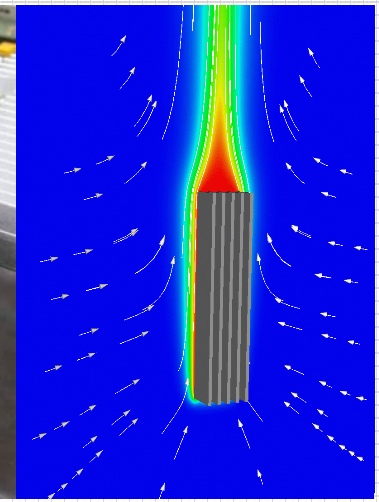
Day 1 Fluent Workshop: Heat Sink Design



https://github.com/anjurgupta/Fluent-Heat-Sink-Workshop

A.R. Gupta
MIME 3440: Heat Transfer

Acknowledgement

Dr. Anju R Gupta would like to thank Ansys for their generous support through the Faculty Development Grant and complimentary Ansys Granta license

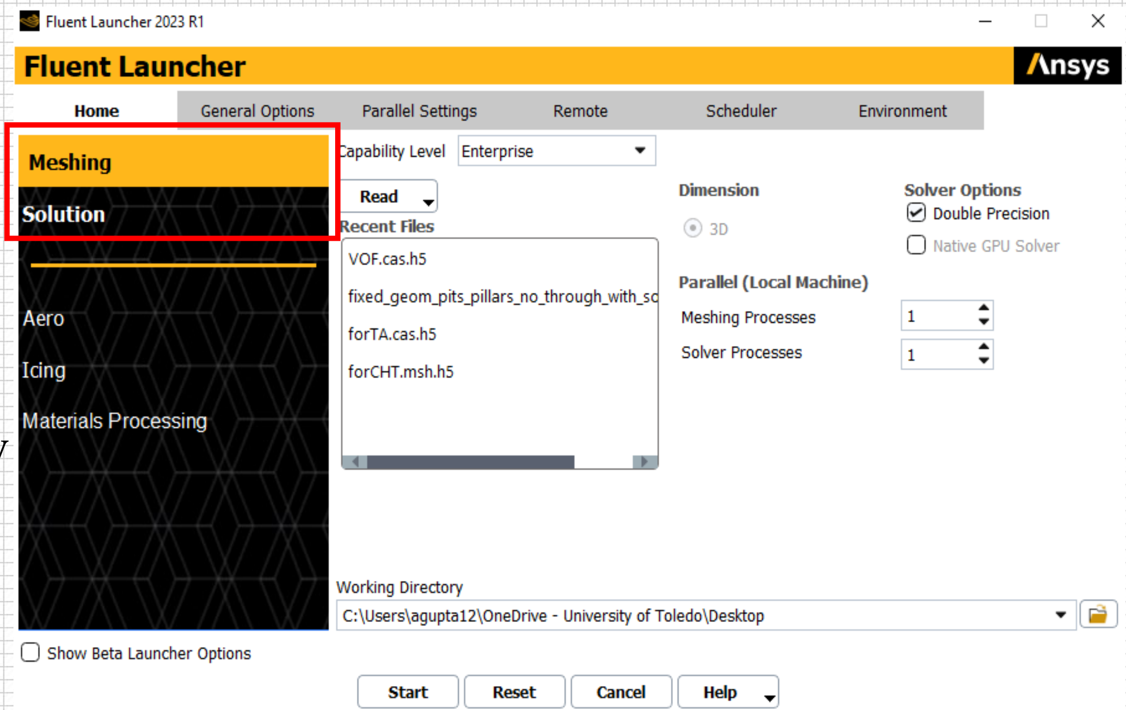
Topics

(Day 1) **Heat Sink**

- Modes of Heat Transfer
- Geometries
- Electronics cooling- Class problem 3.8

(Day 1) **Meshing**

- Geometry import and workflow
- Local sizing, surface mesh, boundary layers
- Volume mesh and quality



(Day 2) Solution

- Mesh import and workflow
- Fluent Set up: solid and fluid zones for electronics cooling- Class problem 3.8
- Check: Convergence and Visualization, Material property (Copper vs Aluminum)

Learning Outcomes

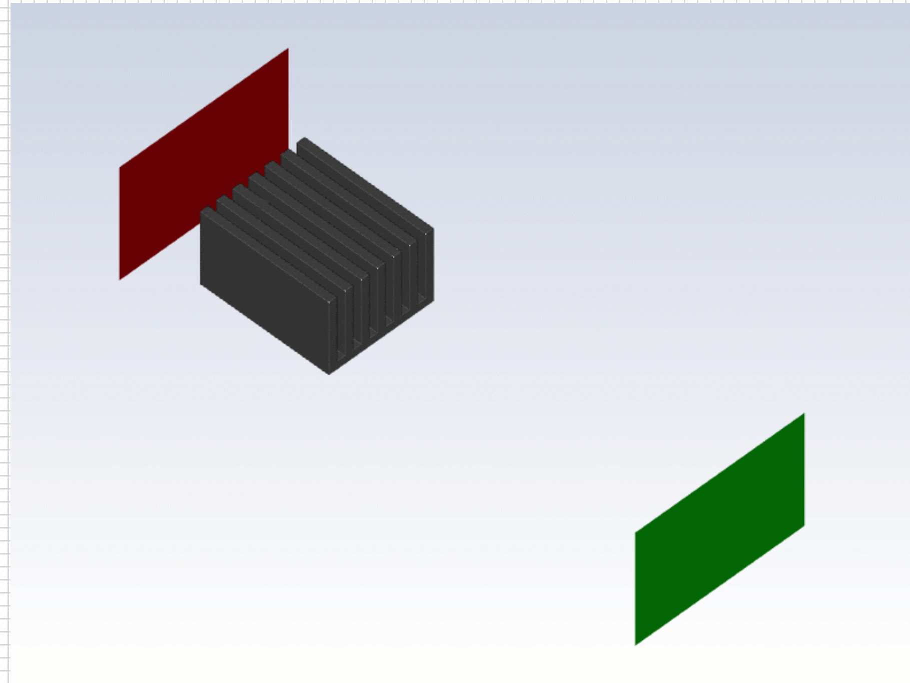
- Apply watertight geometry Meshing Workflow in Ansys Fluent to create volume mesh
- Design high quality surface mesh using imported CAD or mesh geometry by adding multiple unique local sizing functions
- Examine capping surfaces for fluid regions when the imported CAD geometry only includes solid parts
- Define boundary layers with custom and localized definition
- Generate volume mesh using polyhedra, poly-hexcore, hexcore or tetrahedral meshing algorithms
- Execute heat transfer simulation involving conduction and convection
- Perform conjugate heat transfer (CHT) simulations in which heat transfer is coupled between solid and fluid zones.

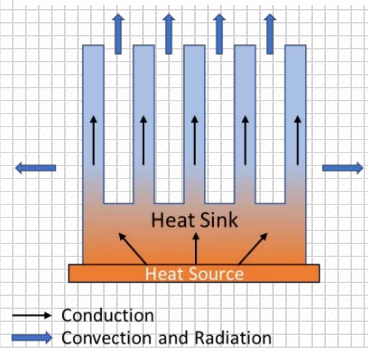
Heat Sinks

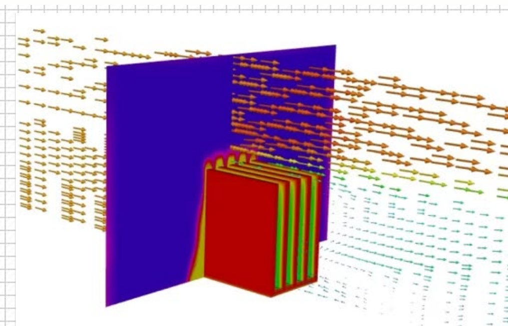
- Heat Sinks are **a type of heat exchanger** that used conduction, convection and radiation to transfer heat from a heat-producing device or heat source into a surrounding fluid
- Heat sinks are commonly made from metals, commonly copper or aluminum
- **Example:** to avoid overheating and system failure in electronics such as CPU
- Electronic components, such as chips and microprocessors generate heat, and proper thermal management plays an important role in improving their longevity and reliability


Heat Sinks: Modes of HT

- The heat generated by the CPUs gets transferred to the heat sink through **conduction**
- Then released into the air via **convection** through the surface of the heat sink
- Designing heat sinks to reliably transport thermal power away from the electronics
- Heat sinks are practically used in applications involving either **natural or forced convection**

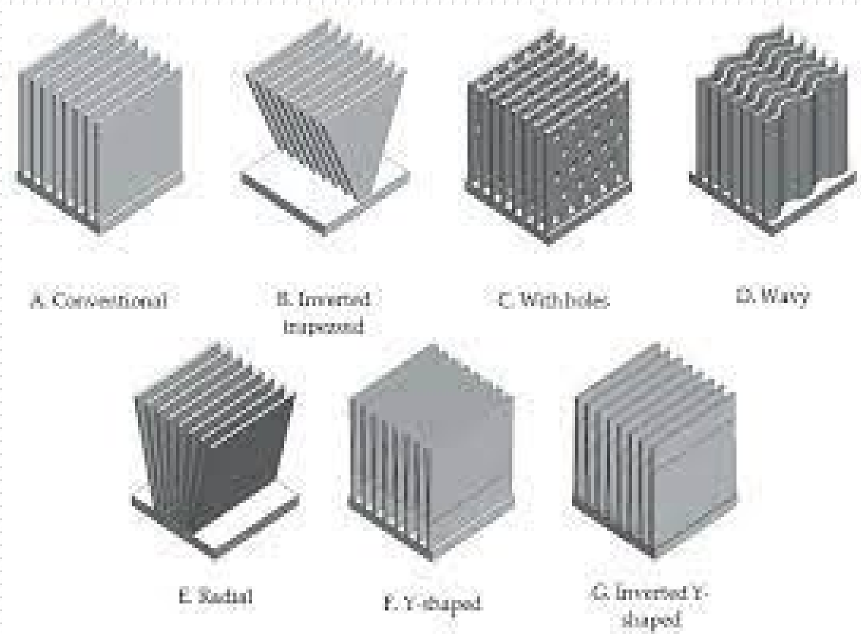


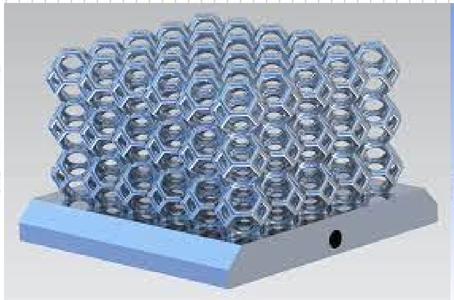
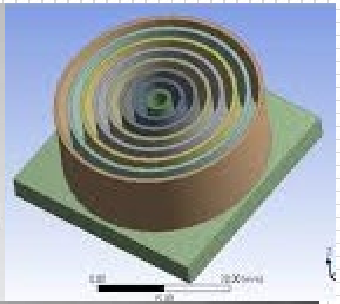




Heat Sinks: Fins

- The fins of the heat sink increase the total surface area and improve its performance when cooling electronic components
- Here are some examples from google images

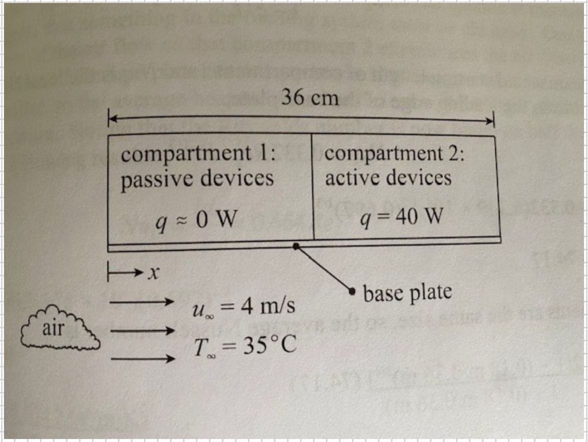



Electronics Cooling using Heat Sink (Example 3.8)

The electronics communications system consists of two adjacent compartments as shown. The compartments share a common base plate that functions as an air-cooled heat sink. The compartment that experiences the air flow first contains passive devices that dissipate a negligible amount of heat. The other compartment contains devices that contribute to a total heat dissipation of 40 W, all of which transfers into the heat sink. Atm air at 35°C is forced over the base plate at a velocity of 4 m/s. The compartments are the same size, and the overall size of the base plate is 36 cm x 36 cm. Find the average surface temperature of the base plate.

Properties: atm air: k = 0.0312 W/m.K,
$\nu = 22.68 \times 10^{-6}\ m^2/s$
Pr = 0.697



Electronics Cooling using Heat Sink (Example 3.8)

Designs to Simulate

- Flat plate (no fins)
- Plate with rectangular fins (user-defined spacing, height, width)
- Plate with triangular fins (same height and base as rectangular, material saving)

Goal:

Simulate, quantify, and compare the average and maximum base plate temperature for each configuration.

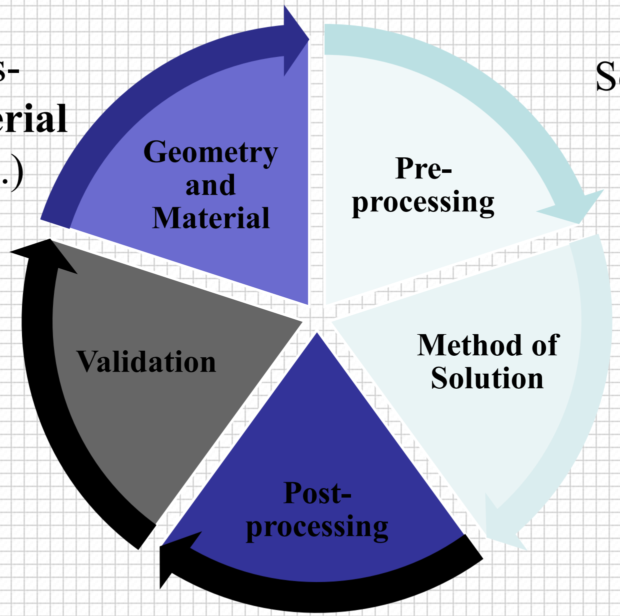
Recommend a design that maintains electronics below the target temperature, with considerations for material use and manufacturability.

Refer to https://github.com/anjurgupta/Fluent-Heat-Sink-Workshop for the design files

Simulations/Digital Twinning

- Simulation is the process of virtually emulating a physical process
- It is used to verify the efficacy of a design or product

Geometry (e.g. shape, cross-sections, assemblies) and **material** (e.g. steel, rubber, wood, etc.)



Setting up the model through **mesh generation** and **boundary conditions**

Numerical methods used to solve the model

Evaluation of outputs (e.g. displacement, stresses and strains)

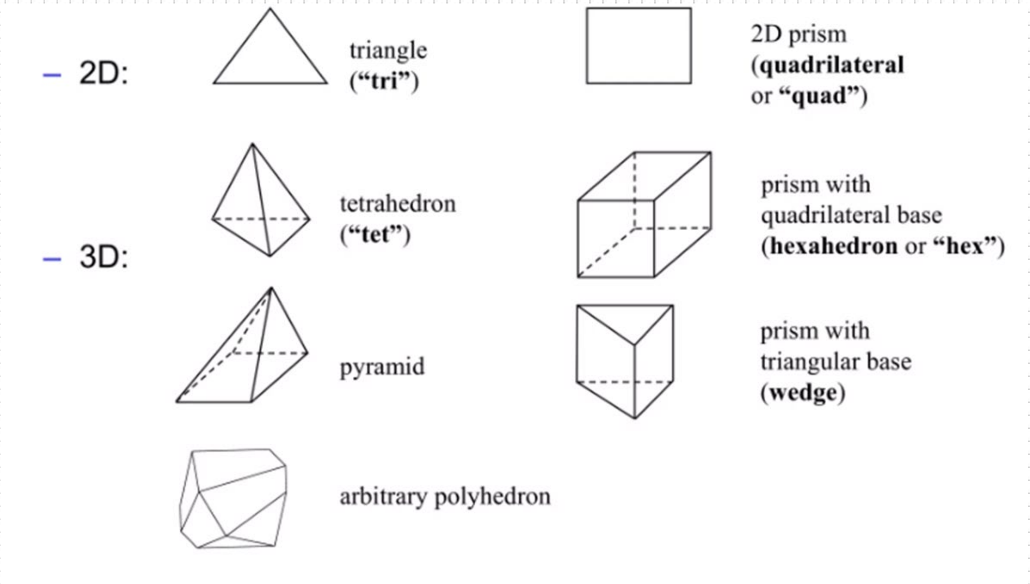
Sanity check (i.e. do the results match the expectation, are the results plausible?) E.g. Does the output indicate a need in change of geometry or material?

Meshing

- Two-dimensional and three-dimensional grid; it is dividing complex geometries into elements that can be used to discretize a domain

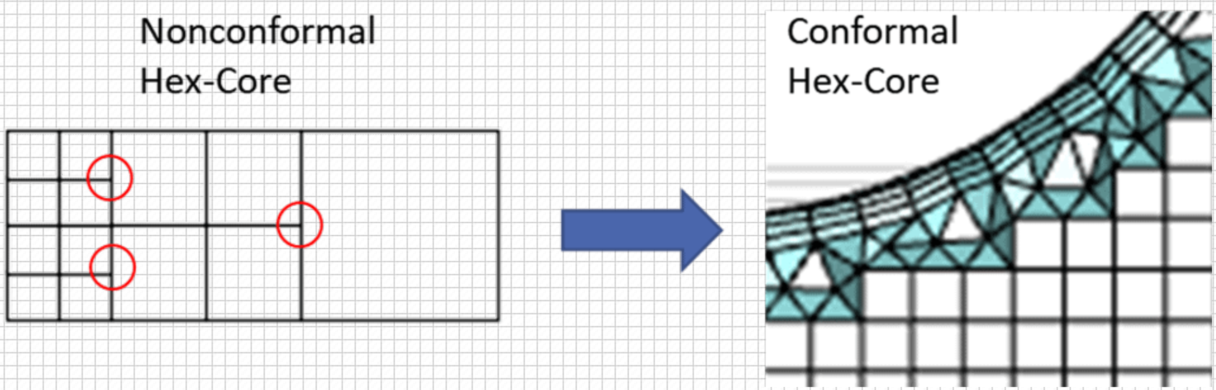
- Mesh elements allow governing equations to be solved on predictably shaped and mathematically defined volumes.

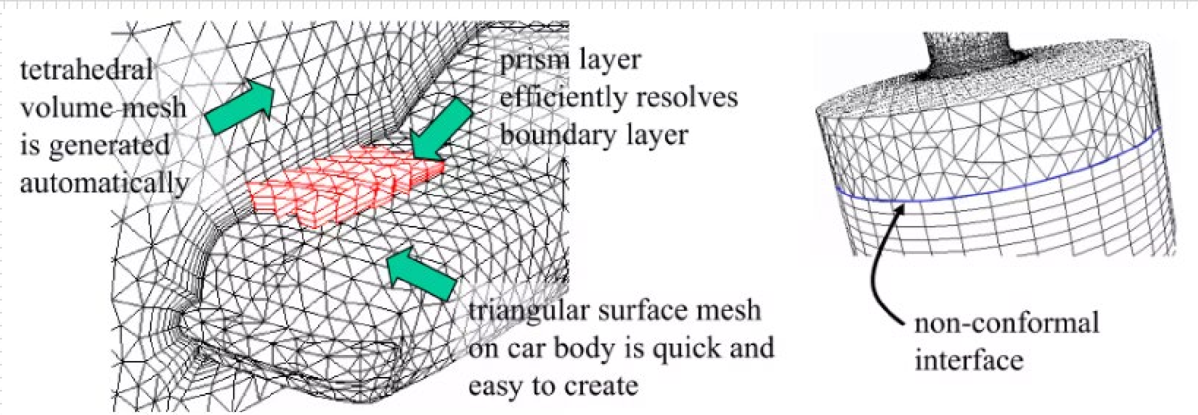
- Typically, the equations solved on these meshes are partial differential equations

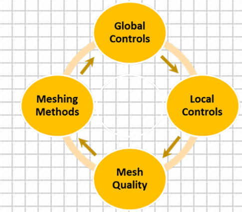


Meshing: Conformal vs Nonconformal

- **Conformal:** conformal meshes match every node to a node in the adjacent cells
- **Nonconformal:** a coarse solid mesh interface with a fine fluid mesh







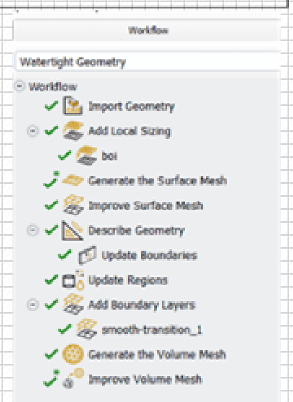
Meshing Workflows

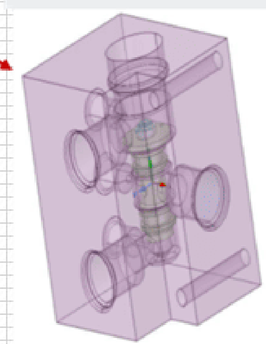
Import geometry → Surface mesh → Create solid/liquid regions → Volume mesh → Solution

- **Watertight:** used for fluid and/or solid geometry that is relatively clean and watertight.
- **Fault-Tolerant:** used for complex geometries with many parts

Watertight

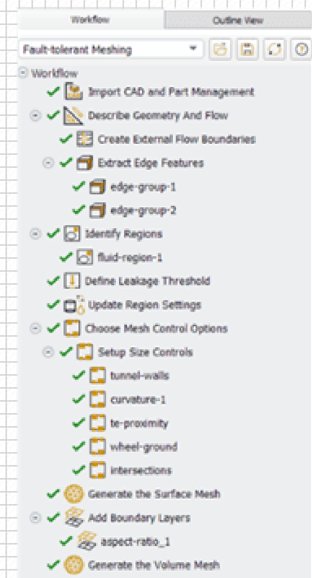
- Most commonly used
- Relatively clean- does not require clean-up and modifications
- Watertight solid and/or fluid regions
- Single body: any imported CAD model
- Multi bodies: share topology at the CAD level
- Disconnected bodies are allowed if one body is entirely inside another.
- Can be meshed by surface meshing and then volume filling





• Fault- Tolerant

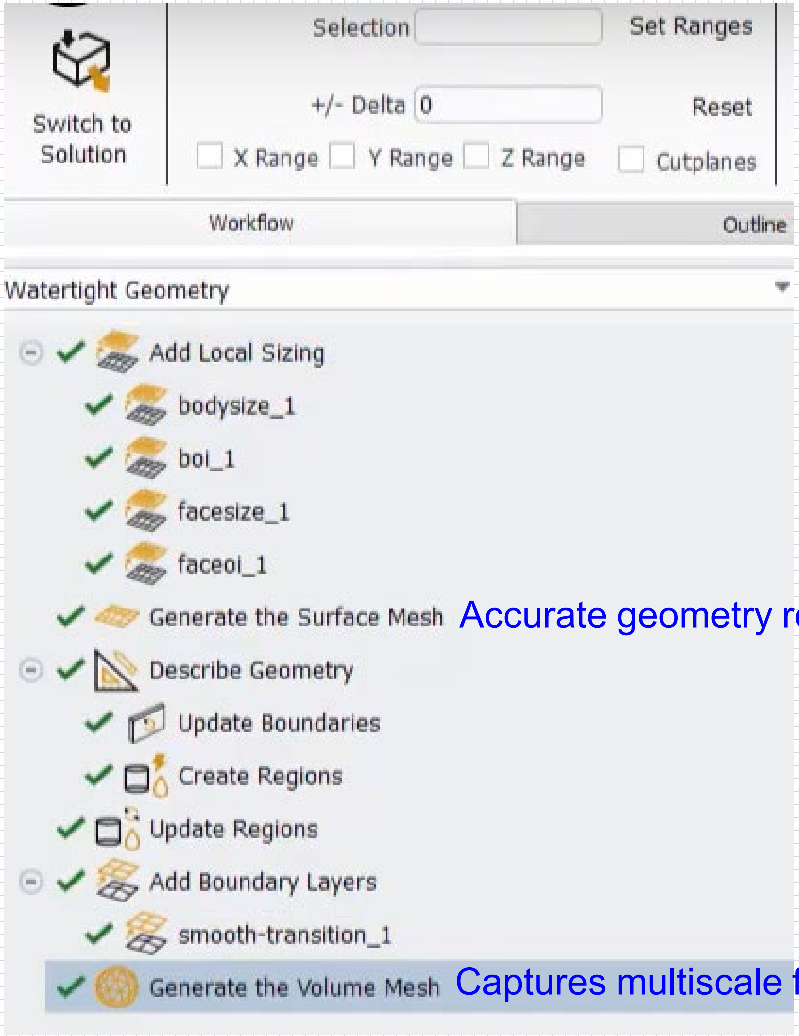
- All major CAD formats including JT, MSH, STL
- Unclosed, overlapping input CAD geometry
- Part management for assembly preparation
- Wrap-extract for external and internal flow domain
- Creation of porous blocks
- Leakage repair workflows
- Automatic offset and wake refinements
- Settings can be reverted and edited
- Non-conformal auto-pairing workflow



https://www.padtinc.com/2021/05/11/comparison-ansys-fluent-meshing-ansys-meshing-cfd/

Meshing Workflows

Import geometry → Surface mesh → Create solid/liquid regions → Volume mesh → Solution …



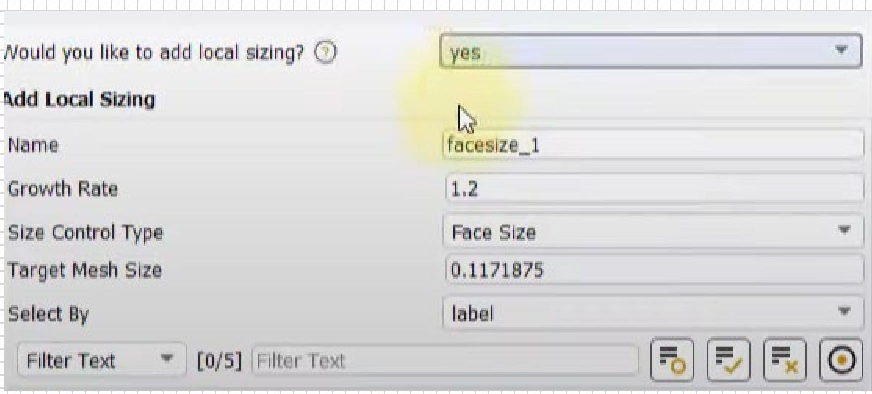
Accurate geometry resolution

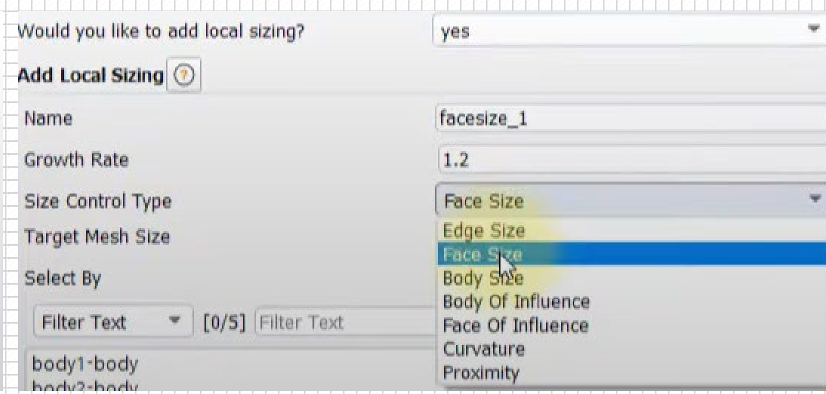
Captures multiscale flow features and flow gradients

https://www.padtinc.com/2021/05/11/comparison-ansys-fluent-meshing-ansys-meshing-cfd/

Local Sizing

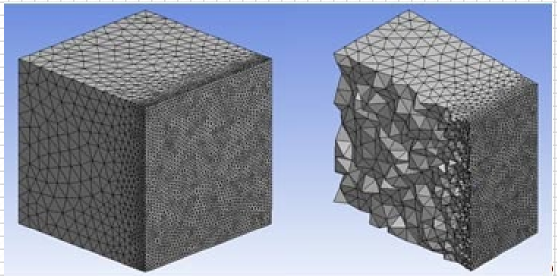
- **Controls:** Edge, Face, Body
- Mesh growth (transition) between small and large sizes based on a specified growth rate
- Curvature based refinement and angles between normals for adjacent mesh elements (curvature-based sizing)
- Number of mesh elements employed in the gaps between two geometric entities (proximity-based sizing)
- **Growth Rate:** increase in element edge length with each succeeding layer of elements from the edge or face.
- For example, a growth rate of 1.2 results in a 20% increase in element edge length with each succeeding layer of elements



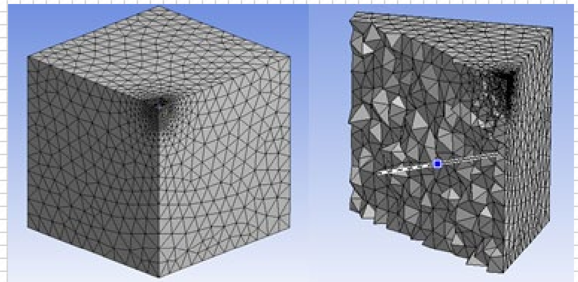


Local Sizing

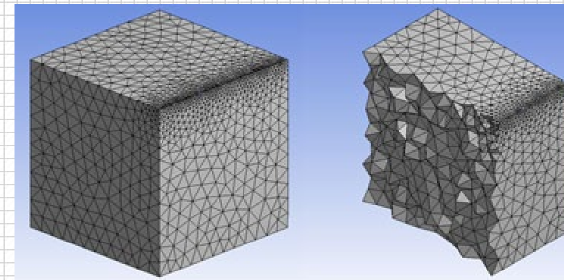
Face sizing



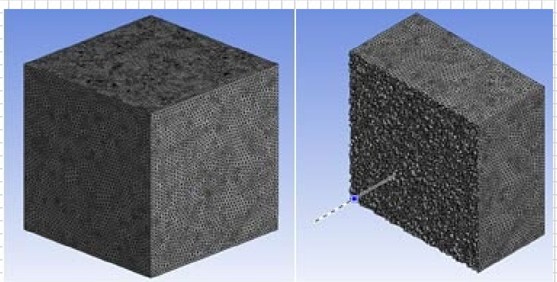
Point Sizing



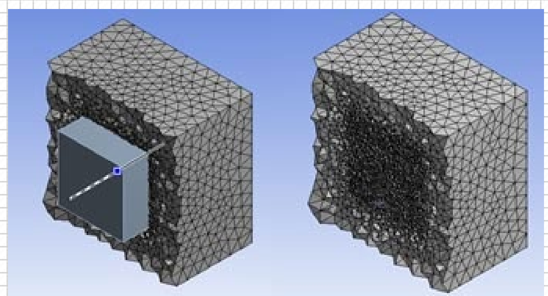
Edge Sizing



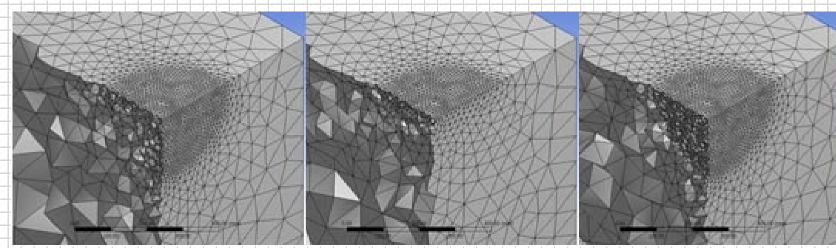
Body sizing



Body sizing using body of influence



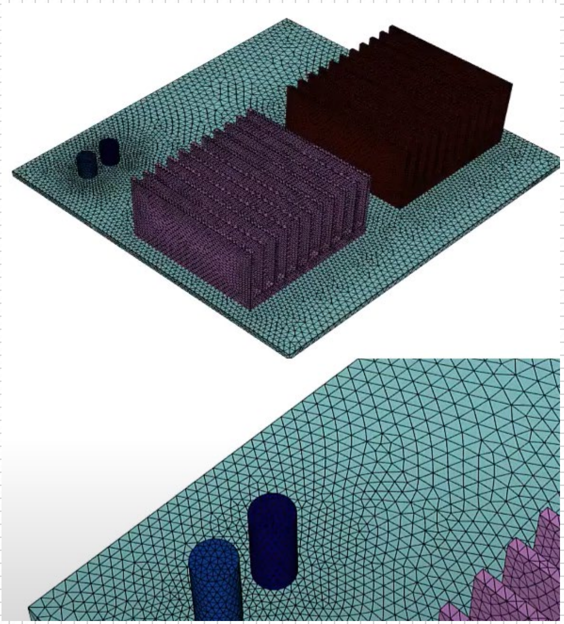
Sphere of Influence on Edge, Face and Body



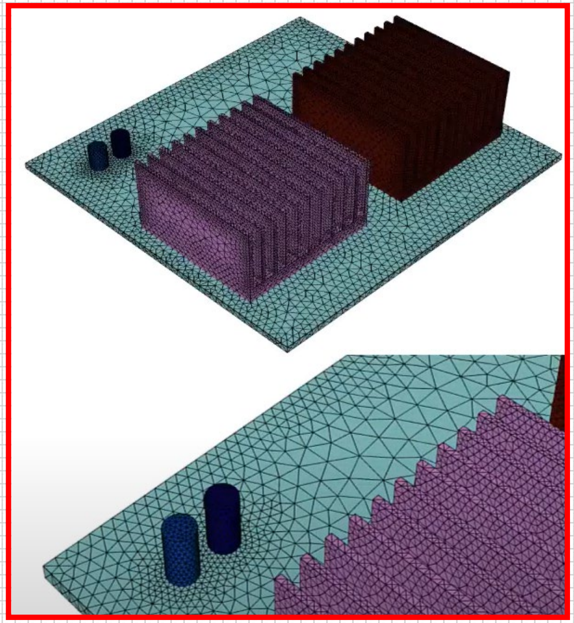
https://www.computationalfluiddynamics.com.au/size-controls/

Local Sizing: Growth Rate

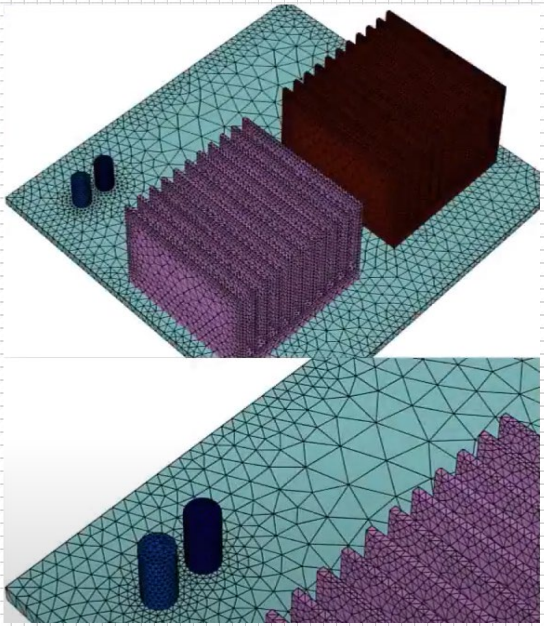
1.05



1.2

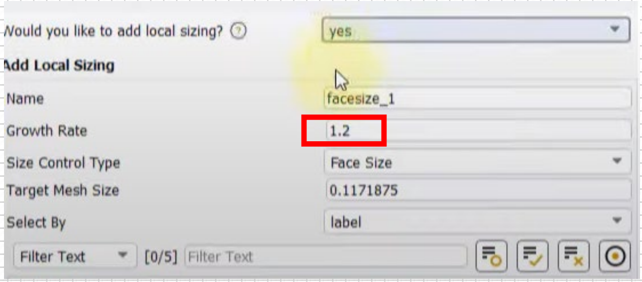


1.3



Finer mesh resolution away from the boundary

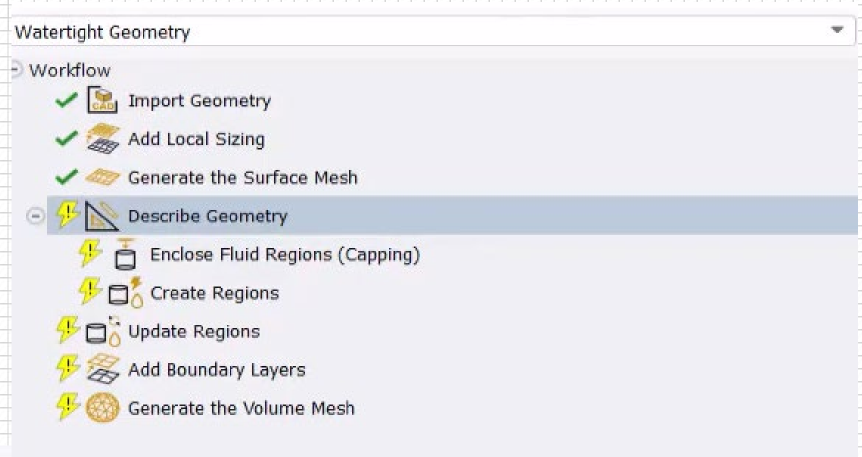
Higher growth rate coarsens the surface mesh

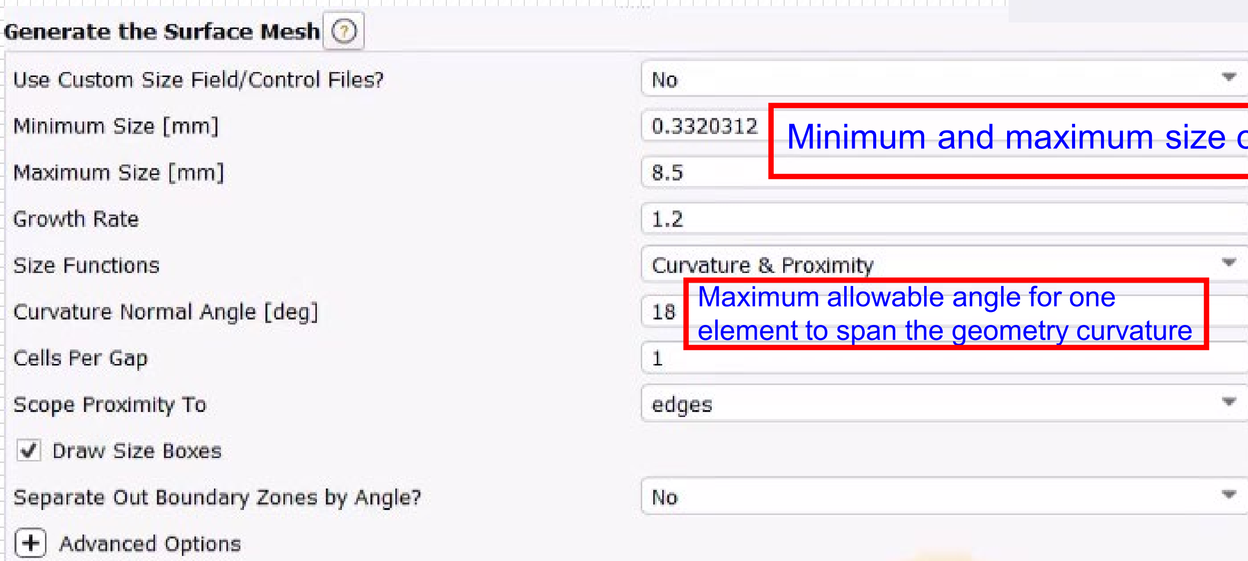


Would you like to add local sizing?	yes
Add Local Sizing	
Name	facesize_1
Growth Rate	1.2
Size Control Type	Face Size
Target Mesh Size	0.1171875
Select By	label
Filter Text [0/5]	Filter Text

Surface Mesh

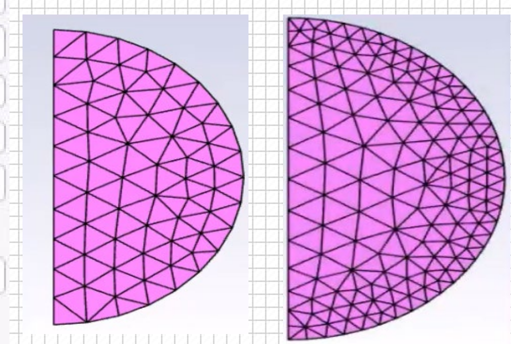
- Boundaries of the computational region
- First step for generating a high-quality mesh





Minimum and maximum size of the elements for the surface mesh

Maximum allowable angle for one element to span the geometry curvature

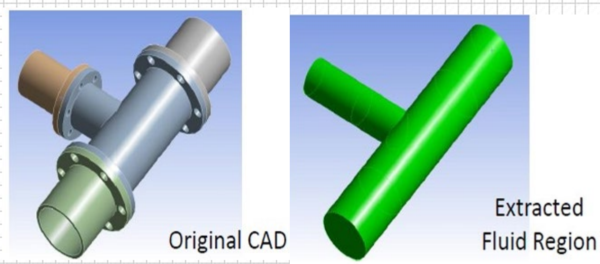


10° (180/10 =18 segments)

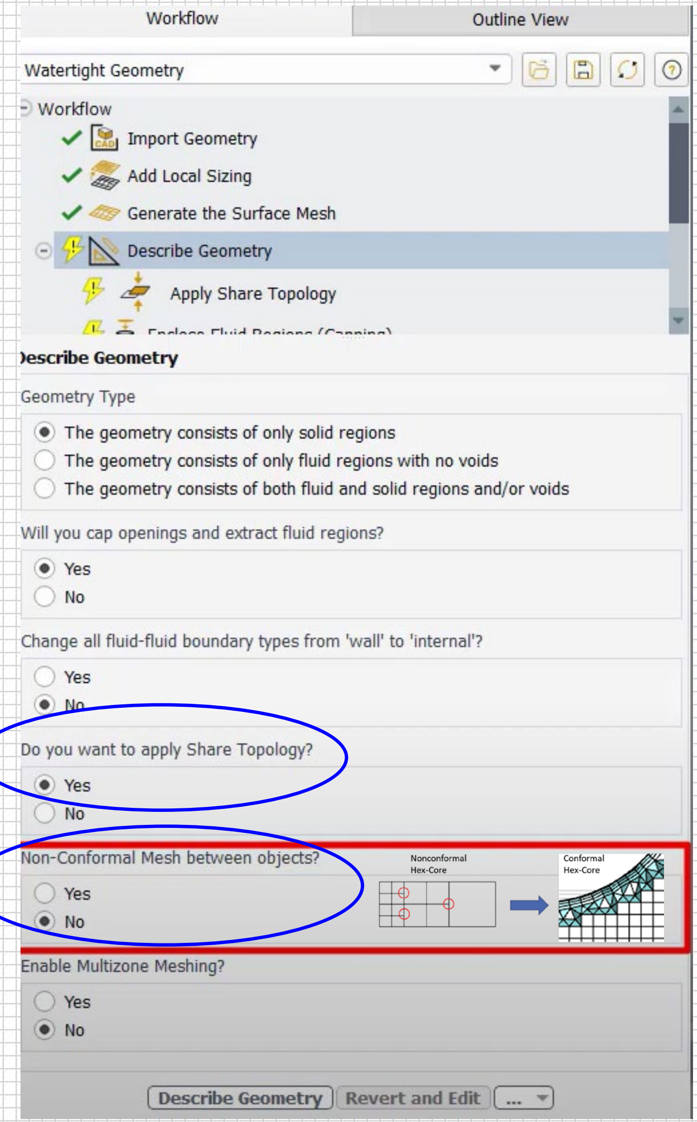
18°

Geometry: Capping and Share Topology

- **Capping surfaces**: for fluid region extraction
- **Fluid-fluid boundary**: Interface between multiple fluid regions

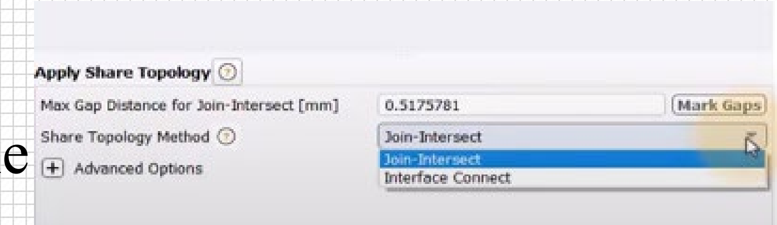


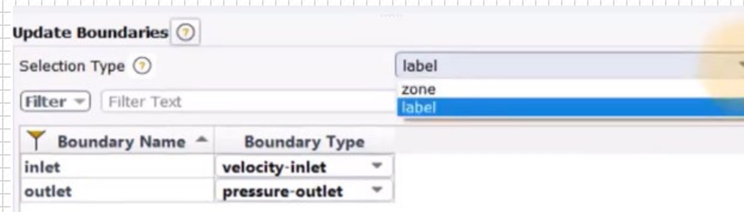
- **Share Topology**: Recommended at CAD creation phase
- Ensures overlapping areas are handled appropriately
- **Conformal mesh:** two bodies share a single face at the interface
- Generally preferred for high numerical efficiency and small simulation times



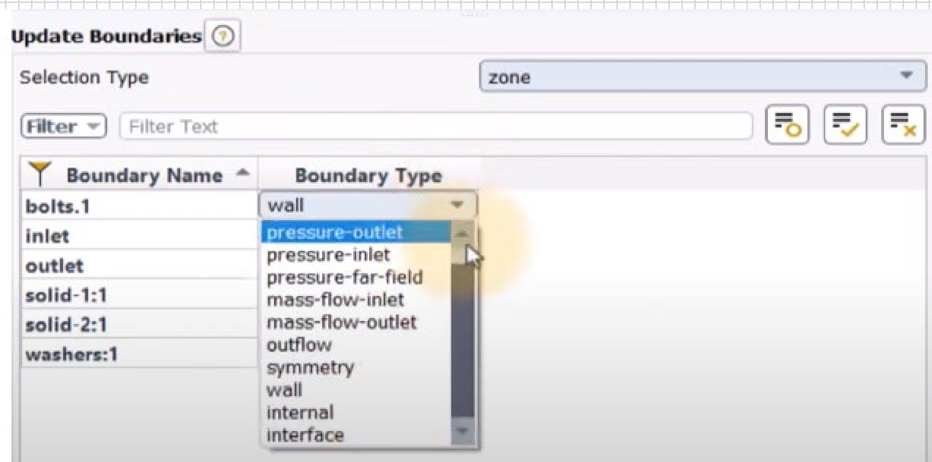
Geometry: Share Topology and Boundary

- **Join Intersect:** overlapping areas of surfaces identifier, separated from their parent surface and then merged into single surface
- **Interface Connect:** used when matching overlapped surfaces (similar shape and dimensions) between bodies
- Update Boundary: unavailable when the geometry type is "Only Solid"
- **Boundary names and types:**



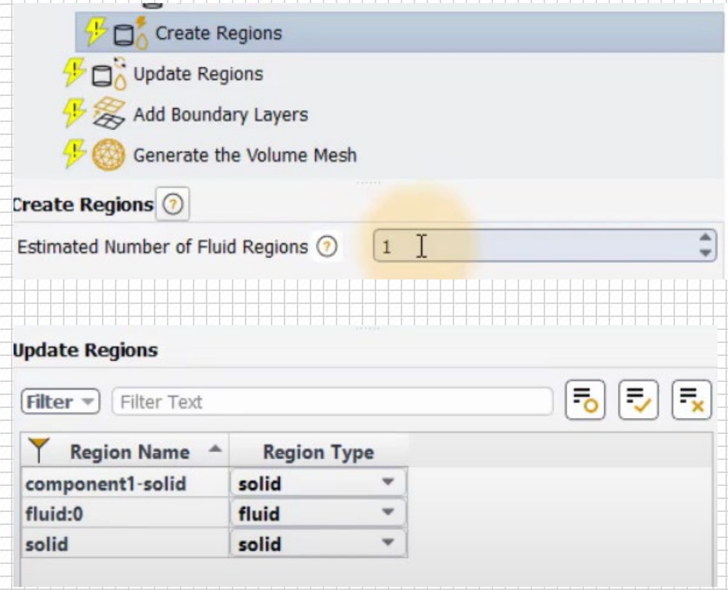


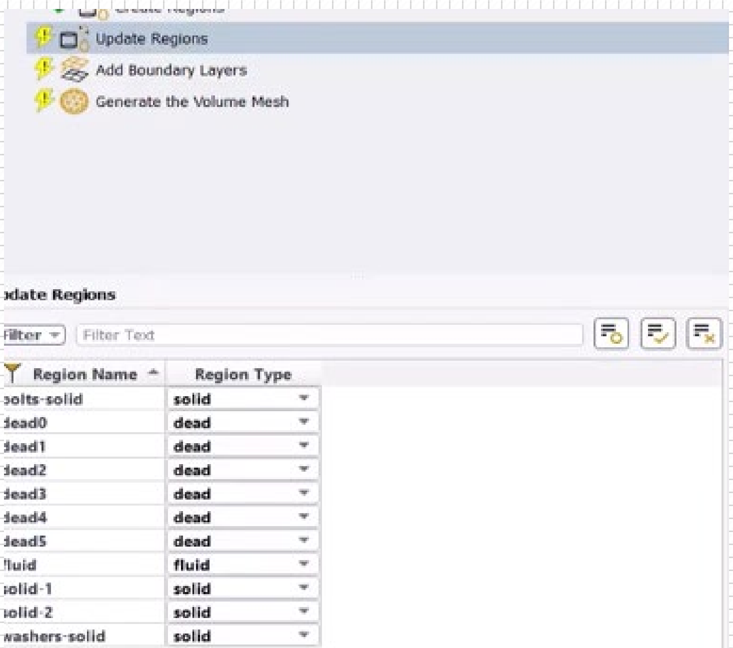
Inlet	Velocity inlet zone
Outlet	Pressure outlet zone
Symmetry	Symmetry
Interior	Internal zones
Farfield	Pressure far fields zones
Wall	Wall zones



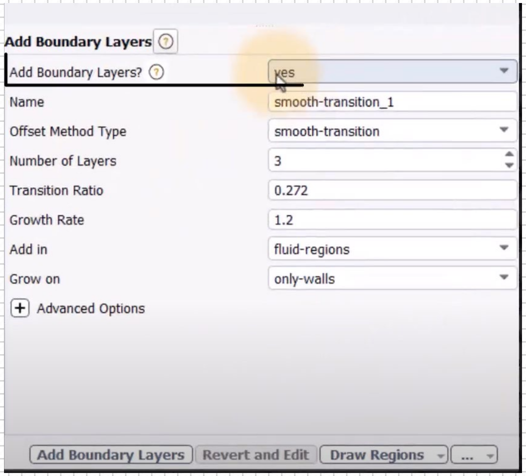
Update Regions

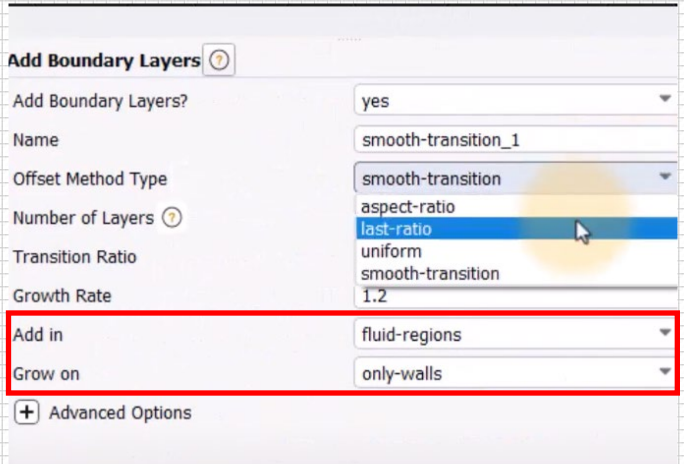
- **Create region:** solid and fluid
- **Update regions:** name and type
- **Dead regions:** void or pocket in the domain that are not transferred to solver

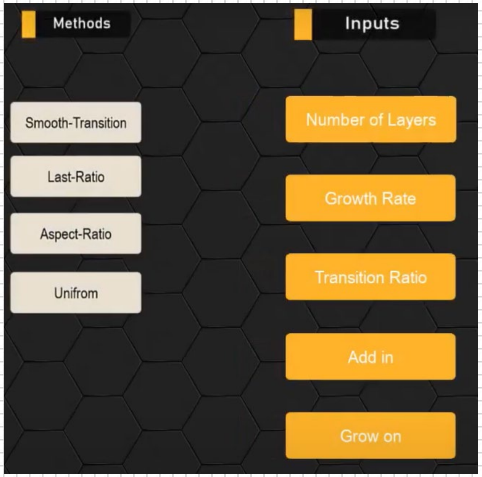


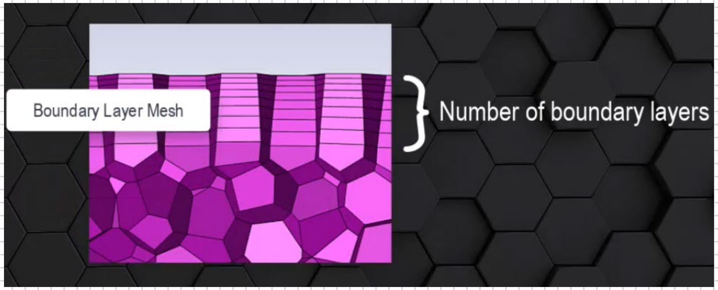


Boundary Layers







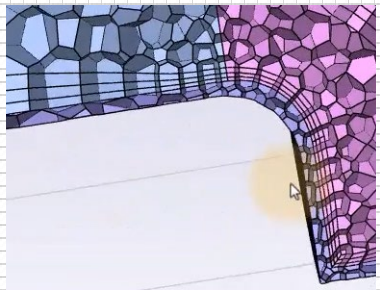


- **Transition Ratio:** ratio of the height of mesh in the last boundary layer and the first cell in volume fill
- **Growth Rate:** ratio of the thickness of the next boundary layer cell to the previous boundary layer cell

Boundary Layers

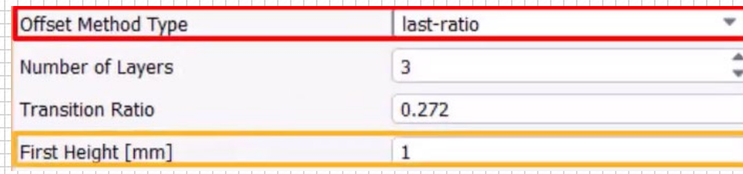
Smooth-Transition

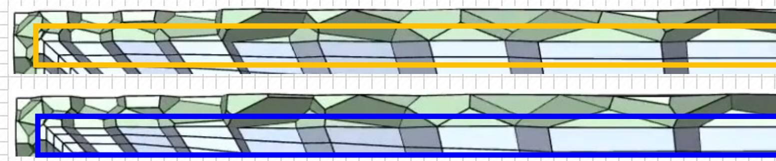
Boundary layer mesh thickness and first layer height vary along the surface (depended on the local surface mesh size)



First layer height or the total height of the boundary layer mesh cannot be controlled

Last-Ratio



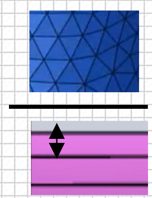


Growth layers is automatically varying to satisfy transition ratio and first height

Results in varying boundary layer thickness

Aspect-Ratio

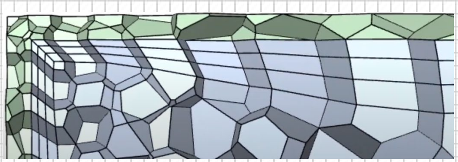
$$\text{Aspect Ratio} = \frac{\text{Local Surface Mesh Size}}{\text{Height of First Boundary Layer Cell}}$$



Lack of control on-

First layer height

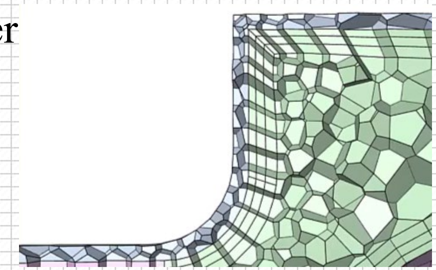
Total height of boundary layer mesh



Uniform

Thickness of each layer remains constant

Flat or lightly curves surfaces

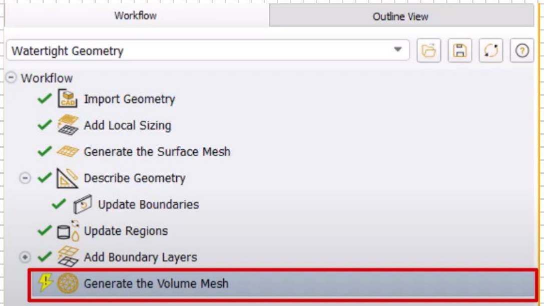


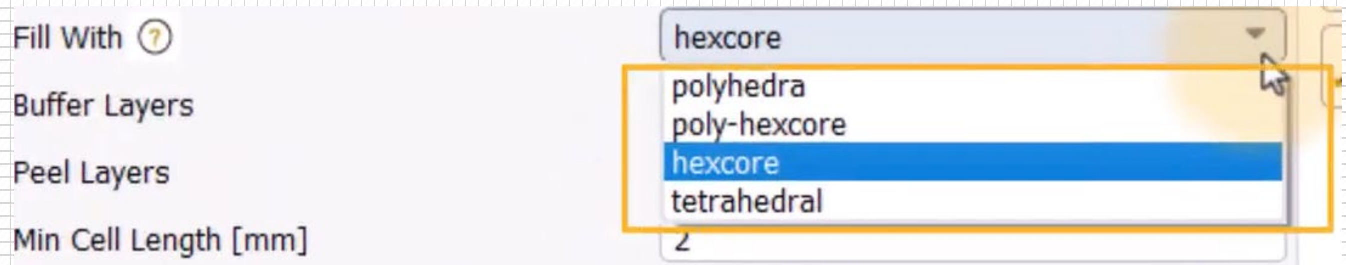
At sharp corners, this method can lead to formation of crevices

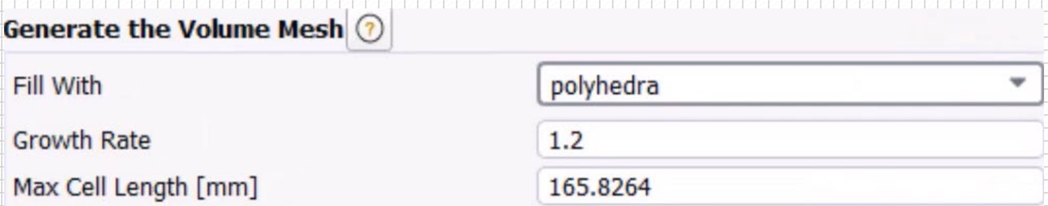
Transition between the boundary layer mesh and volume mesh cannot be controlled

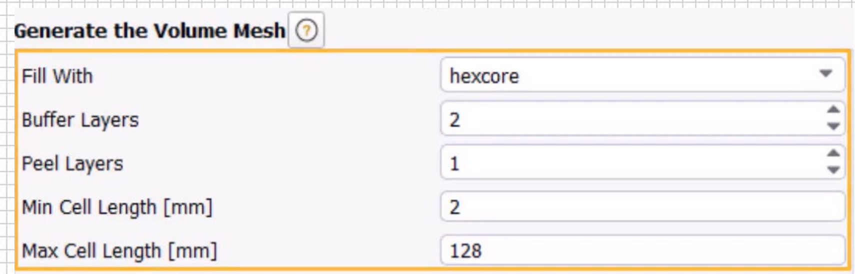
Volume Mesh

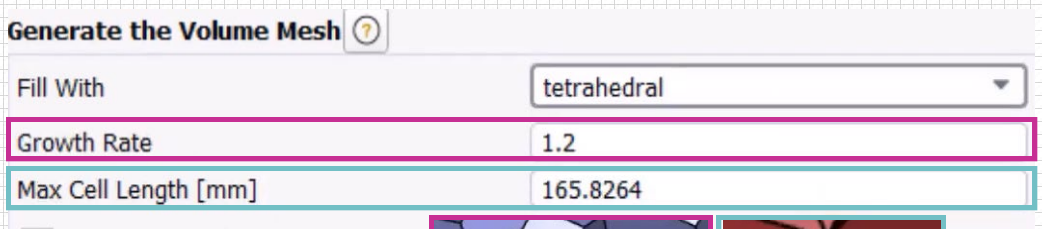
Accuracy , Convergence, Simulation Time

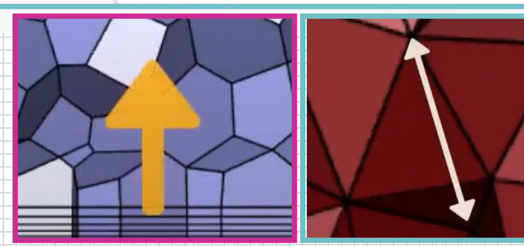




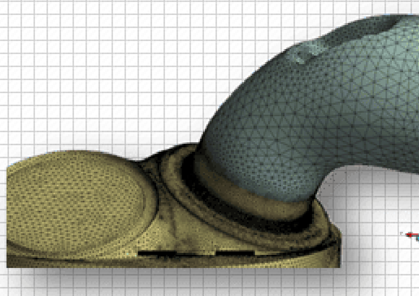
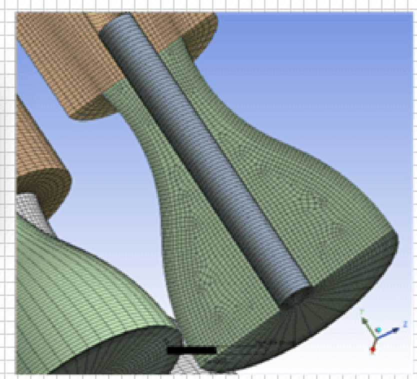
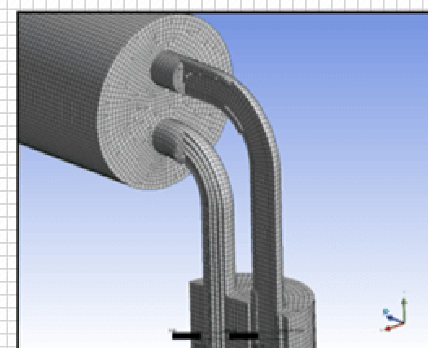






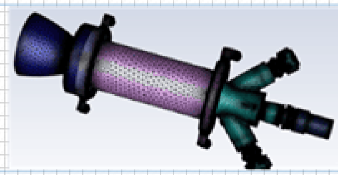
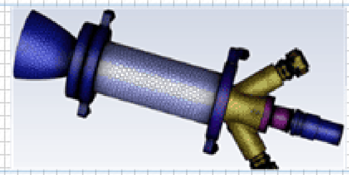
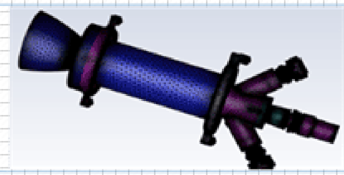
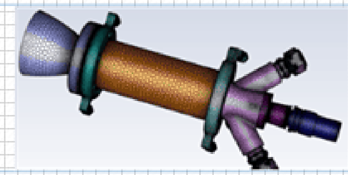
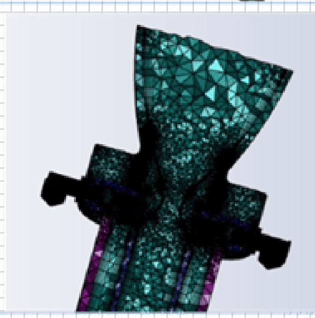
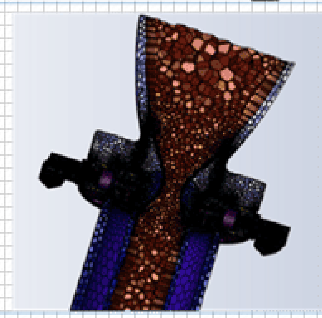
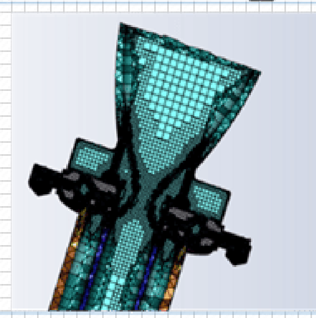
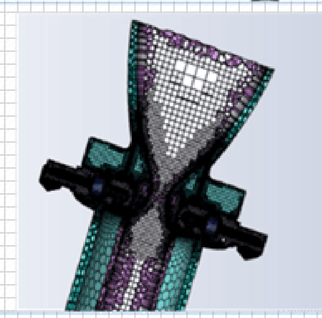


Volume Mesh

Method	Tetrahedral	Sweep	Multi Zone
Element Types	Tetrahedral + tri-prisms	Mostly Hex, some Tri Prisms	Mostly Hex, some Tri Prisms and Tets in transitions
Handle Complex Geometry?	Easiest	Difficult – requires decomposing into simple shapes	Medium- automatically decomposes geometry into blocks and fills with Hex core or Tets
Cell Count	Largest	Lowest (typically)	Low- Medium
Compatible with CFX/ Polyflow/ Fluent	Yes	Yes	Yes
Parallel Generation?	Yes	No	Yes
Mesh Overview			

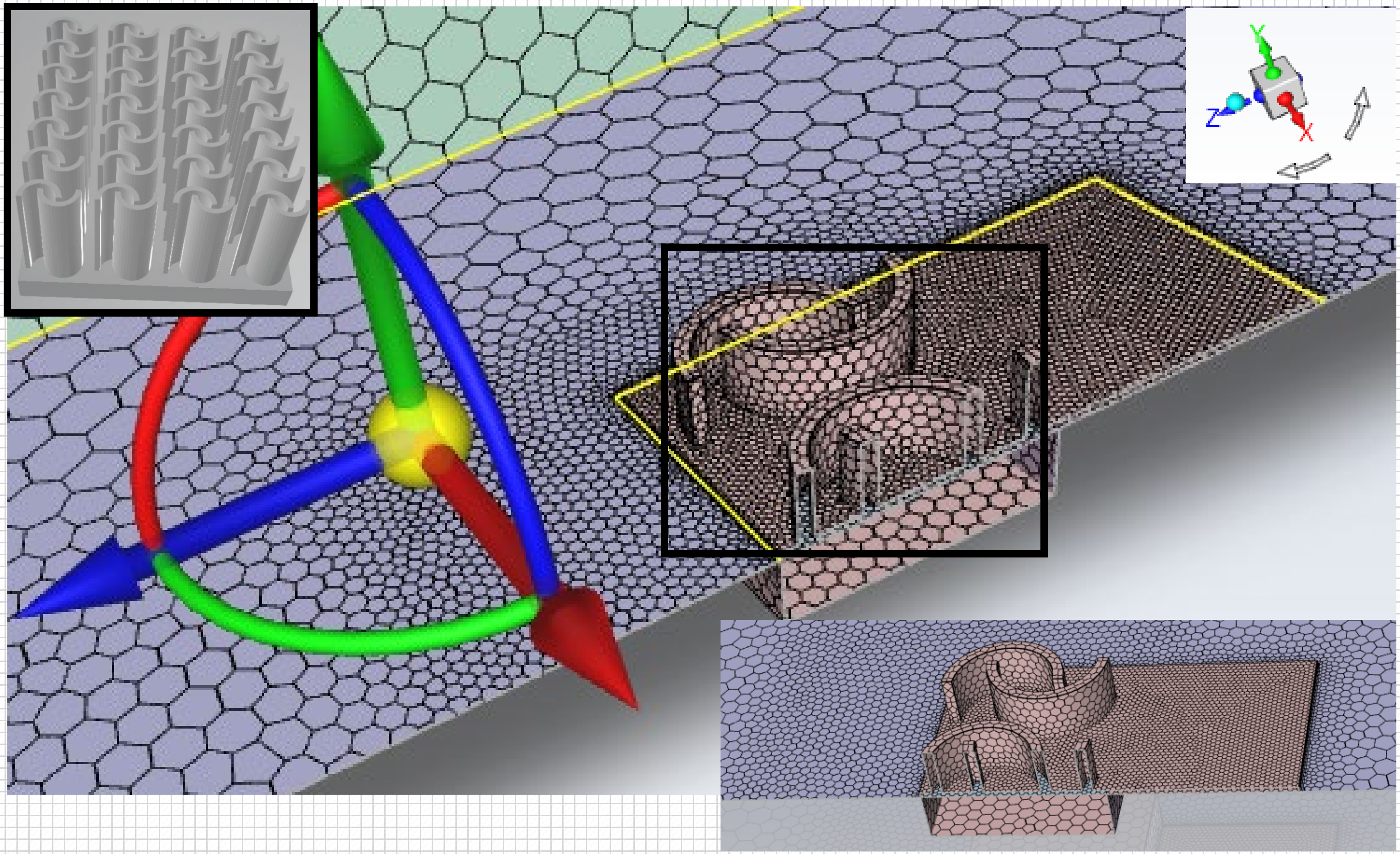
https://www.padtinc.com/2021/05/11/comparison-ansys-fluent-meshing-ansys-meshing-cfd/

Volume Mesh

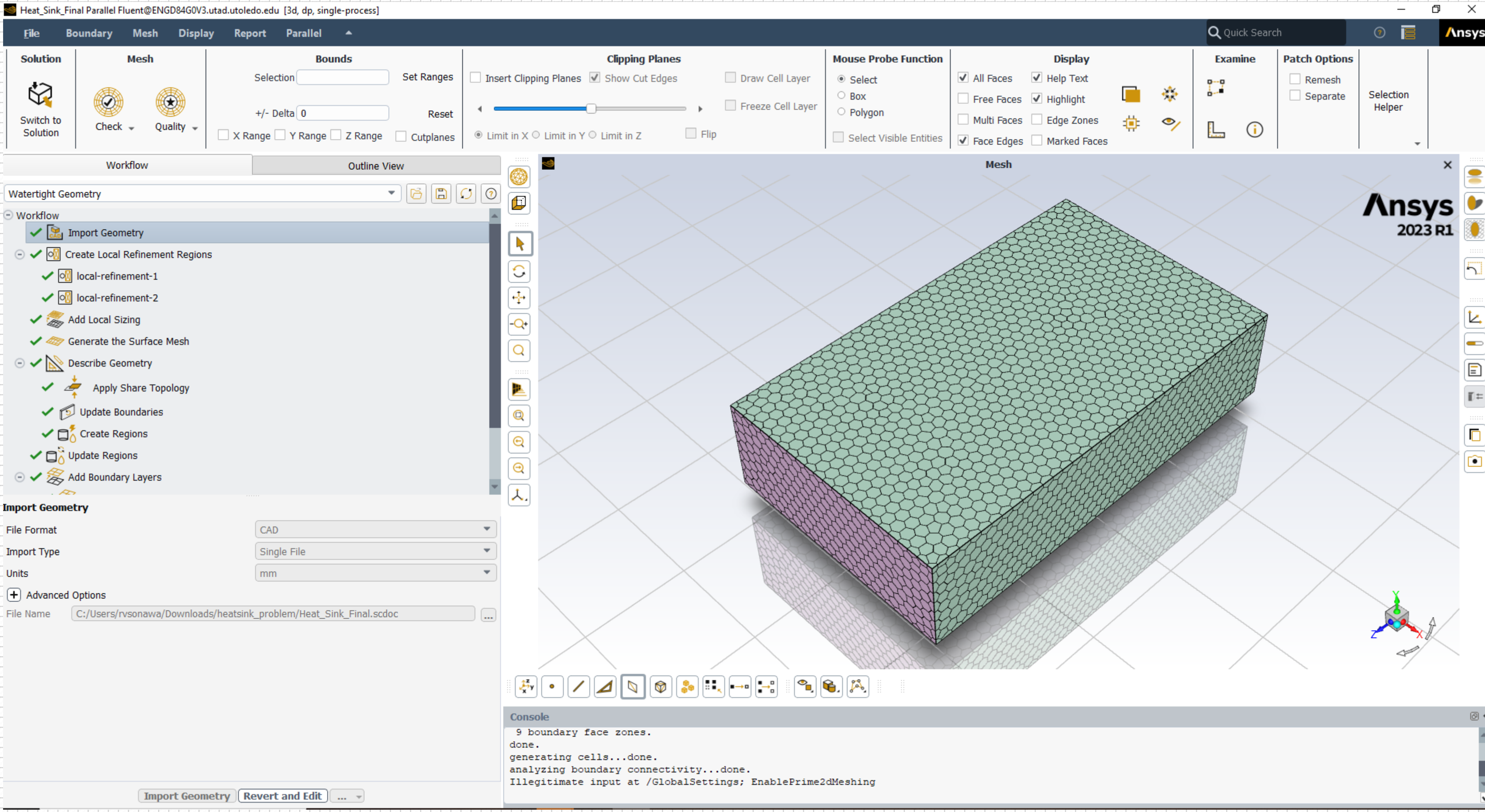
Method	Tetrahedral	Polyhedral	Hex-Core	Poly Hex-Core
Element Types	Tetrahedral + tri-prisms	Polyhedral + poly-prisms	Hexahedral + tri-prisms + Tet/Pyramid transition	Polyhedral + poly-prisms + Polyhedral transition
Handle Complex Geometry?	Easiest	Easy	Medium	Medium
Cell Count	Largest	Low	Medium	Lowest
Compatible with CFX/ Polyflow	Yes	No	Yes, [setting to avoid 1/8 octree transition = yes]	No
Parallel Generation?	Tets No, Prisms Yes	Yes	Yes	Yes
Surface Mesh				
Volume Mesh				

https://www.padtinc.com/2021/05/11/comparison-ansys-fluent-meshing-ansys-meshing-cfd/

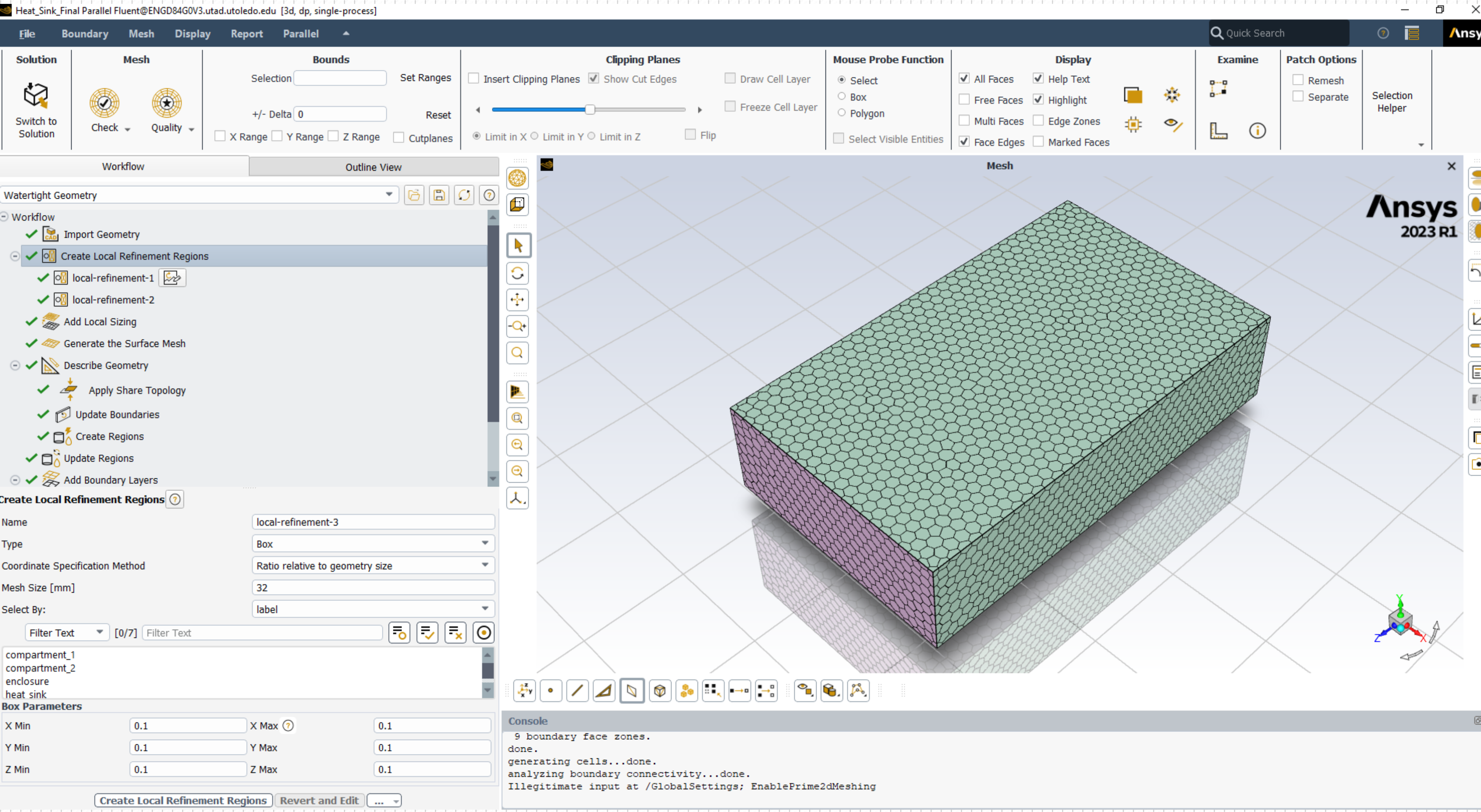
Heat Sink Mesh: Sample Geometry



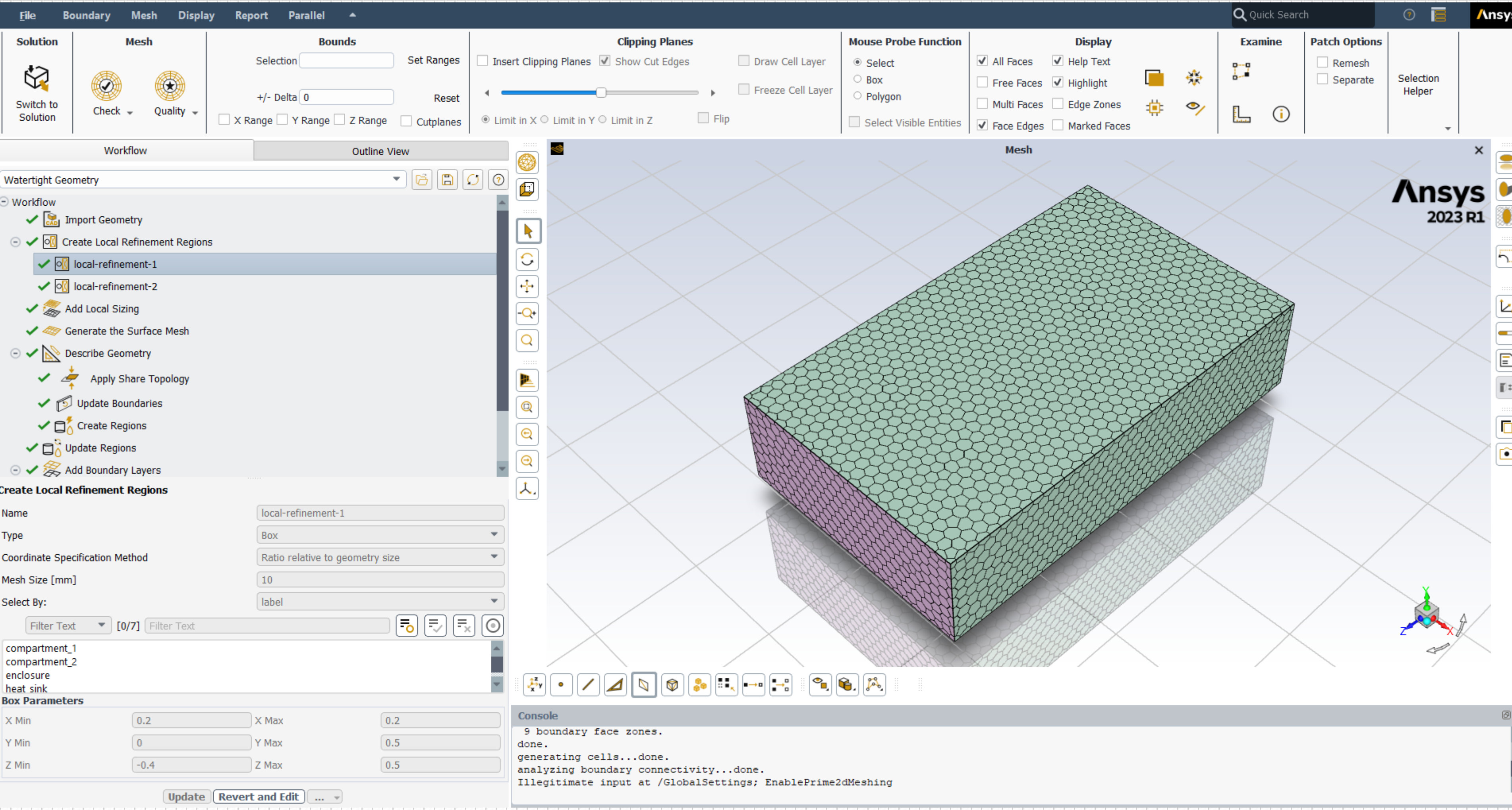
Heat Sink Mesh: Import Geometry



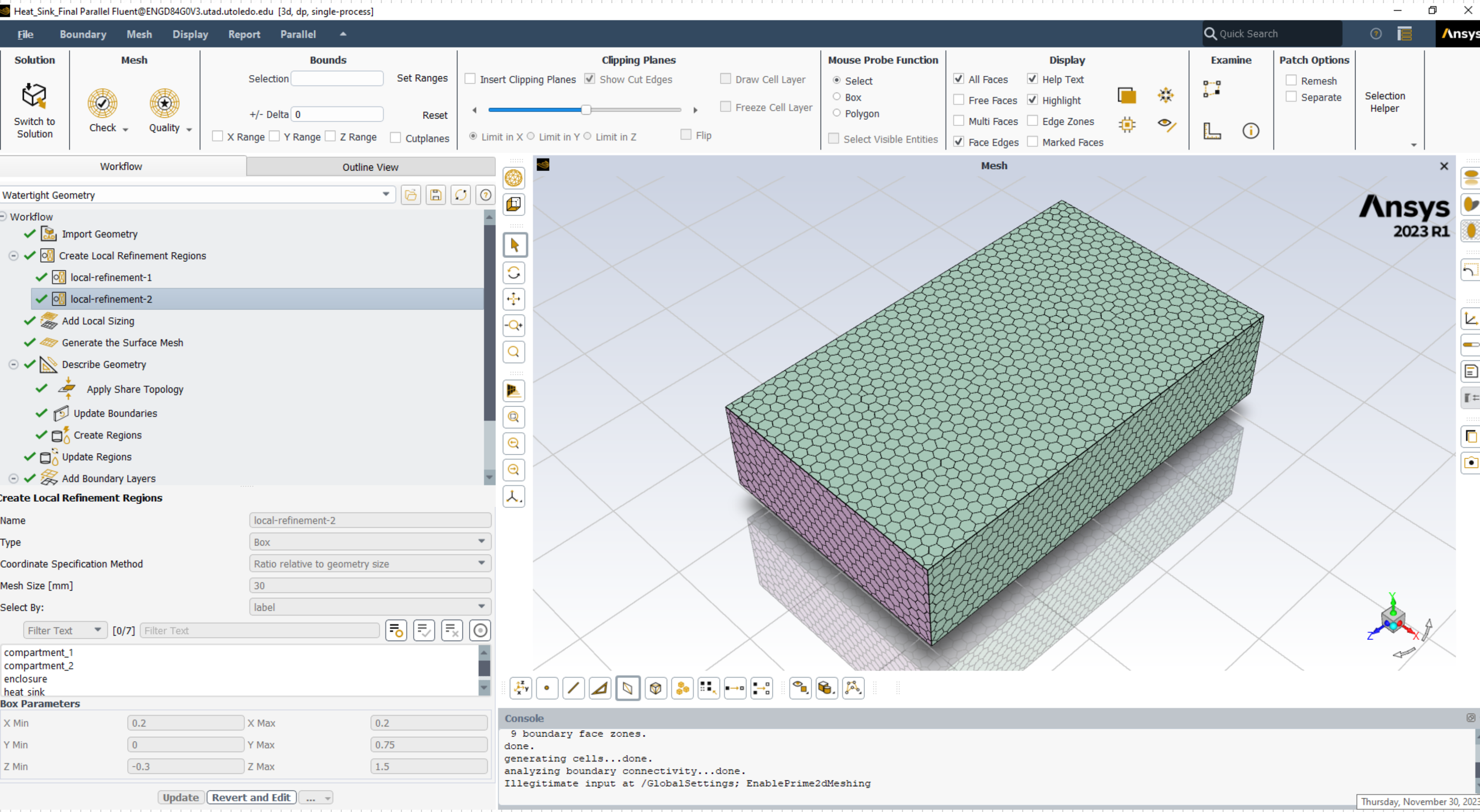
Heat Sink Mesh: Local Refinements



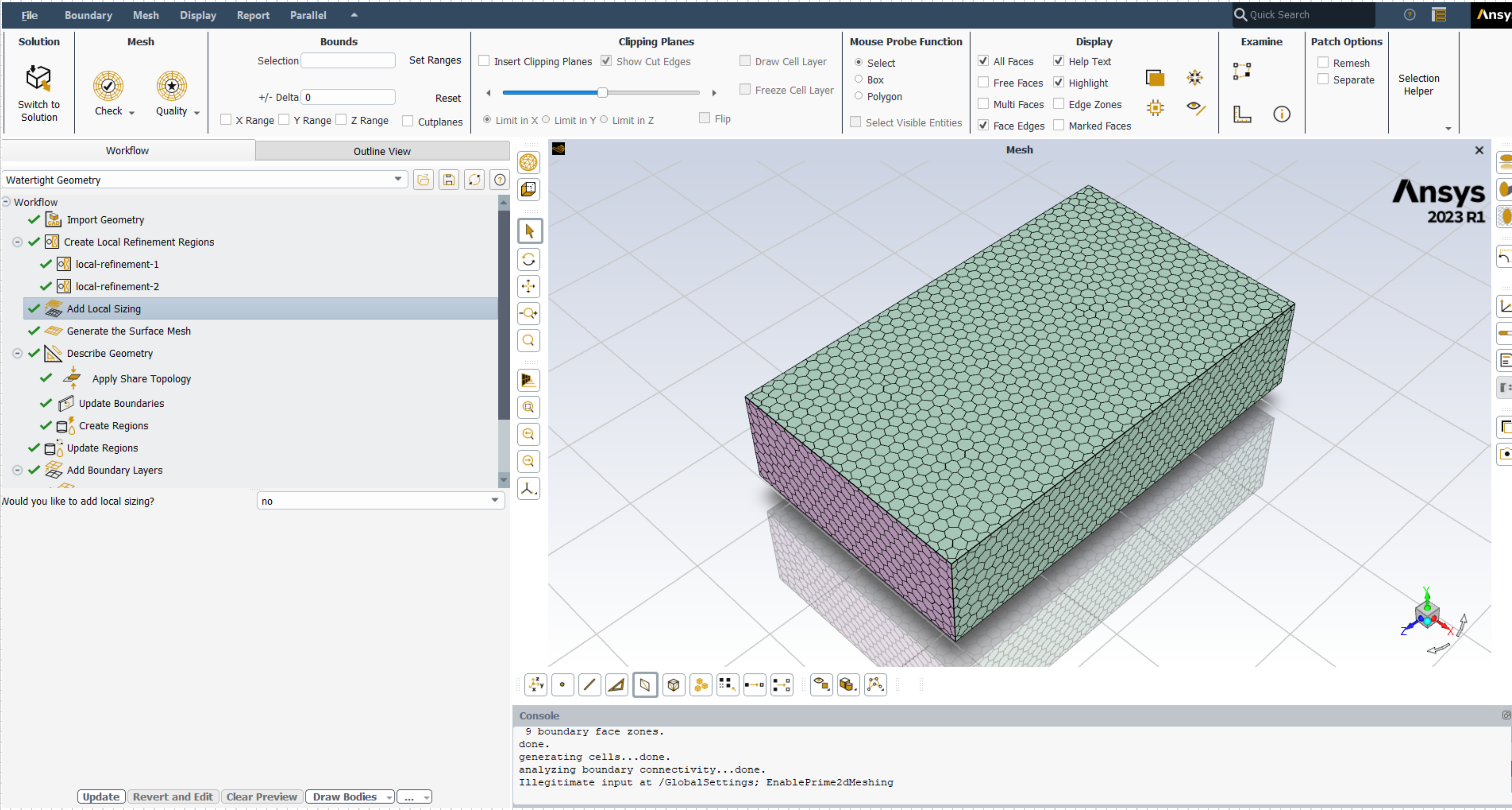
Heat Sink Mesh: Local Refinements



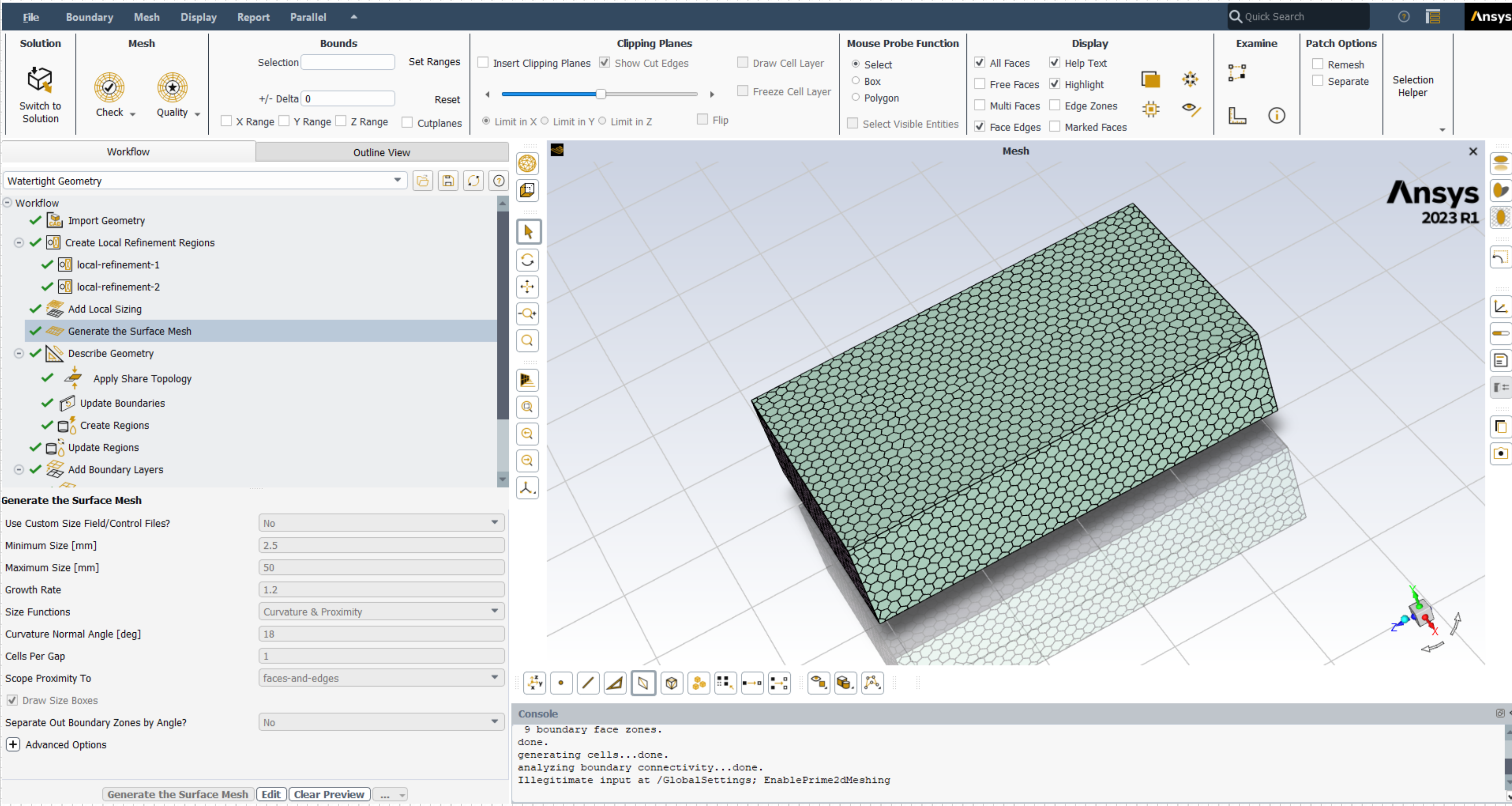
Heat Sink Mesh: Local Refinements



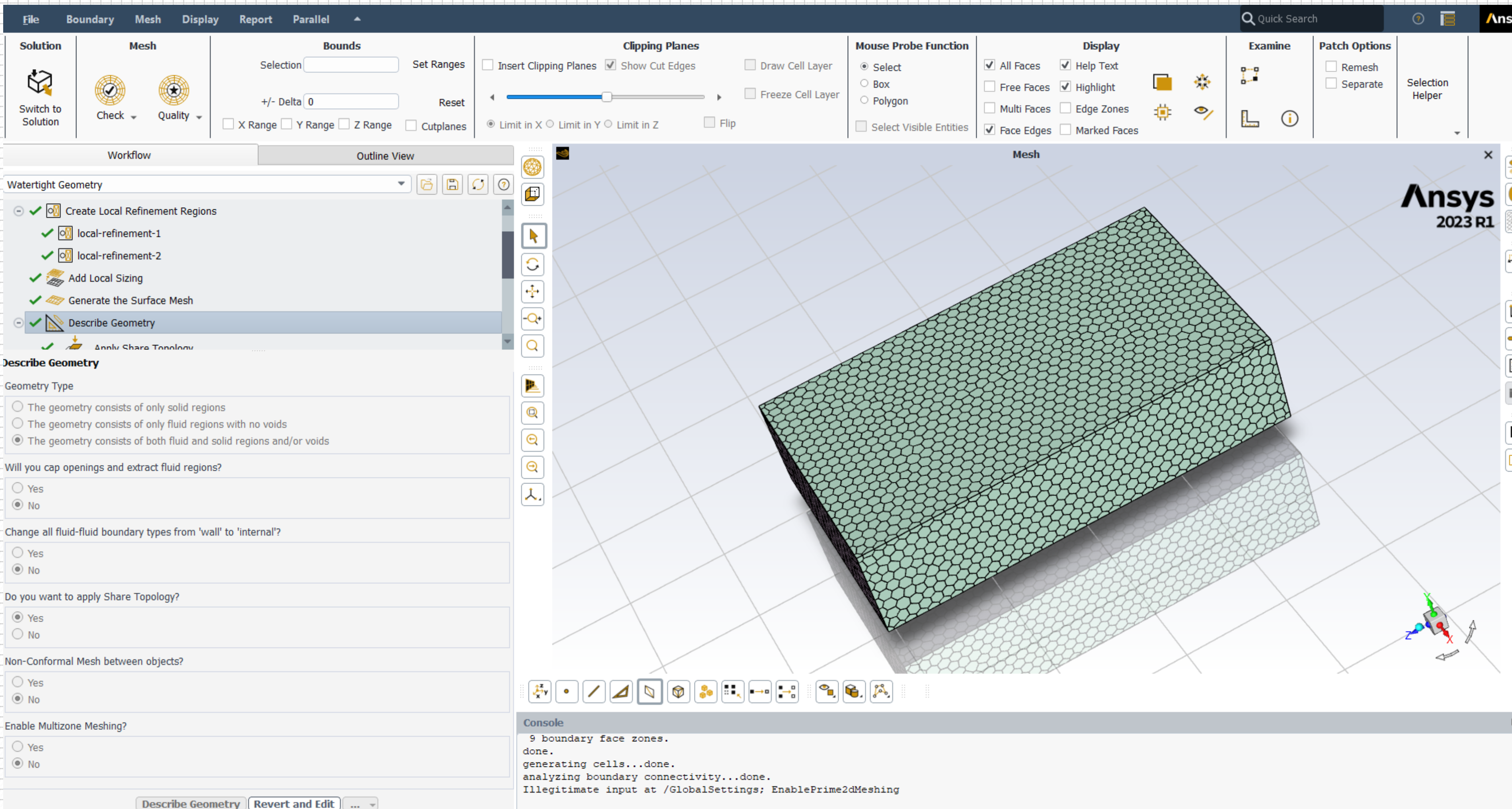
Heat Sink Mesh: Local Sizing



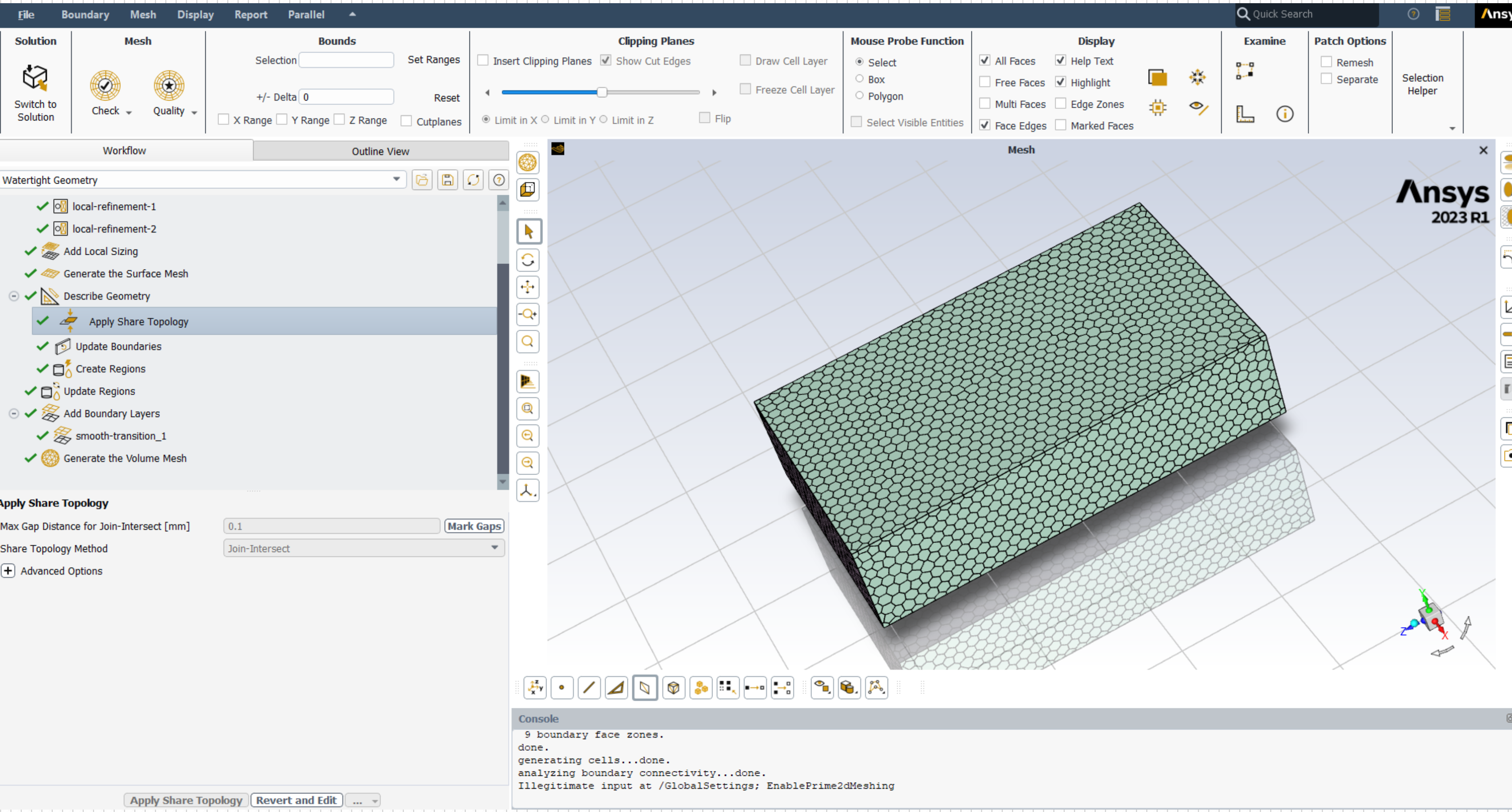
Heat Sink Mesh: Surface Mesh



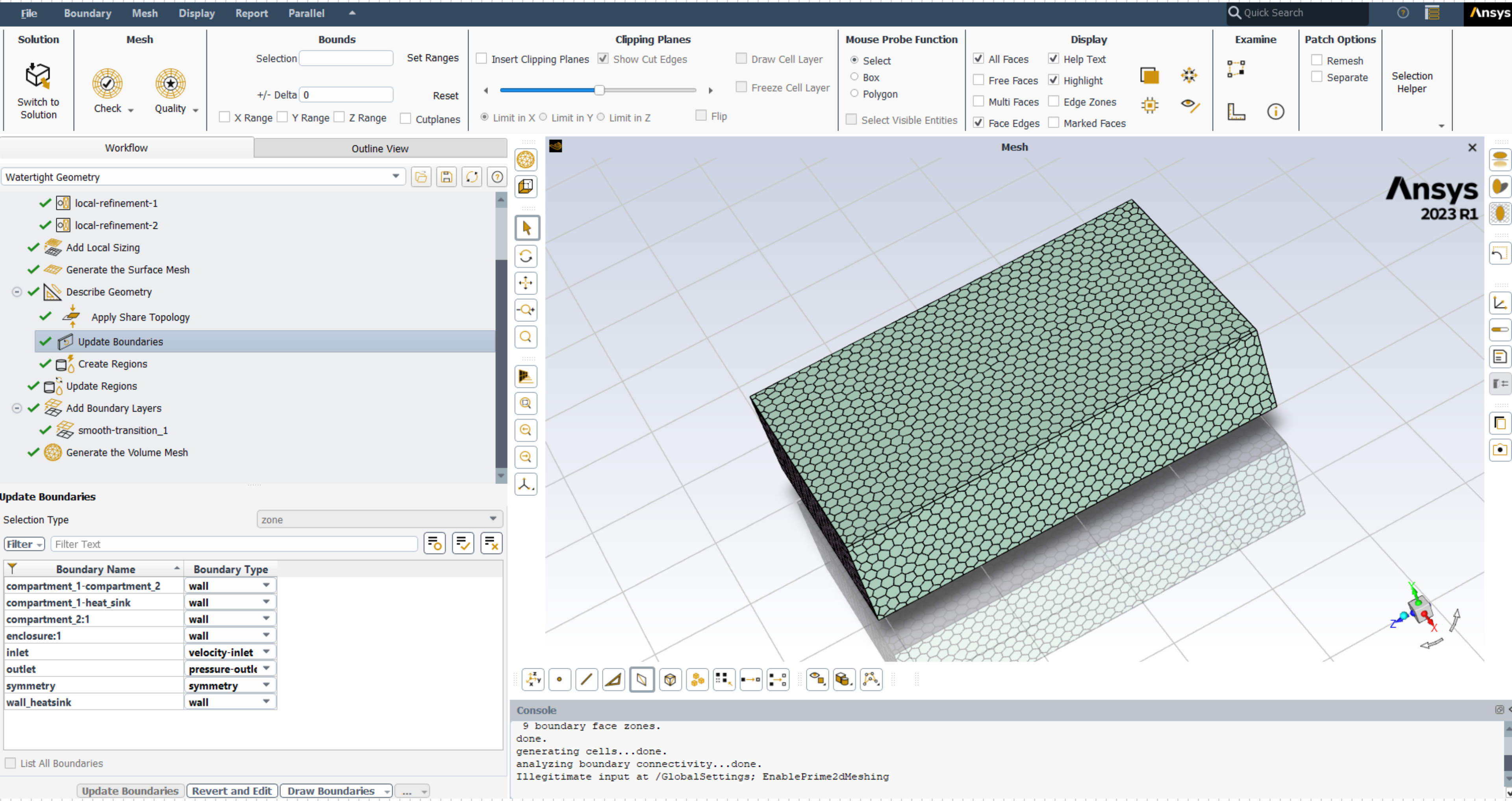
Heat Sink Mesh: Describe Geometry



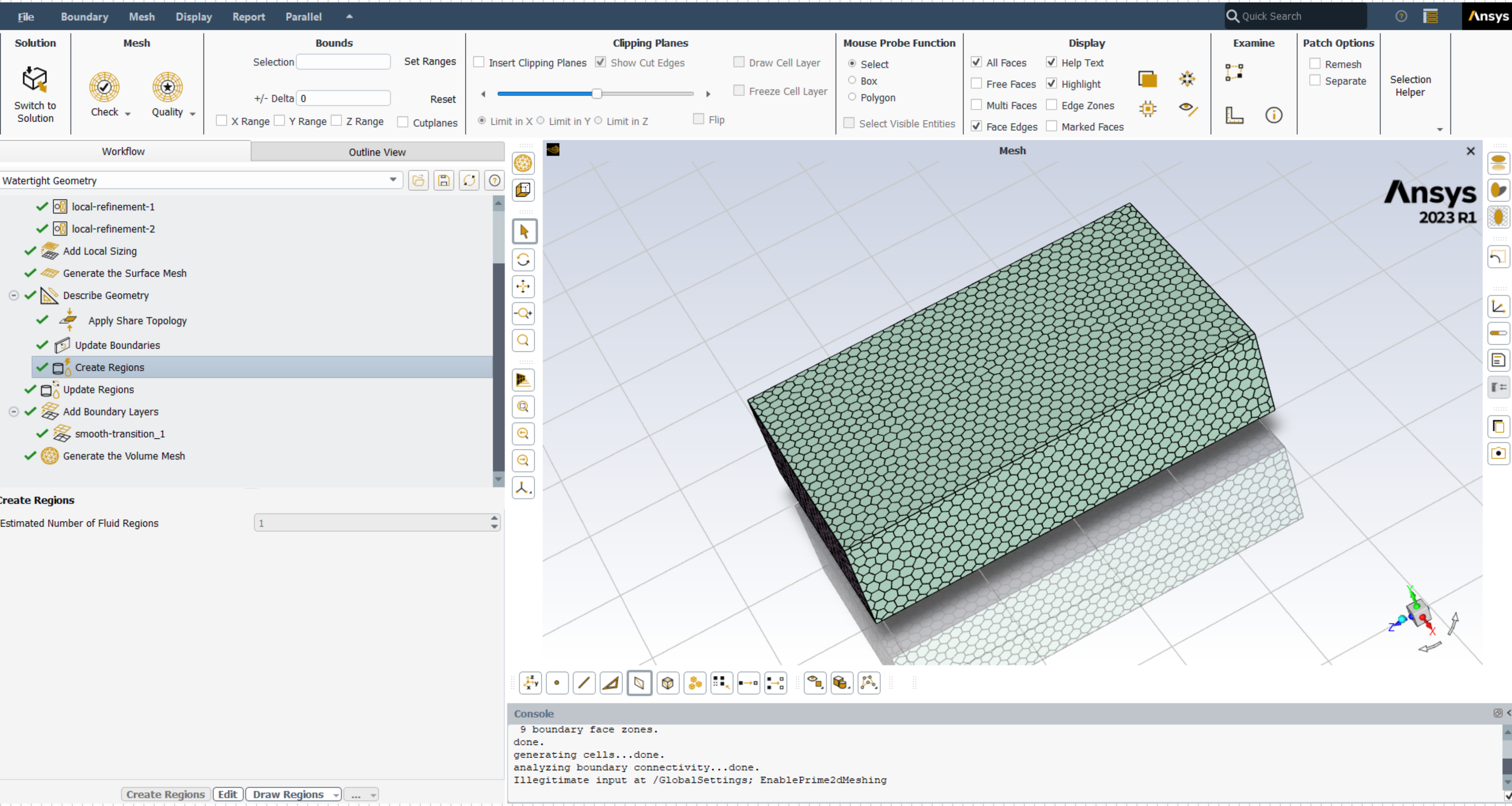
Heat Sink Mesh: Apply Share Topology



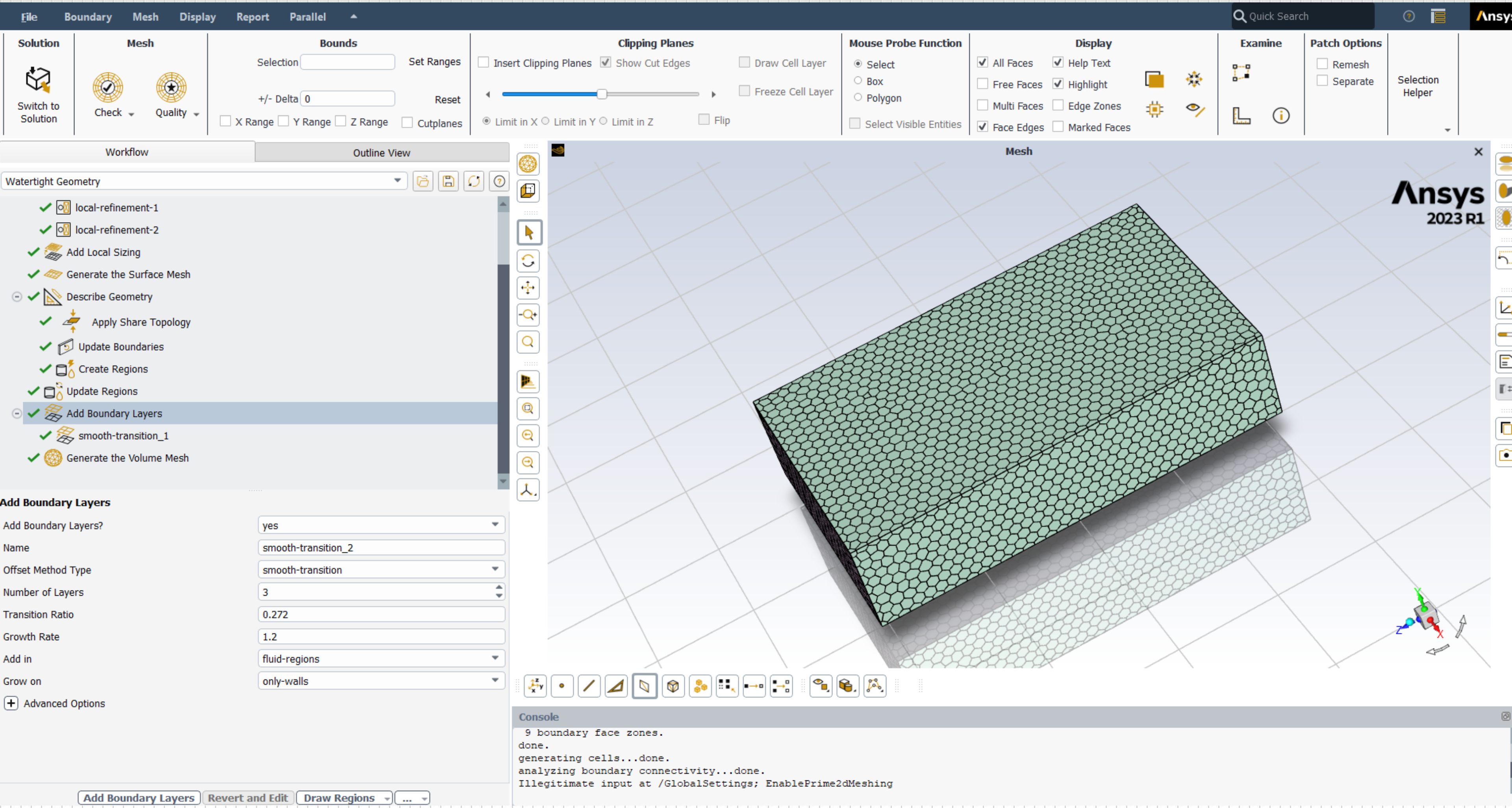
Heat Sink Mesh: Update Boundaries



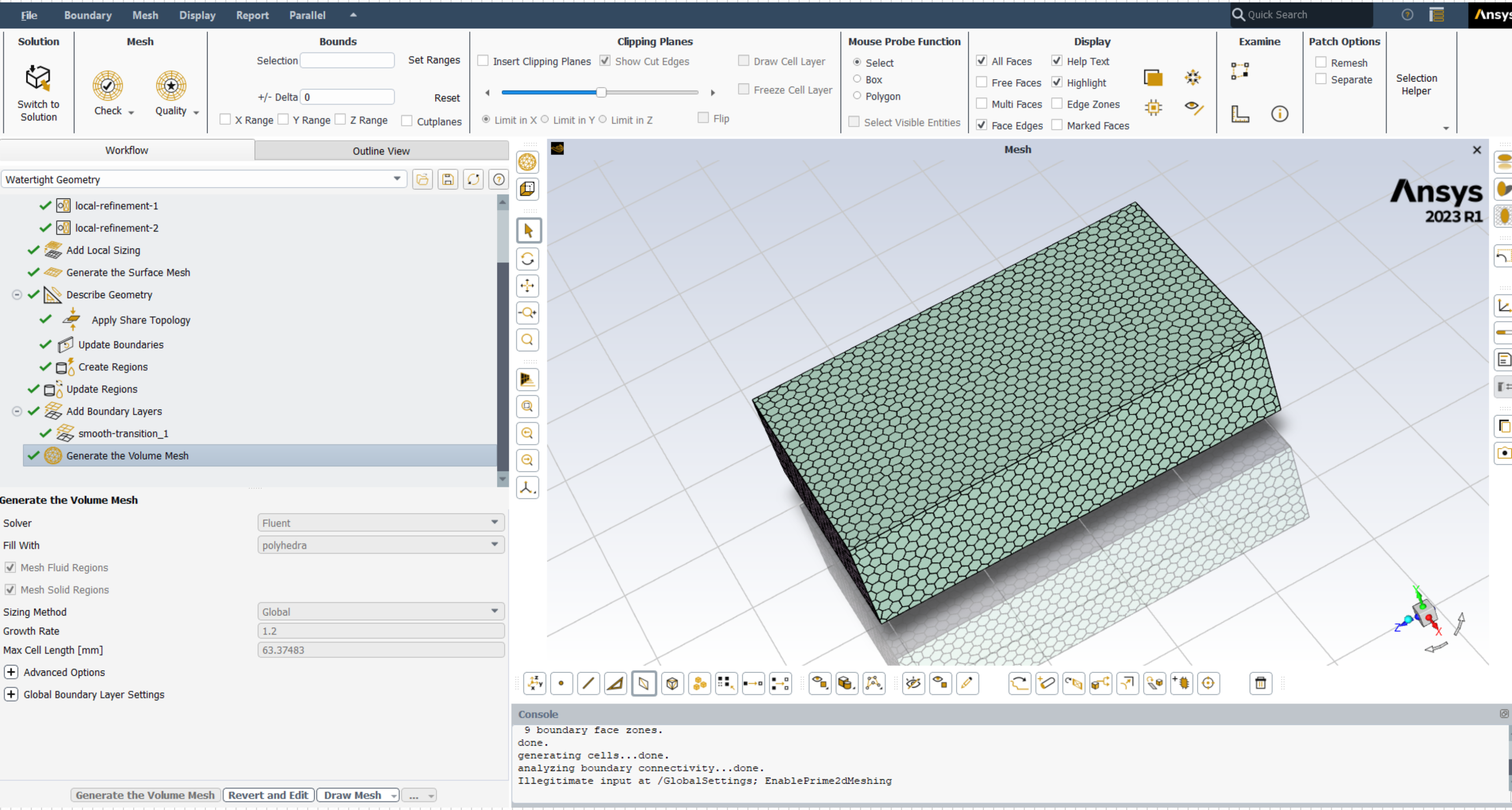
Heat Sink Mesh: Create Regions



Heat Sink Mesh: Add Boundary Layers



Heat Sink Mesh: Generate the Volume Mesh



Mesh Quality

Factors

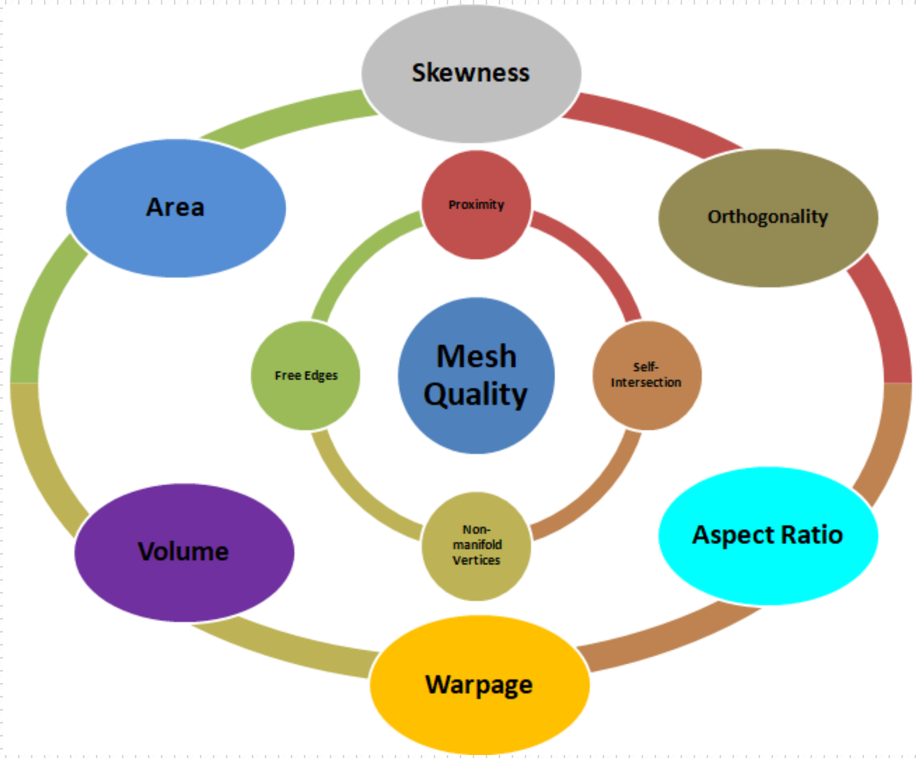
Geometry issues
Meshing parameters
Meshing methods

Skewness mesh metrics spectrum

Excellent	Very good	Good	Acceptable	Bad	Unacceptable
0-0.25	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Orthogonal Quality mesh metrics spectrum

Unacceptable	Bad	Acceptable	Good	Very good	Excellent
0-0.001	0.001-0.14	0.15-0.20	0.20-0.69	0.70-0.95	0.95-1.00



https://www.cfdyna.com/CFDHT/Meshing.html

Mesh Parameters and Quality

Mesh parameters

```
3D mesh
nodes: 671942
edges: 9288
faces: 844300
cells: 141661
reading 17 node zones
reading 25 edge zones
reading 12 face zones
reading 3 cell zones
generating pointers...done.
extracting boundary entities...
 85843 boundary nodes.
 24942 boundary faces.
 9 boundary face zones.
done.
generating cells...done.
analyzing boundary connectivity...done.
Illegitimate input at /GlobalSettings; EnablePrime2dMeshing
```

Mesh check

```
Domain extents.
  x-coordinate: min = -5.400000e+02, max = 5.400000e+02.
  y-coordinate: min = -5.000000e+01, max = 4.150000e+02.
  z-coordinate: min = -5.400000e+02, max = 1.260000e+03.
Volume statistics.
  minimum volume: 4.187806e-01.
  maximum volume: 1.402583e+05.
    total volume: 8.100000e+08.
Face area statistics.
   minimum face area: 4.241841e-02.
   maximum face area: 2.920631e+03.
   average face area: 8.916646e+01.
Checking number of nodes per edge.
Checking number of nodes per face.
Checking number of nodes per cell.
Checking number of faces/neighbors per cell.
Checking cell faces/neighbors.
Checking isolated cells.
Checking face handedness.
Checking periodic face pairs.
Checking face children.
Checking face zone boundary conditions.
Checking for invalid node coordinates.
Checking poly cells.
Checking zones.
Checking neighborhood.
Checking interfaces.
Checking modified centroid.
Checking non-positive or too small area.
Done.
----------------------------------------
```

Volume mesh quality

```
Cell Quality [Measure : Orthogonal]
Minimum = 0.19093208, Maximum =    1, Average = 0.92670673.

Marking Cells in Range (0.19093208 0.24093208)...
--------------- ----------
 Cluster Number Entity Count
--------------- ----------
              1          3
              2          1
              3          2
              4          1
              5          2
              6          2
              7          2
              8          1
              9          1
```

Surface mesh quality

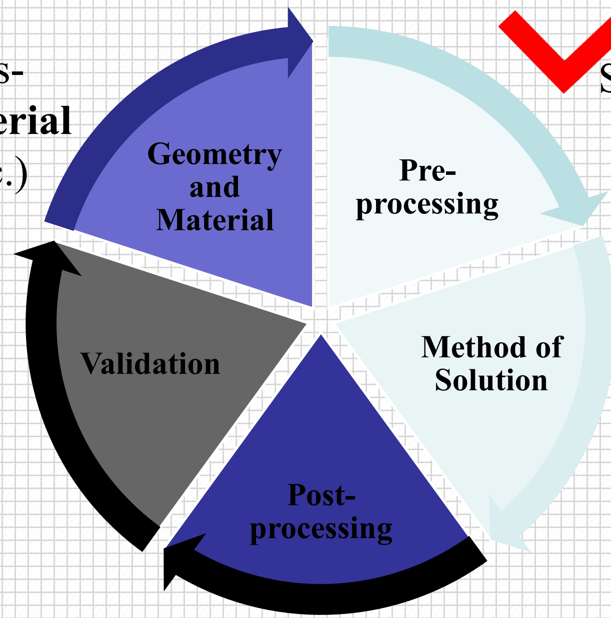
```
Surface Quality [Measure : Skewness]
Minimum = 2.220446e-16, Maximum = 0.58181627, Average = 0.018292677.

Marking Face in Range (0.53181627 0.58181627)...
--------------- ----------
 Cluster Number Entity Count
--------------- ----------
              1          1
              2          1
              3          1
              4          1
```

Simulations/Digital Twinning

- Simulation is the process of virtually emulating a physical process
- It is used to verify the efficacy of a design or product

Geometry (e.g. shape, cross-sections, assemblies) and **material** (e.g. steel, rubber, wood, etc.)

Setting up the model through **mesh generation** and **boundary conditions**



Sanity check (i.e. do the results match the expectation, are the results plausible?) E.g. Does the output indicate a need in change of geometry or material?

Numerical methods used to solve the model

Evaluation of outputs (e.g. displacement, stresses and strains)

Day 2 Fluent Workshop: Heat Sink Design

https://github.com/anjurgupta/Fluent-Heat-Sink-Workshop

Ansys
FLUENT

A.R. Gupta
MIME 3440: Heat Transfer

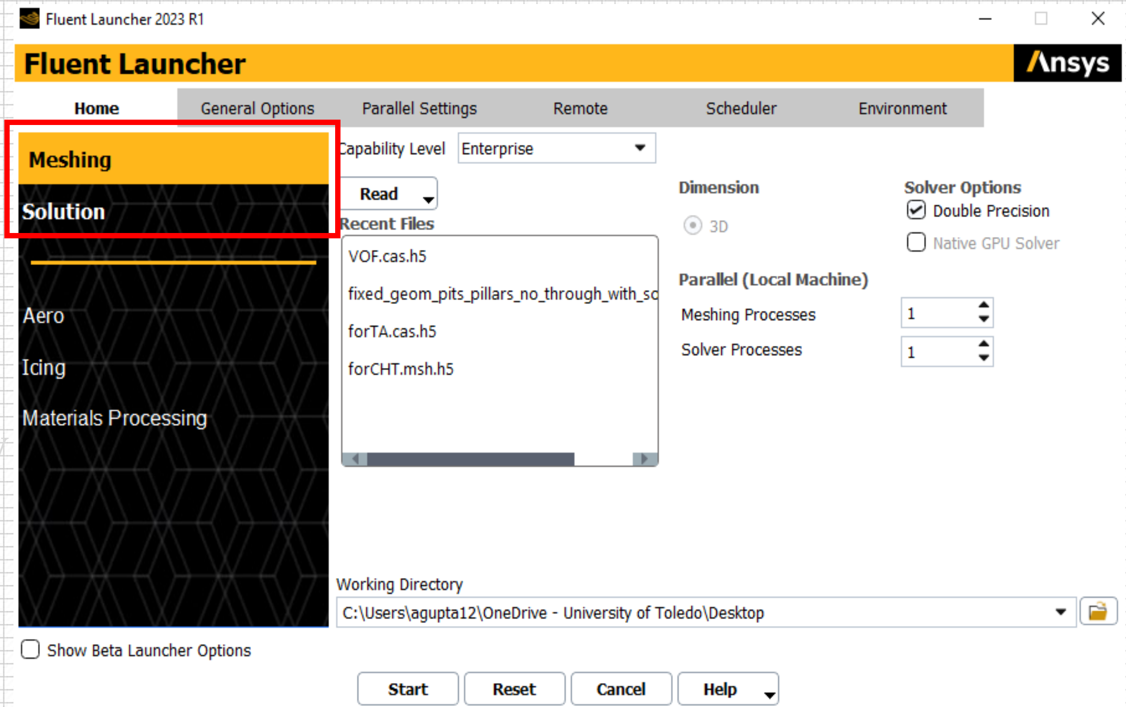
Topics

(Day 1) Heat Sink

- Modes of Heat Transfer
- Geometries
- Electronics cooling- Class problem 3.8

(Day 1) Meshing

- Geometry import and workflow
- Local sizing, surface mesh, boundary layers
- Volume mesh and quality

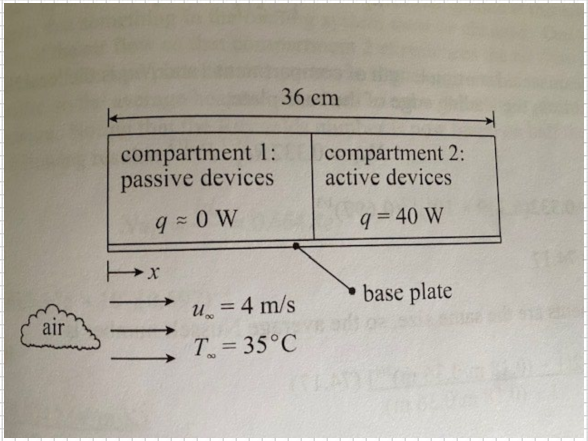


(Day 2) **Solution**

- Mesh import and workflow
- Fluent Set up: solid and fluid zones for electronics cooling- Class problem 3.8
- Check: Convergence and Visualization, Material property (Copper vs Aluminum)

Electronics Cooling using Heat Sink (Example 3.8)

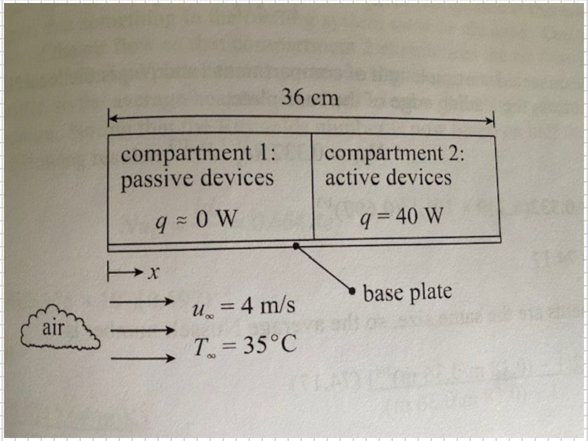
Electronic component in compartment 2 is generating 40 W of thermal power. This CFD model simulates the dissipation of this thermal energy into a stream of cold air blowing over a heat sink



- **Materials:** fluid (air) , solid (copper)
- **Material property**:
  Copper: $\rho = 8928$ kg/m^3, k = 387.6 W/m.K
  Air: $\rho = 1.225$ kg/m^3, k = 0.0242 W/m.K
  $\mu = 1.7894 \times 10^{-5}$ kg/m.s
- **Physical models:** viscous and turbulent, Steady State using pressure-based coupled
- **Boundary conditions:** air inlet velocity = 4 m/s, T = 308 K,
  outlet (pressure outlet): open to atmosphere (Gauge pressure = 0 Pa)
  heating surface: q = 40 W, fluid walls (wall): adiabatic

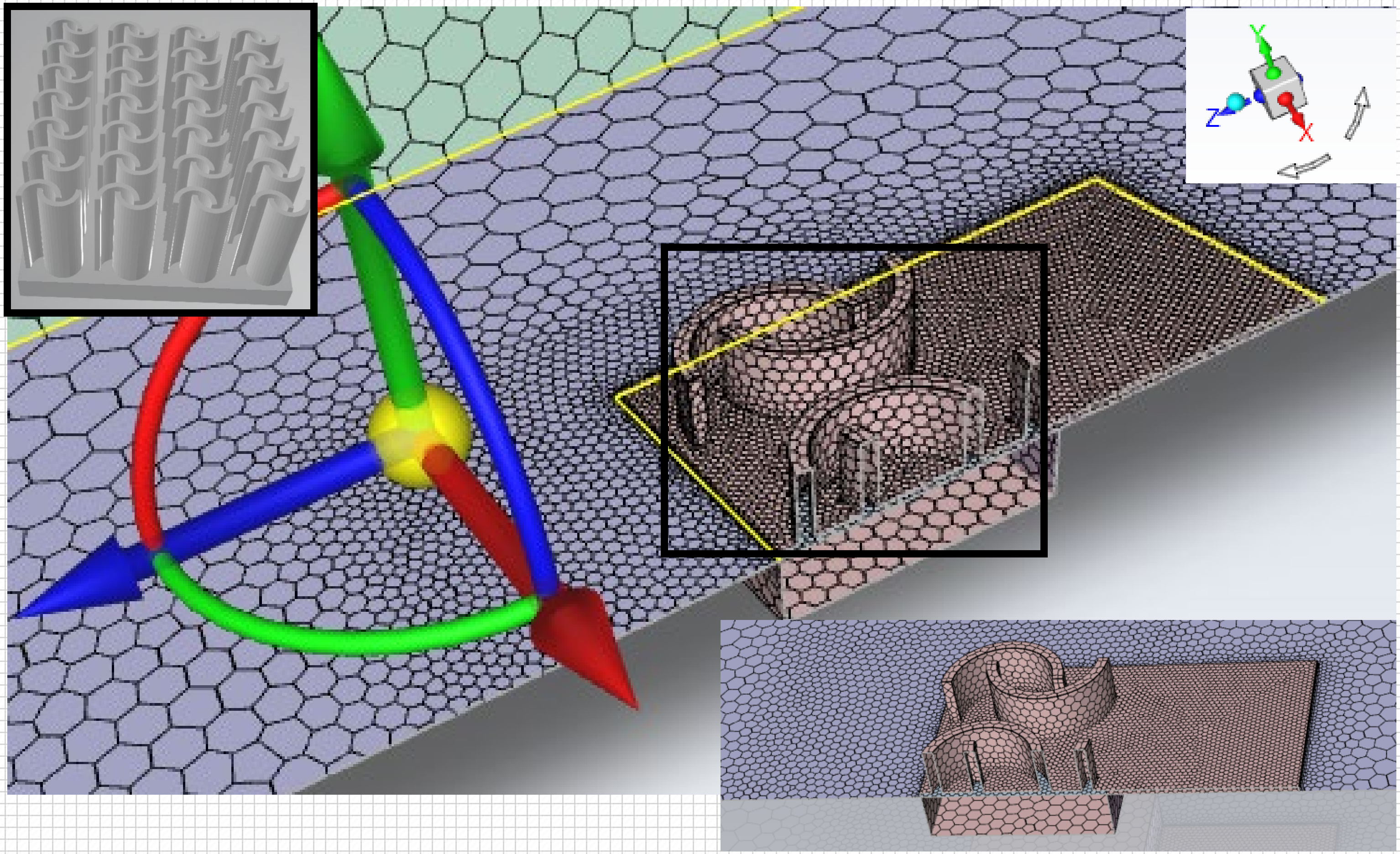
Electronics Cooling using Heat Sink (Example 3.8)

Electronic component in compartment 2 is generating 40 W of thermal power. This CFD model simulates the dissipation of this thermal energy into a stream of cold air blowing over a heat sink

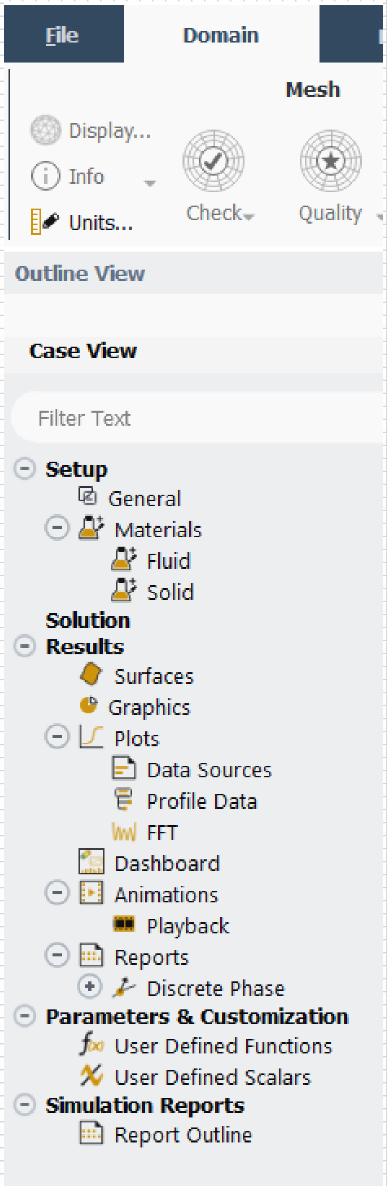


- **Materials:** fluid (air) , solid (aluminum)
- **Material property**:
  Aluminum: : ρ = 2719 kg/m^3, k = 202.4 W/m.K
  Air: ρ = 1.225 kg/m^3, k = 0.0242 W/m.K
  μ = 1.7894 x 10^{-5} kg/m.s
- **Physical models:** viscous and turbulent, Steady State using pressure-based coupled
- **Boundary conditions:** air inlet velocity = 4 m/s, T = 308 K,
  outlet (pressure outlet): open to atmosphere (Gauge pressure = 0 Pa)
  heating surface: q = 40 W, fluid walls (wall): adiabatic

Heat Sink Mesh: Sample Geometry



Solution

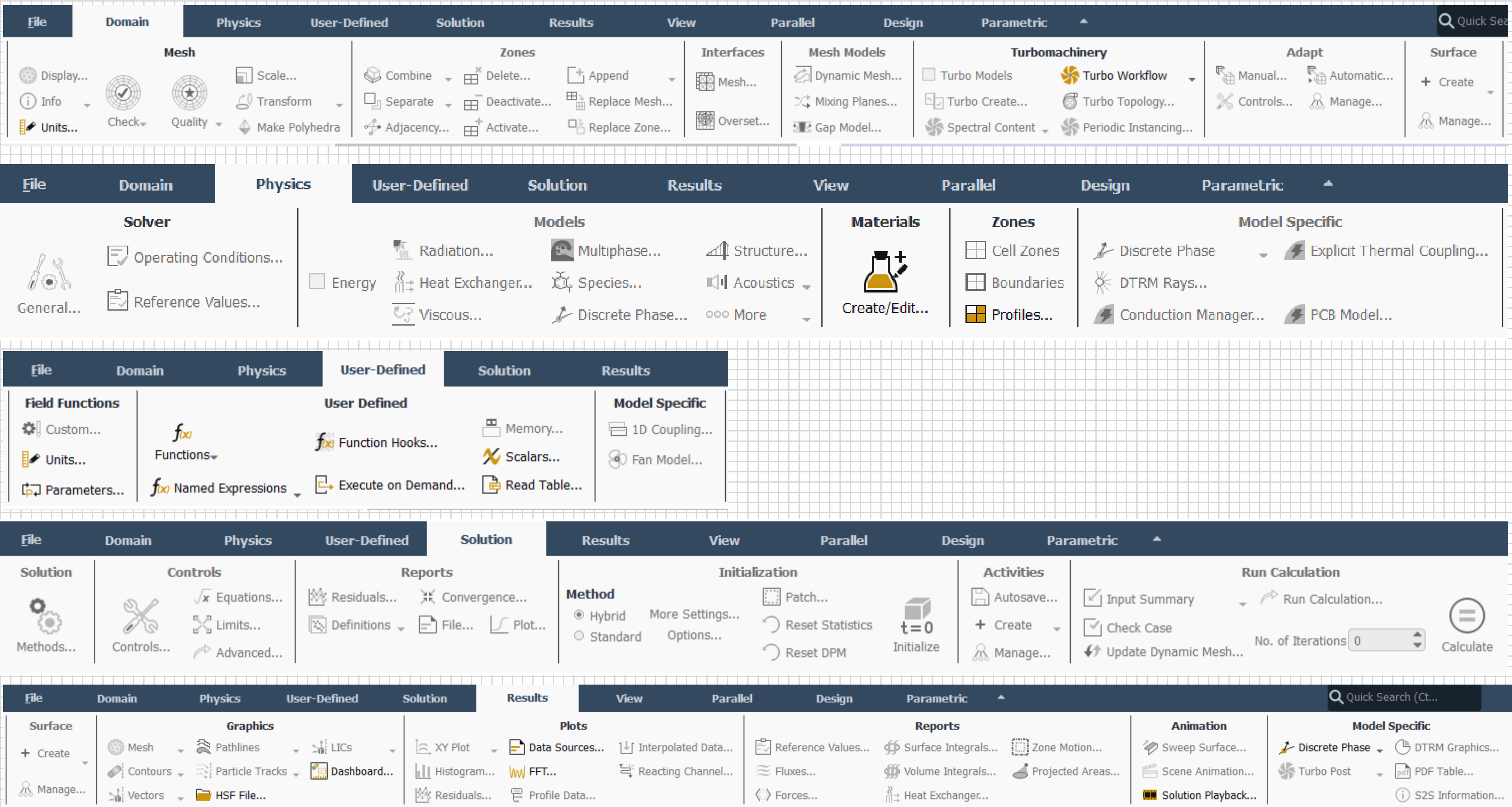


- Materials: fluid, solid, mixtures
- Physical models: turbulence, combustion, multiphase
- Operating conditions
- Boundary conditions
- Initial Values
- Solver Controls
- Convergence monitors
- Visualization: sheal layers, flow pattern, shocks
- Numerical reporting: forces, moments, surface and volume integrated quantities, flux balances

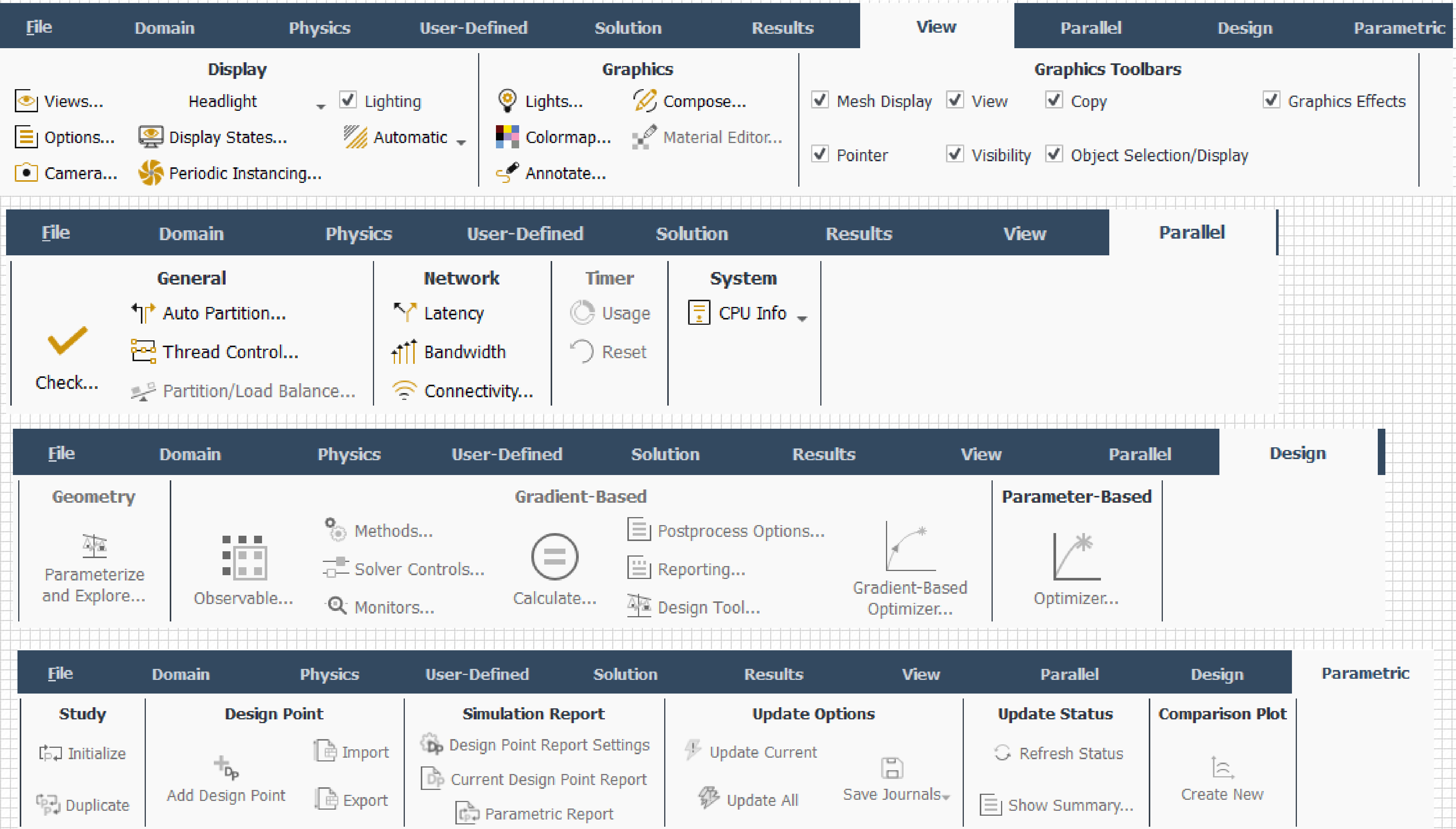
The electronics communications system consists of two adjacent compartments as shown. The compartments share a common base plate that functions as an air-cooled heat sink. The compartment that experiences the air flow first contains passive devices that dissipate a negligible amount of heat. The other compartment contains devices that contribute to a total heat dissipation of 40 W, all of which transfers into the heat sink. Atm air at 35°C is forced over the base plate at a velocity of 4 m/s. The compartments are the same size, and the overall size of the base plate is 36 cm x 36 cm. Find the average surface temperature of the base plate.

Properties: atm air: $k = 0.0312$ W/m.K, $\nu = 22.68 \times 10^{-6}$ m^2/s, $Pr = 0.697$

Solution: Workflow



Solution: Workflow

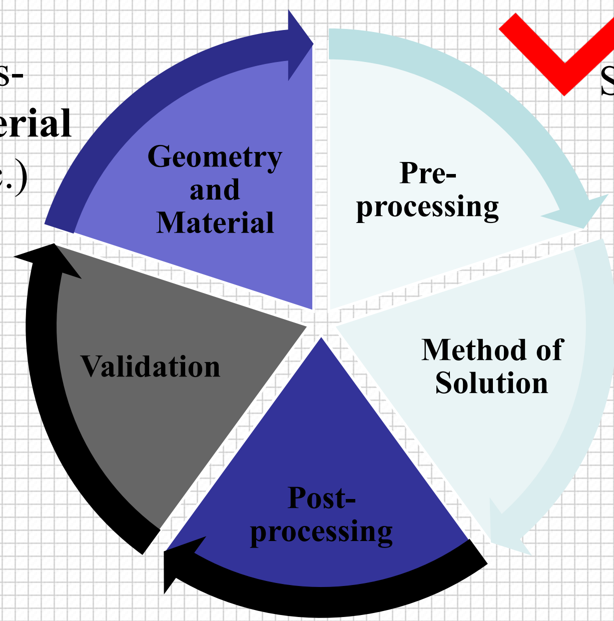


Simulations/Digital Twinning

- Simulation is the process of virtually emulating a physical process
- It is used to verify the efficacy of a design or product

Geometry (e.g. shape, cross-sections, assemblies) and **material** (e.g. steel, rubber, wood, etc.)

Setting up the model through **mesh generation** and **boundary conditions**



Numerical methods used to solve the model

Sanity check (i.e. do the results match the expectation, are the results plausible?) E.g. Does the output indicate a need in change of geometry or material?

Evaluation of outputs (e.g. displacement, stresses and strains)

Electronics Cooling using Heat Sink (Example 3.8)

Results:

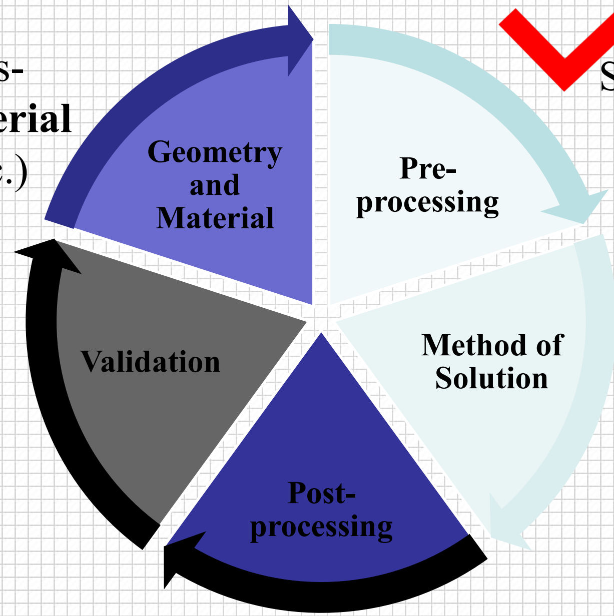
Configuration	**Avg Temp (°C)**	**Max Temp (°C)**	**Al Volume (cm^3)**	**% Temp Reduction (vs flat)**	**% Material Savings (vs rect)**
Flat plate					
Rectangular fins					
Triangular fins					

Simulations/Digital Twinning

- Simulation is the process of virtually emulating a physical process
- It is used to verify the efficacy of a design or product

Geometry (e.g. shape, cross-sections, assemblies) and **material** (e.g. steel, rubber, wood, etc.)

Setting up the model through **mesh generation** and **boundary conditions**



Sanity check (i.e. do the results match the expectation, are the results plausible?) E.g. Does the output indicate a need in change of geometry or material?

Numerical methods used to solve the model

Evaluation of outputs (e.g. displacement, stresses and strains)

Electronics Cooling using Heat Sink (Example 3.8)

DIY: Compare Copper vs Aluminum (solid)
Air vs Water (Fluid)

- **Materials:** fluid (air) , solid (copper)

- **Material property**:
  Copper: ρ = 8928 kg/m^3, k = 387.6 W/m.K
  Air: ρ = 1.225 kg/m^3, k = 0.0242 W/m.K
  μ = 1.7894 x 10^{-5} kg/m.s

- **Materials:** fluid (air) , solid (aluminum)

- **Material property**:
  Aluminum: : ρ = 2719 kg/m^3, k = 202.4 W/m.K
  Air: ρ = 1.225 kg/m^3, k = 0.0242 W/m.K
  μ = 1.7894 x 10^{-5} kg/m.s