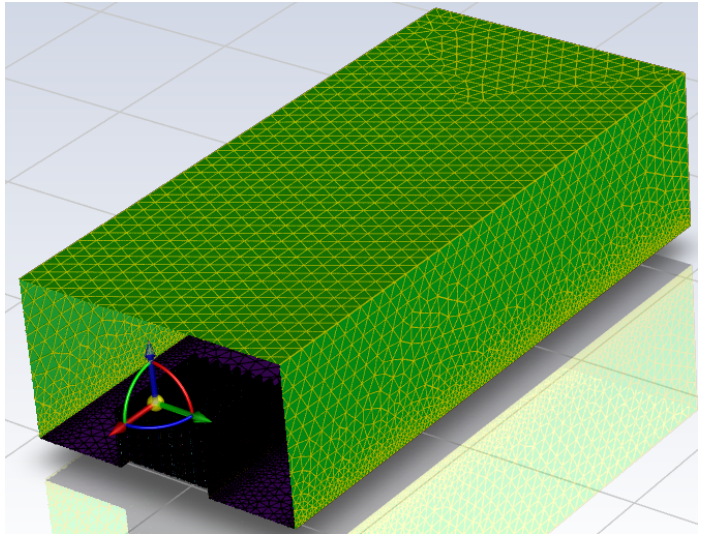
2.

- **In the Update Regions task:**
- **Change the name of the fluid region to "fluid"**
- **Change the Region type of "solid_housing" to "dead" because it is not desired to include it in the simulation.**

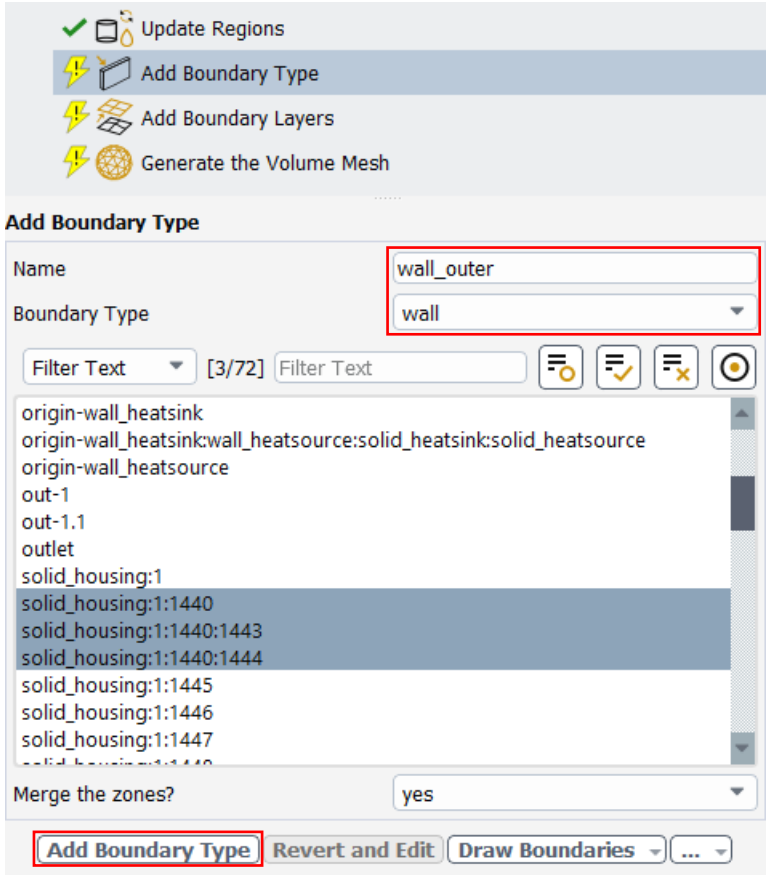
Add Boundary Type for Outer Walls



- **Before creating the volume mesh, right-click on Update Regions and insert an "Add Boundary Type" task.**



- **Select the three outer surfaces of the fluid zone by right clicking in the graphics window.**

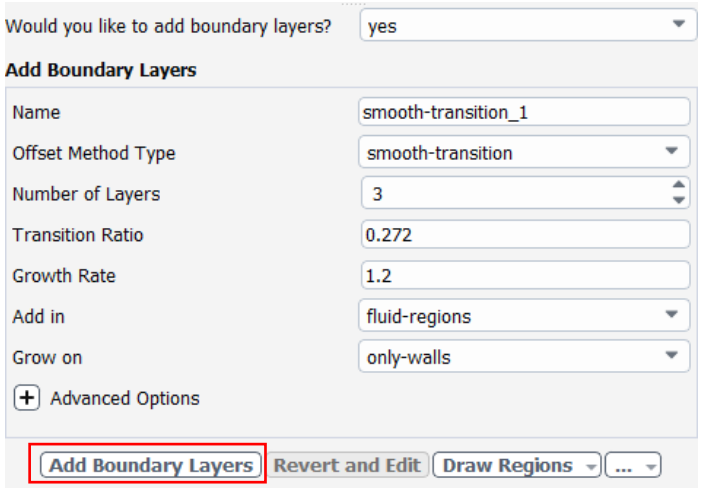


- **Set the boundary type to wall, enter a name of "wall_outer" and click Add Boundary Type.**

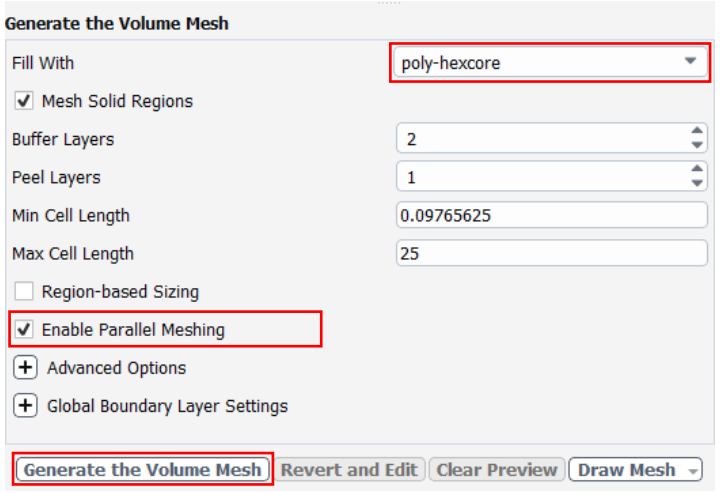
It is not a requirement to add the outer walls as a boundary type. It simply makes the case set up easier to understand after you go to solution mode.

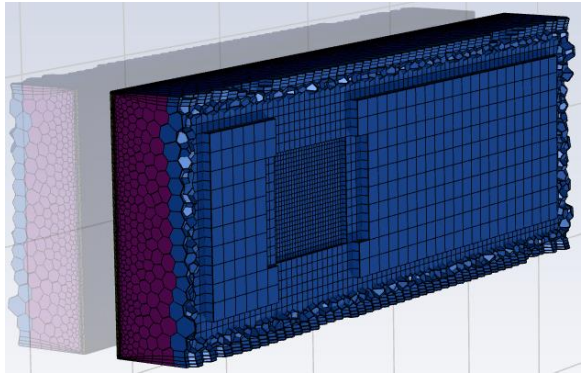
Add Boundary Layers and Generate the Volume Mesh

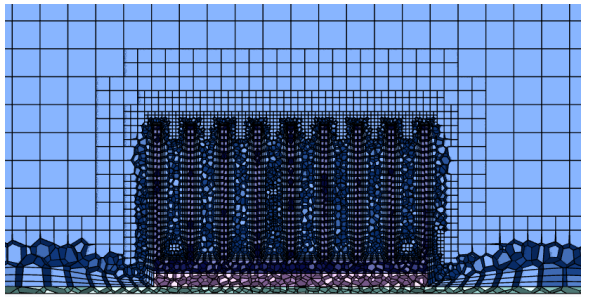
- In the Add Boundary Layers task, keep the defaults and click Add Boundary Layers
- In the Generate the Volume Mesh task choose poly-hexcore, and if more than one meshing process was selected in slide 5, ensure Enable Parallel Meshing is checked
  - On completion of the volume mesh, Fluent reports the minimum orthogonal quality of the mesh in the console window. Review the console output to ensure this value is 0.1 or higher



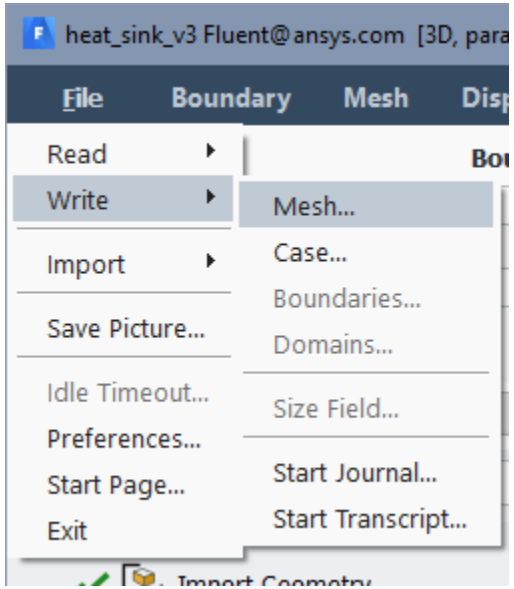
The default value of 3 is used here as larger values might be problematic in the small gaps between the fins







Write Mesh



- Go to File > Write > Mesh and save the file as heat_sink_v3_volume_mesh.msh.gz
- The mesh will be used later in the heat transfer workshop.
- Exit Fluent when finished.

End of presentation