

# Day 1 Fluent Workshop: Heat Sink Design

---



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



# Topics

## (Day 1) Heat Sink

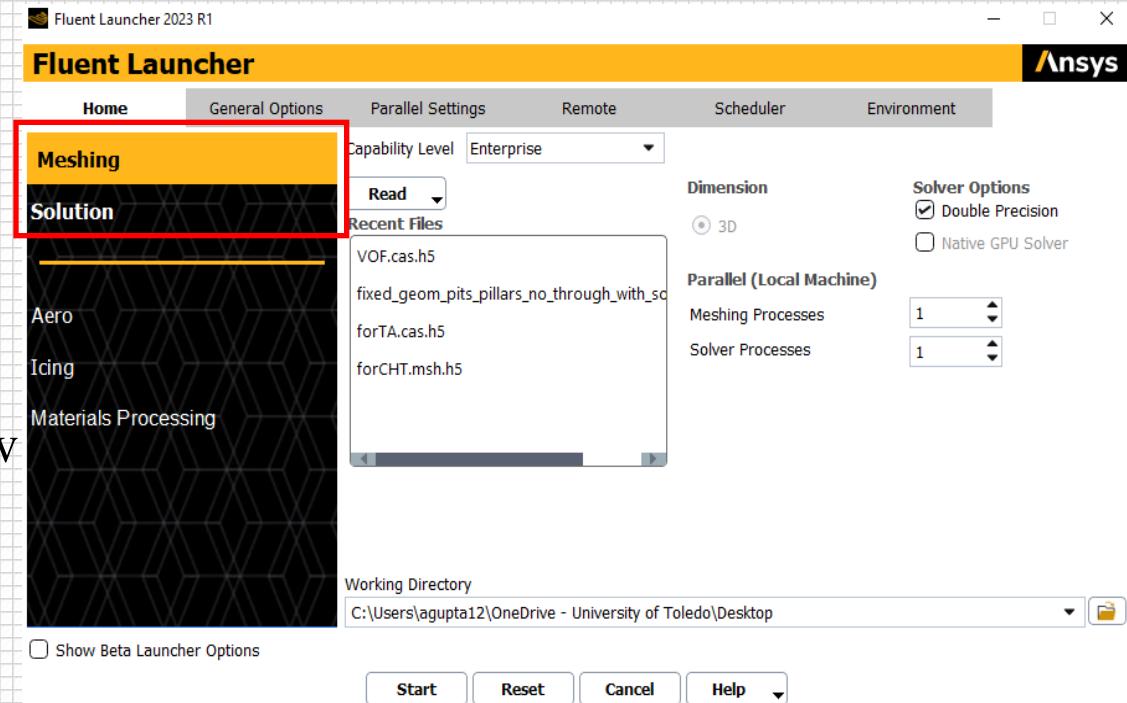
- Modes of Heat Transfer
- Geometries
- Electronics cooling- Class problem 3.8

## (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

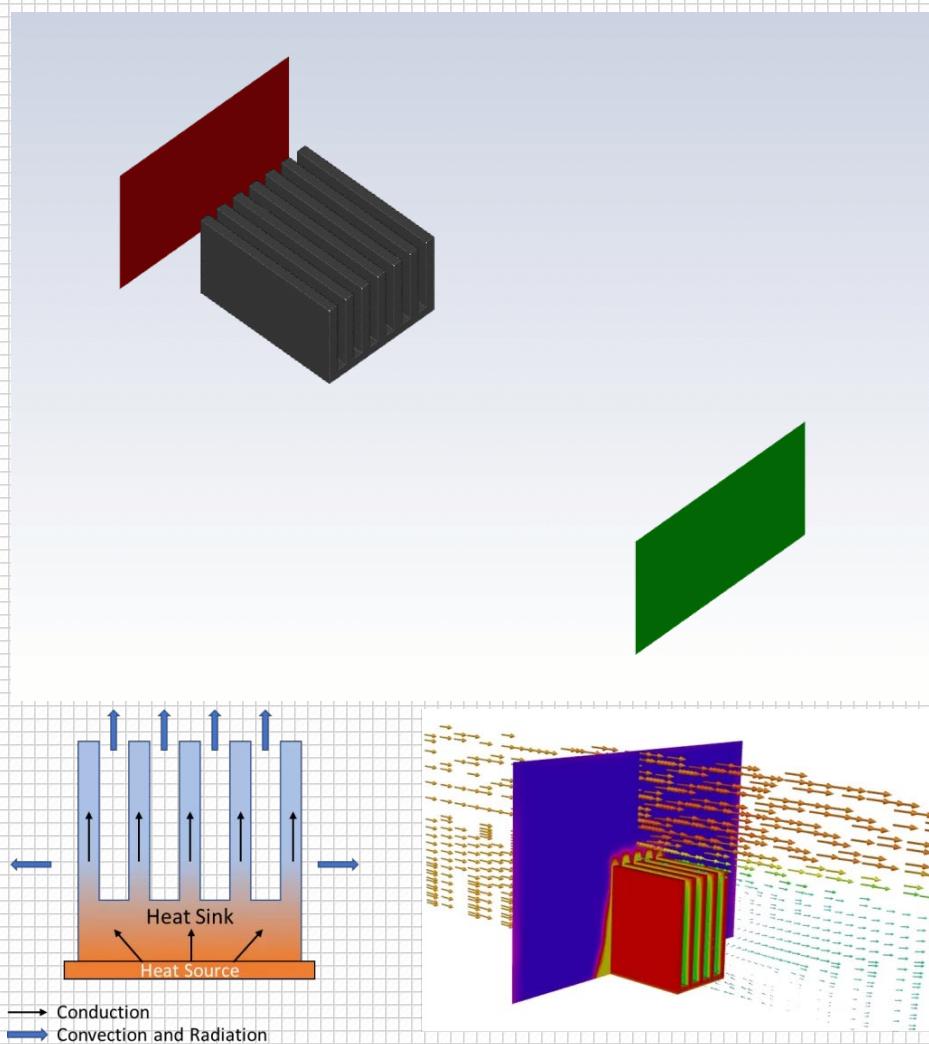
---

- 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



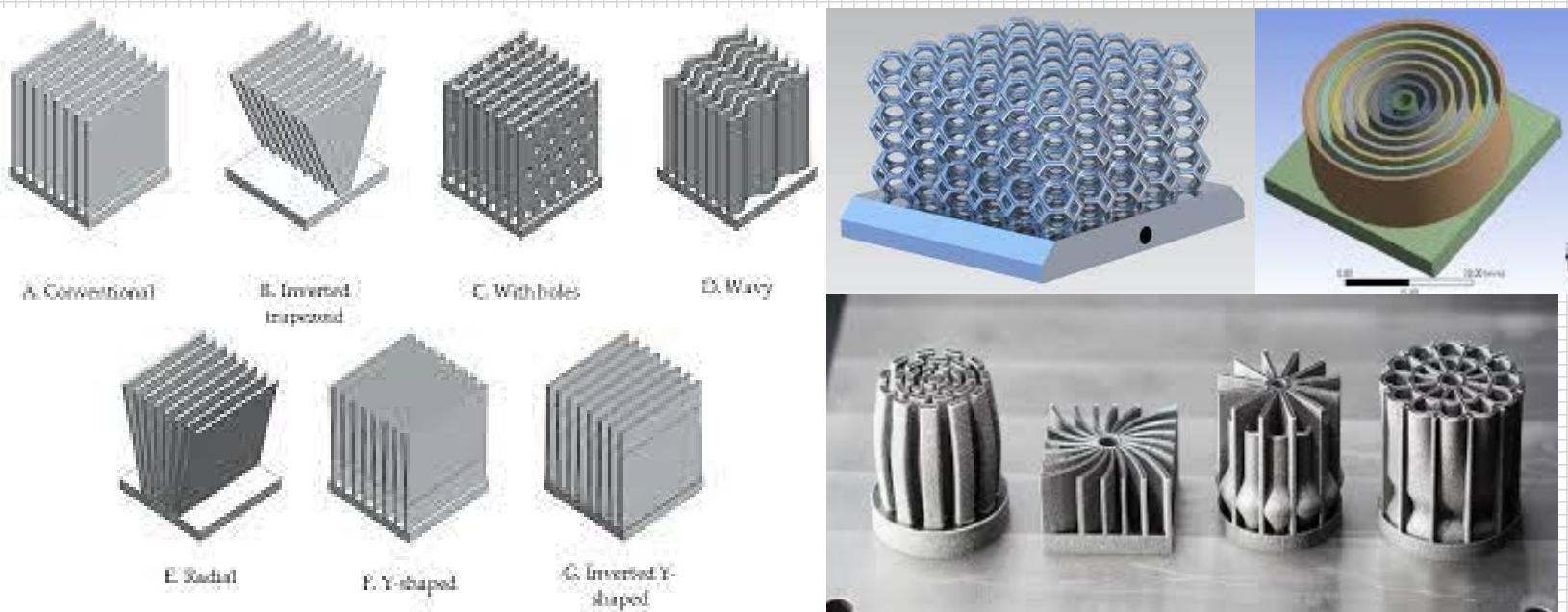
# Heat Sinks: Modes of HT

- 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

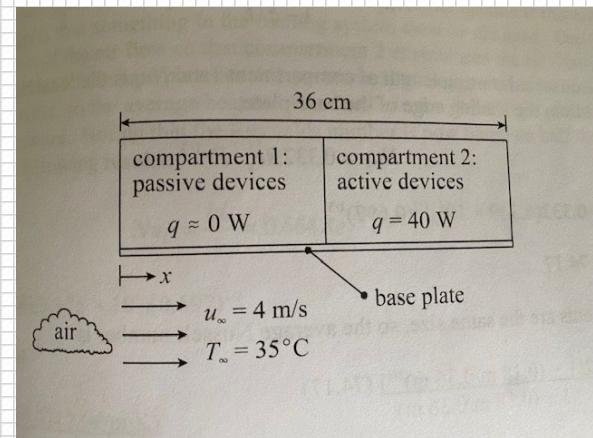
- The fins of the heat sink increase the total surface area and improve its performance when cooling electronic components
- Here are some examples from google images



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



# 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:

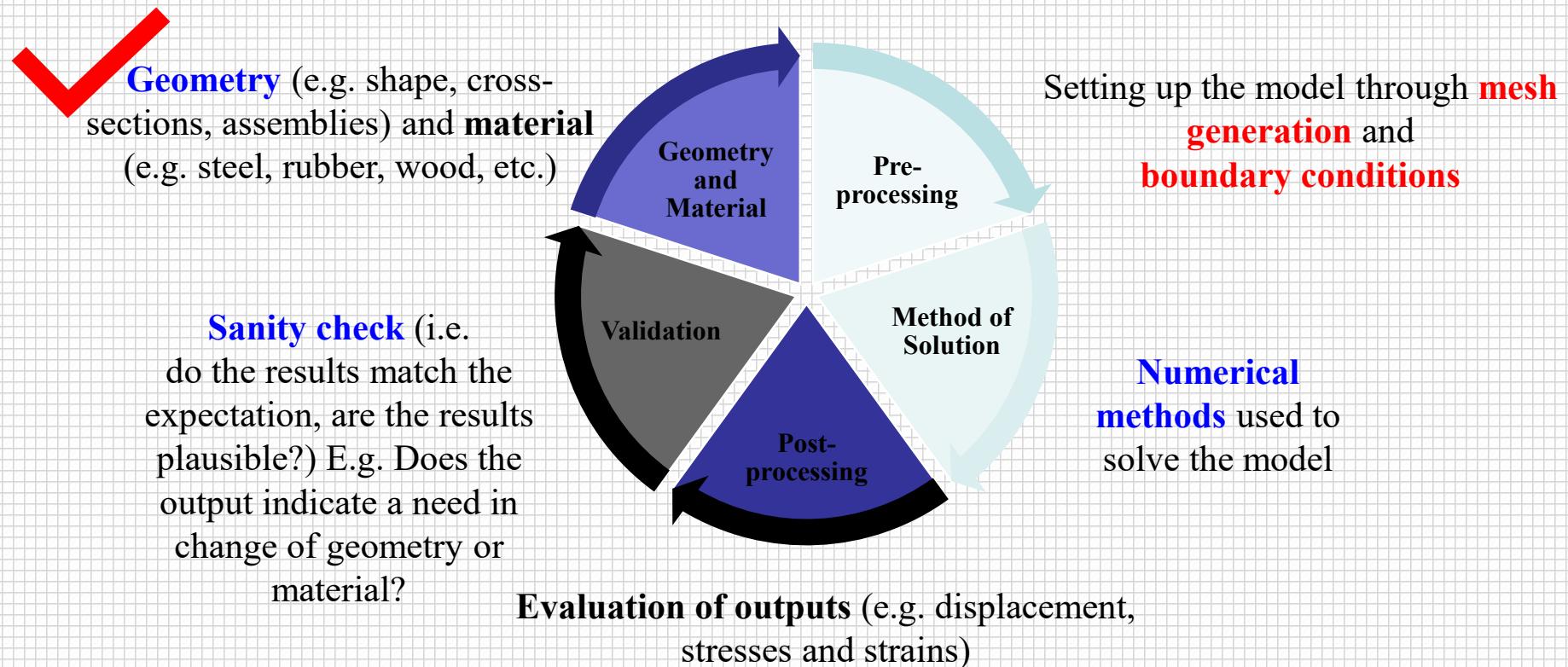
Simulate, quantify, and compare the average and maximum base plate temperature for each configuration.

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

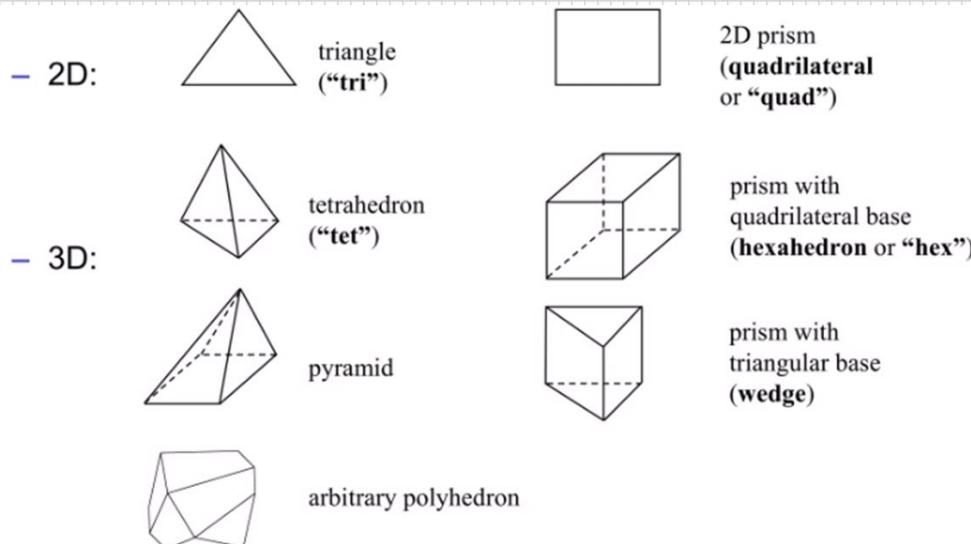
# Simulations/Digital Twinning

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



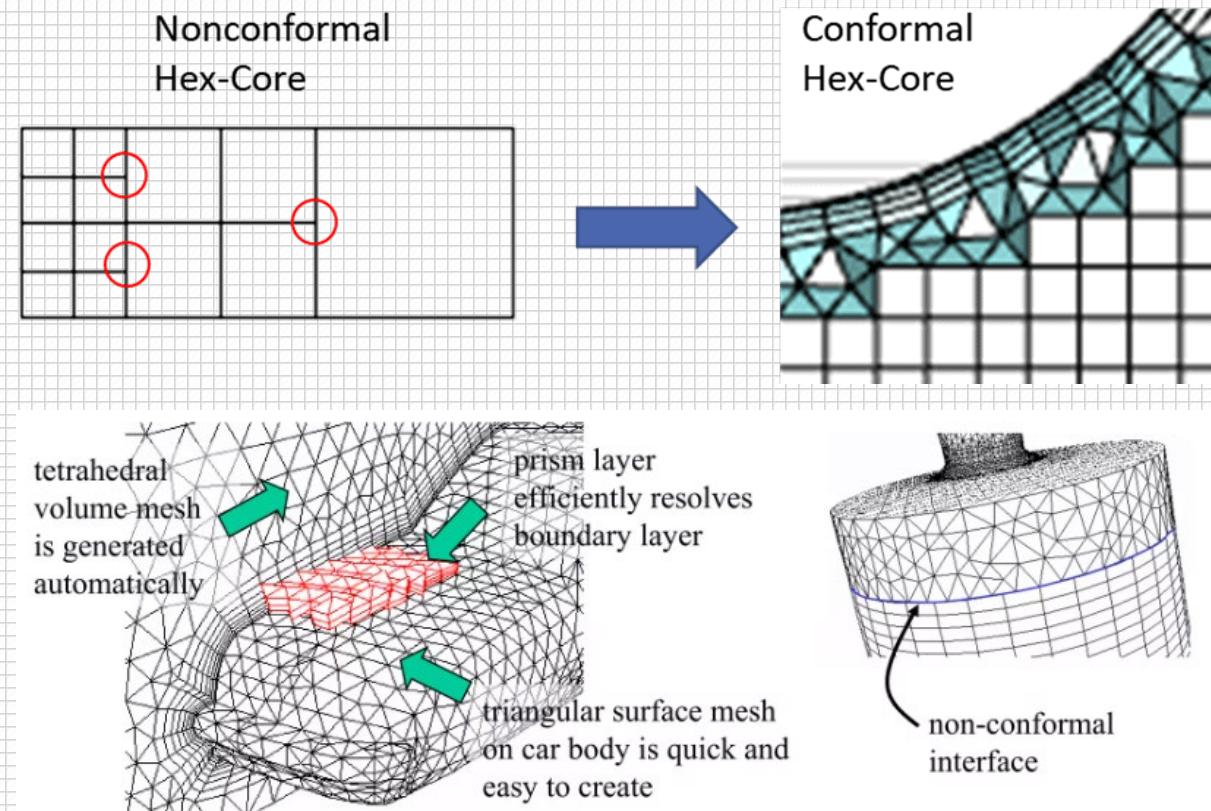
# Meshing

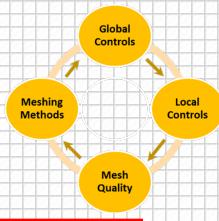
- Two-dimensional and three-dimensional grid; it is dividing complex geometries into elements that can be used to discretize a domain
- Mesh elements allow governing equations to be solved on predictably shaped and mathematically defined volumes.
- Typically, the equations solved on these meshes are 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





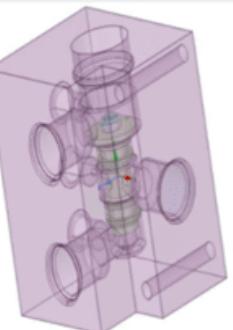
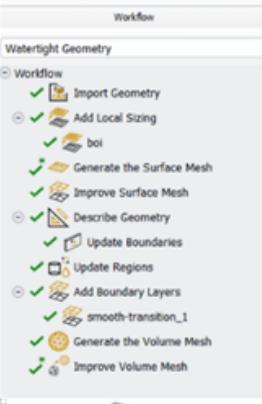
# Meshing Workflows

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

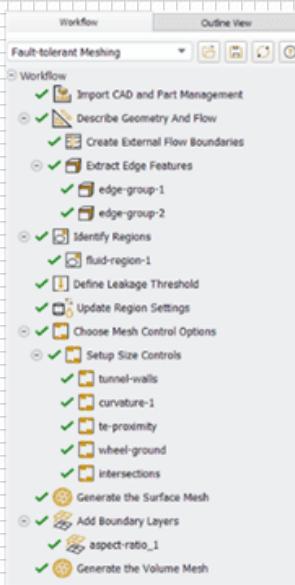
## Watertight

- 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



# Meshing Workflows

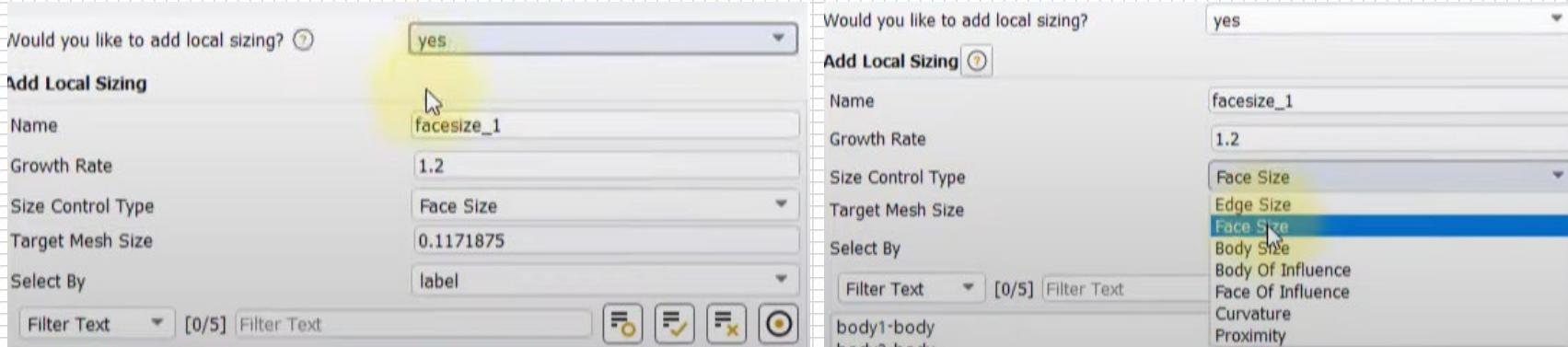
Import geometry → Surface mesh → Create solid/liquid regions → Volume mesh → Solution ...

The screenshot shows a software interface for meshing workflows. At the top, there's a toolbar with a "Switch to Solution" button, a "Selection" dropdown, a "Set Ranges" button, a "+/- Delta" input field (set to 0), a "Reset" button, and checkboxes for X Range, Y Range, Z Range, and Cutplanes. Below the toolbar, there are two tabs: "Workflow" (selected) and "Outline". The main area is titled "Watertight Geometry" and contains a list of completed steps, each with a green checkmark and an icon:

- Add Local Sizing
- bodysize\_1
- boi\_1
- facesize\_1
- faceoi\_1
- Generate the Surface Mesh **Accurate geometry resolution**
- Describe Geometry
- Update Boundaries
- Create Regions
- Update Regions
- Add Boundary Layers
- smooth-transition\_1
- Generate the Volume Mesh **Captures multiscale flow features and flow gradients**

# Local Sizing

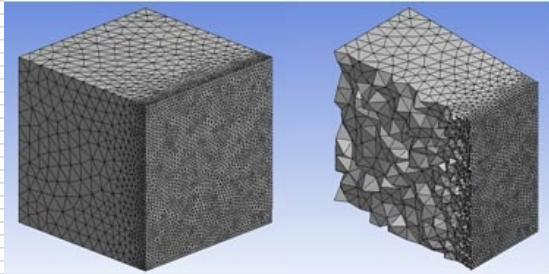
- **Controls:** Edge, Face, Body
- Mesh growth (transition) between small and large sizes based on a specified growth rate
- Curvature based refinement and angles between normals for adjacent mesh elements (curvature-based sizing)
- Number of mesh elements employed in the gaps between two geometric entities (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



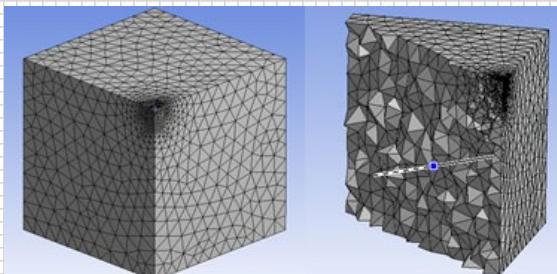
# Local Sizing

---

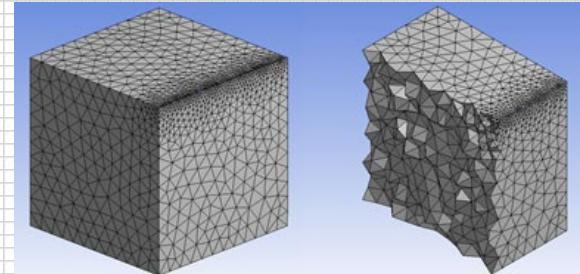
Face sizing



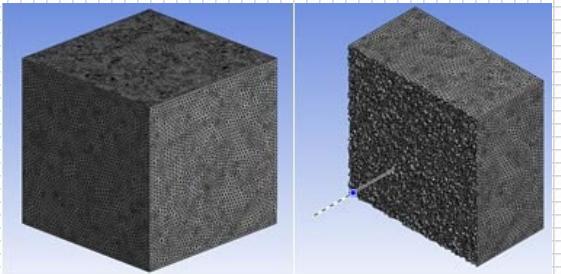
Point Sizing



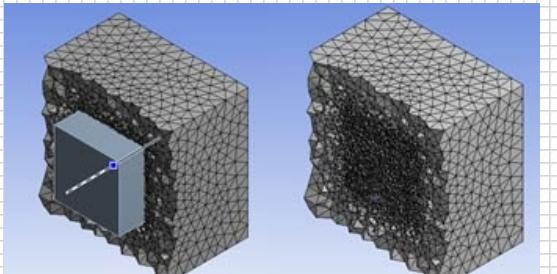
Edge Sizing



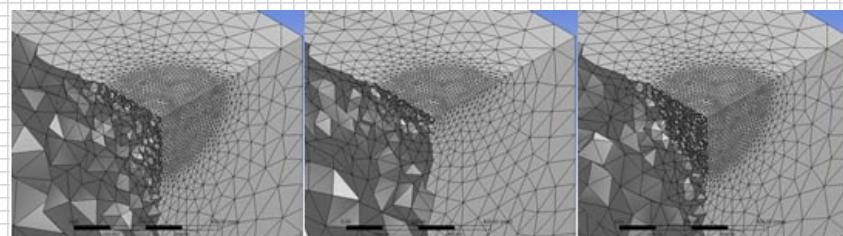
Body sizing



Body sizing using body  
of influence

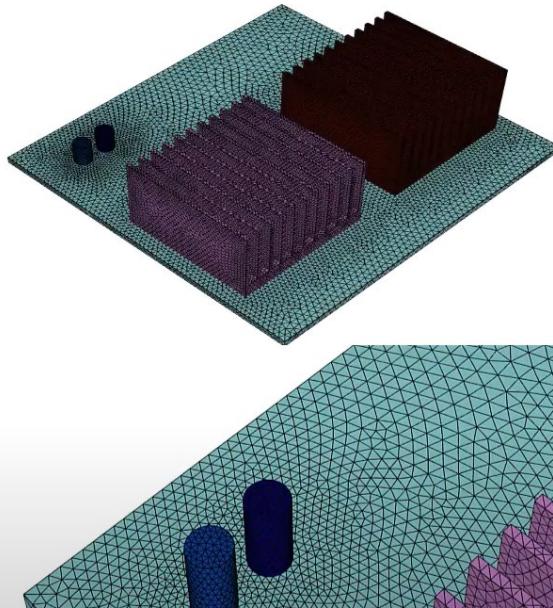


Sphere of Influence on  
Edge, Face and Body

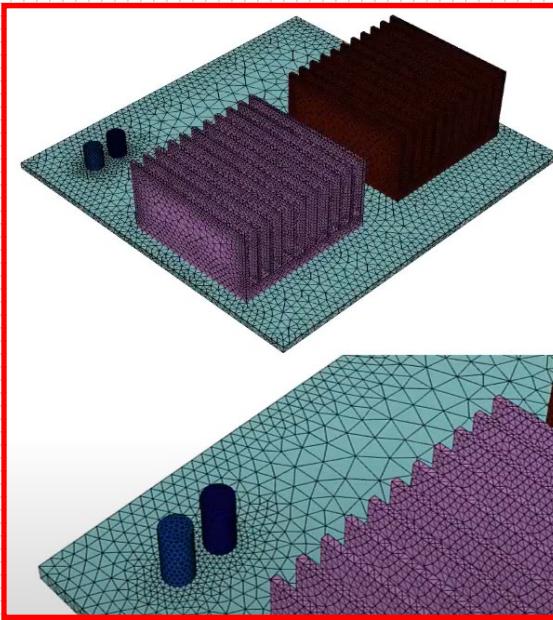


# Local Sizing: Growth Rate

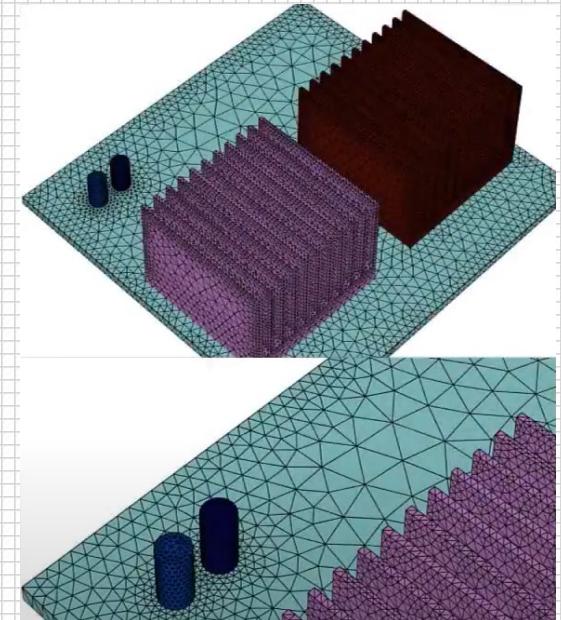
1.05



1.2

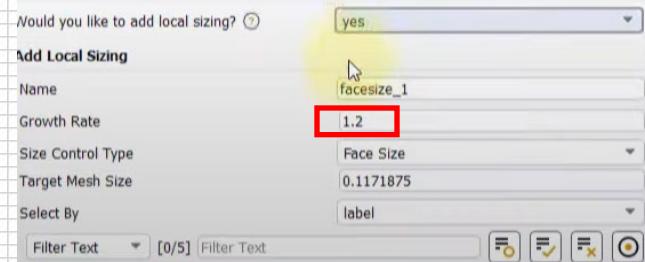


1.3



Finer mesh resolution  
away from the boundary

Higher growth rate coarsens the surface mesh



# Surface Mesh

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



**Generate the Surface Mesh**

Use Custom Size Field/Control Files? No

Minimum Size [mm] 0.3320312 Minimum and maximum size of the elements for the surface mesh

Maximum Size [mm] 8.5

Growth Rate 1.2

Size Functions Curvature & Proximity

Curvature Normal Angle [deg] 18 Maximum allowable angle for one element to span the geometry curvature

Cells Per Gap 1

Scope Proximity To edges

Draw Size Boxes

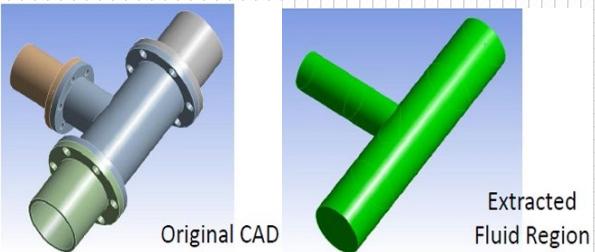
Separate Out Boundary Zones by Angle? No

Advanced Options

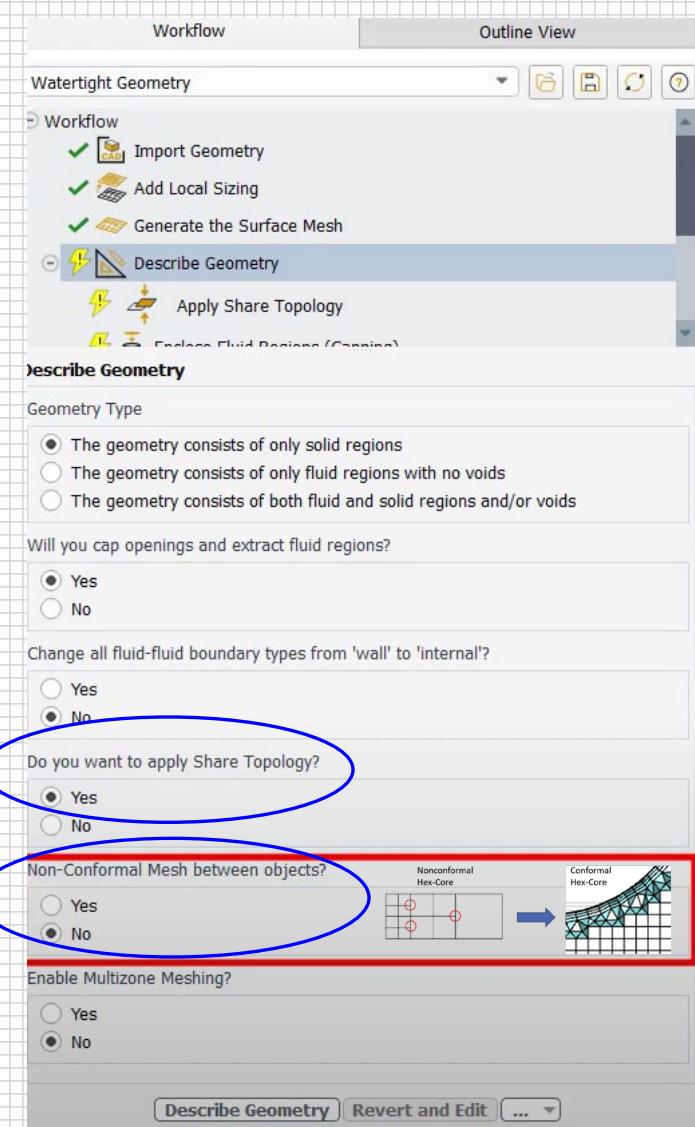
10° 18°  
(180/10 = 18 segments)

# Geometry: Capping and Share Topology

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



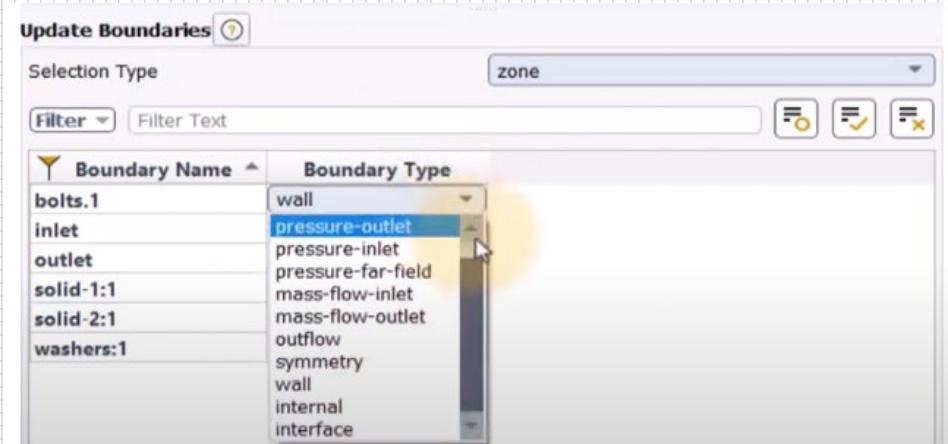
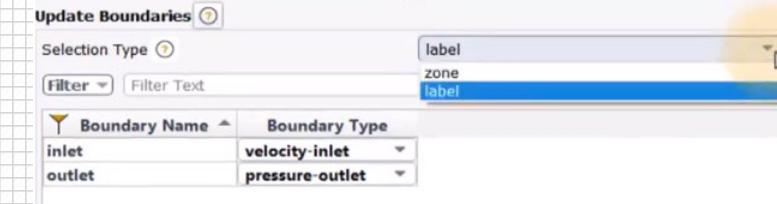
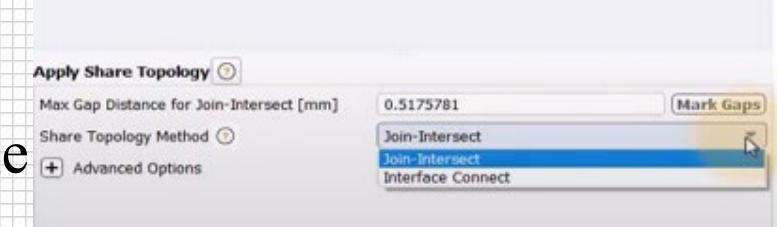
- **Share Topology:** Recommended at CAD creation phase
- Ensures overlapping areas are handled appropriately
- **Conformal mesh:** two bodies share a single face at the interface
- Generally preferred for high numerical efficiency and small simulation times



# 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

Region Name	Region Type
bolts-solid	solid
dead0	dead
dead1	dead
dead2	dead
dead3	dead
dead4	dead
dead5	dead
fluid	fluid
solid-1	solid
solid-2	solid
washers-solid	solid

Region Name	Region Type
component1-solid	solid
fluid:0	fluid
solid	solid

# Boundary Layers

**Add Boundary Layers**

Add Boundary Layers?

Name: smooth-transition\_1

Offset Method Type: smooth-transition

Number of Layers: 3

Transition Ratio: 0.272

Growth Rate: 1.2

Add in: fluid-regions

Grow on: only-walls

**Add Boundary Layers**

Add Boundary Layers?

Name: smooth-transition\_1

Offset Method Type: smooth-transition

Number of Layers: 3

Transition Ratio: 0.272

Growth Rate: 1.2

Add in: fluid-regions

Grow on: only-walls

**Methods**

- Smooth-Transition
- Last-Ratio
- Aspect-Ratio
- Uniform

**Inputs**

- Number of Layers
- Growth Rate
- Transition Ratio
- Add in
- Grow on

**Boundary Layer Mesh**

Number of boundary layers

**Where to apply boundary layer mesh**

- **Add In : Specifies region**
- **Grow On : Specifies surfaces**

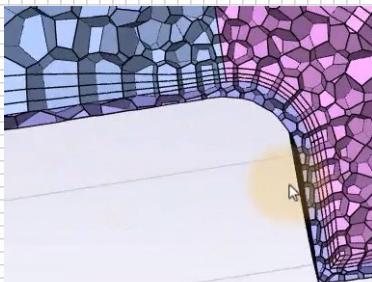
- **Transition Ratio:** ratio of the height of mesh in the last boundary layer and the first cell in volume fill
- **Growth Rate:** ratio of the thickness of the next boundary layer cell to the previous boundary layer cell

# 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

Offset Method Type

last-ratio

Number of Layers

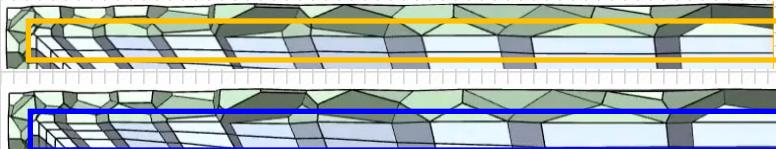
3

Transition Ratio

0.272

First Height [mm]

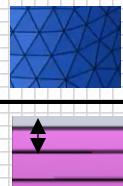
1



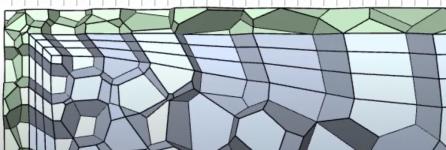
Growth layers is automatically varying to satisfy transition ratio and first height  
Results in varying boundary layer thickness

## Aspect-Ratio

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

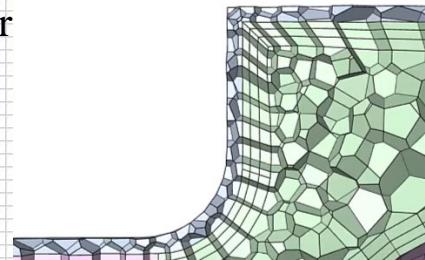


Lack of control on-  
First layer height  
Total height of  
boundary layer mesh



## 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

Workflow      Outline View

Watertight Geometry

Workflow

- Import Geometry
- Add Local Sizing
- Generate the Surface Mesh
- Describe Geometry
  - Update Boundaries
  - Update Regions
- Add Boundary Layers
- Generate the Volume Mesh

Fill With ?

Buffer Layers

Peel Layers

Min Cell Length [mm]

hexcore

polyhedra

poly-hexcore

hexcore

tetrahedral

Generate the Volume Mesh ?

Fill With

Growth Rate

Max Cell Length [mm]

hexcore

2

1

2

128

Generate the Volume Mesh ?

Fill With

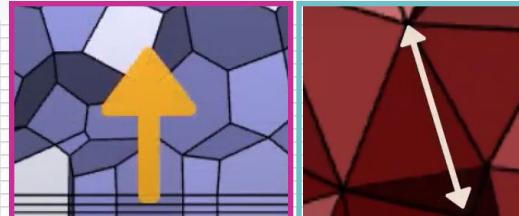
Growth Rate

Max Cell Length [mm]

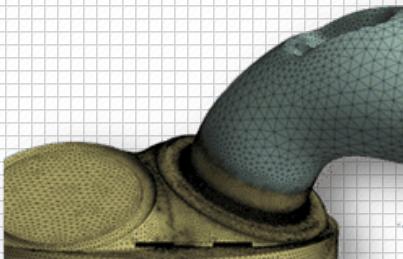
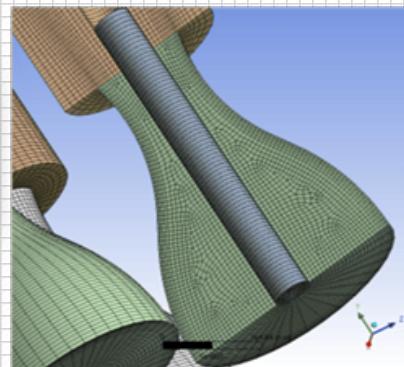
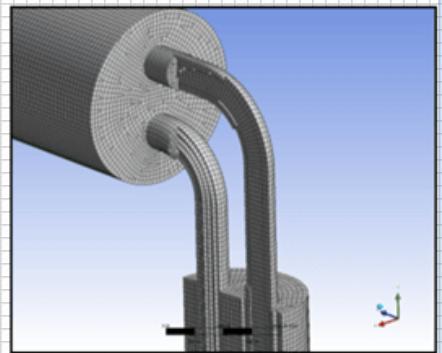
tetrahedral

1.2

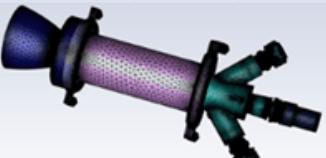
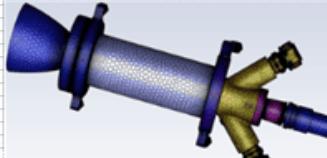
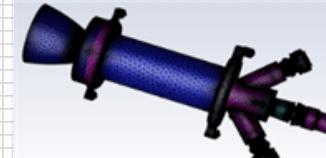
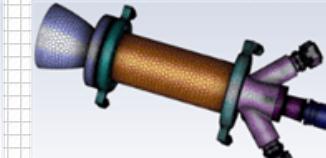
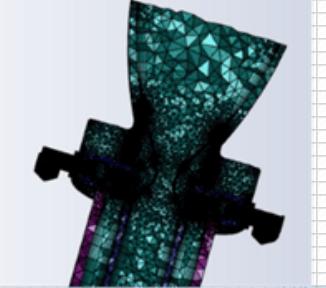
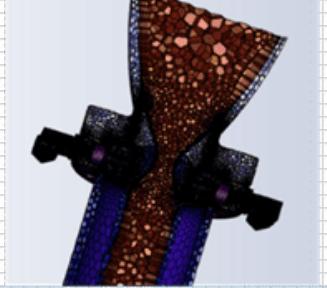
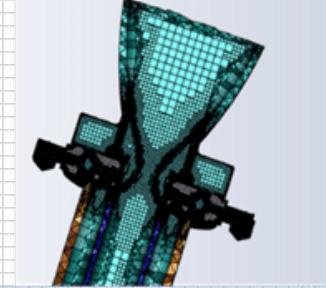
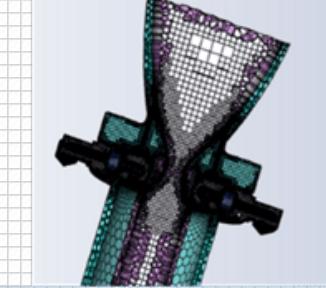
165.8264



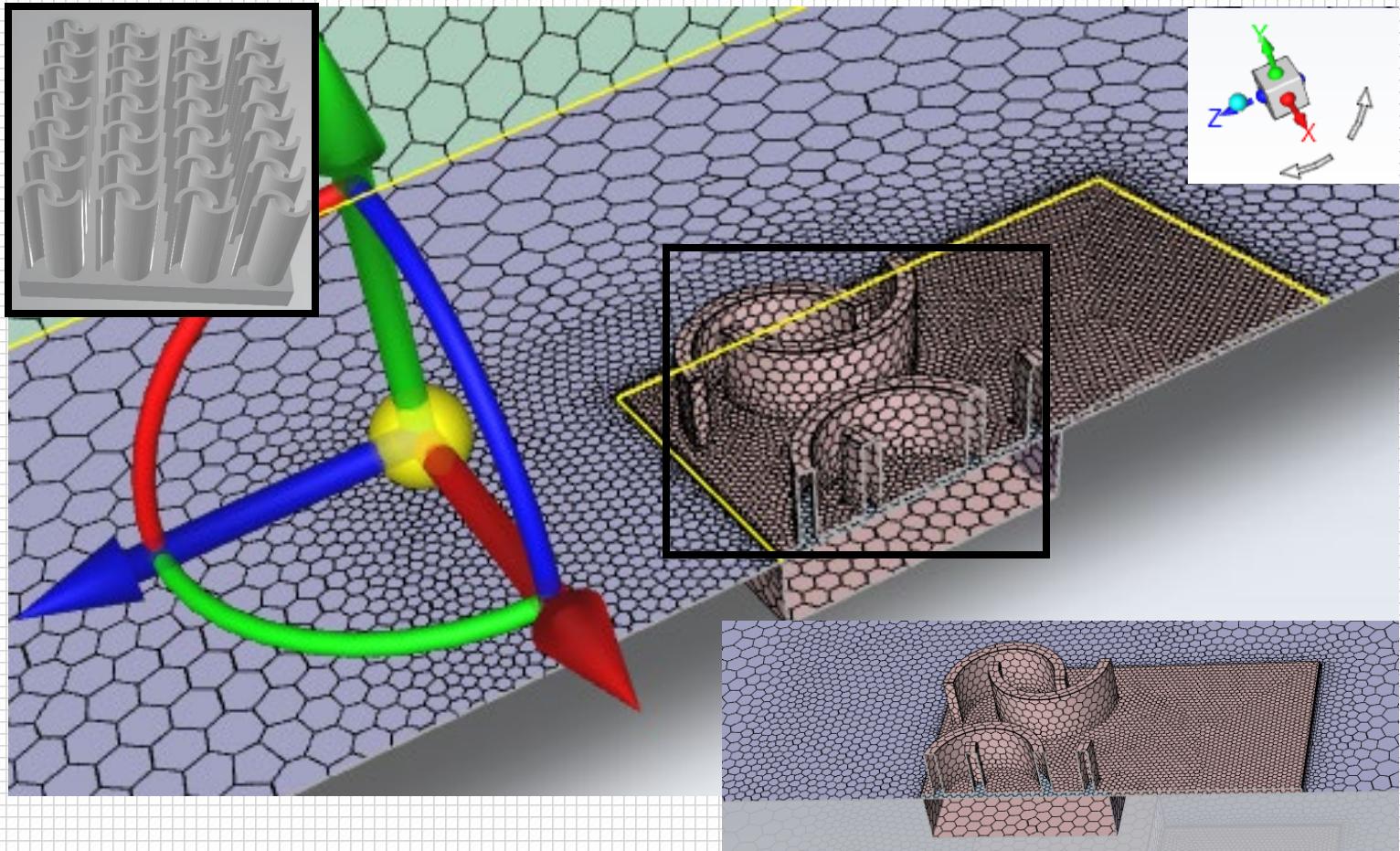
# Volume Mesh

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			

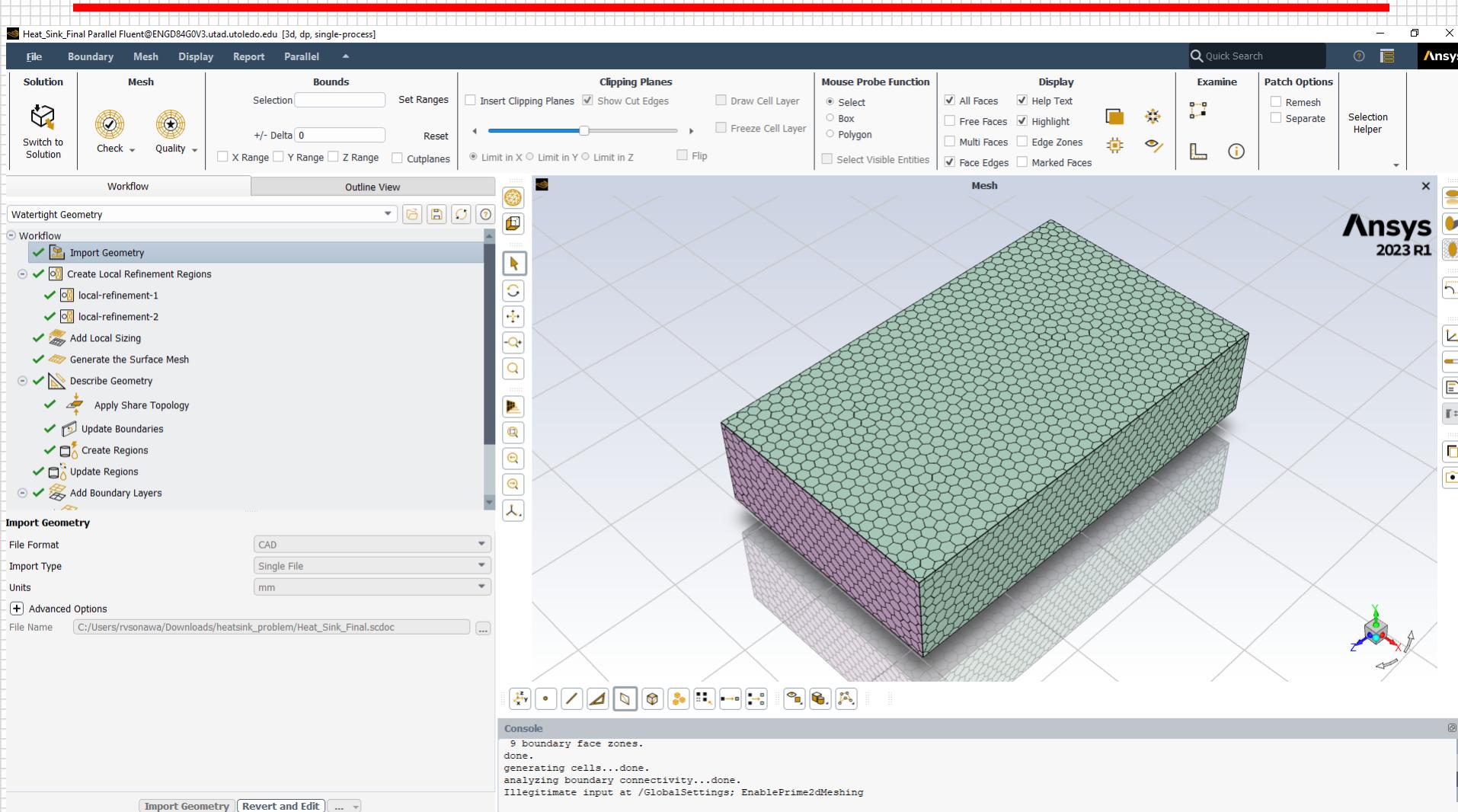
# Volume Mesh

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				

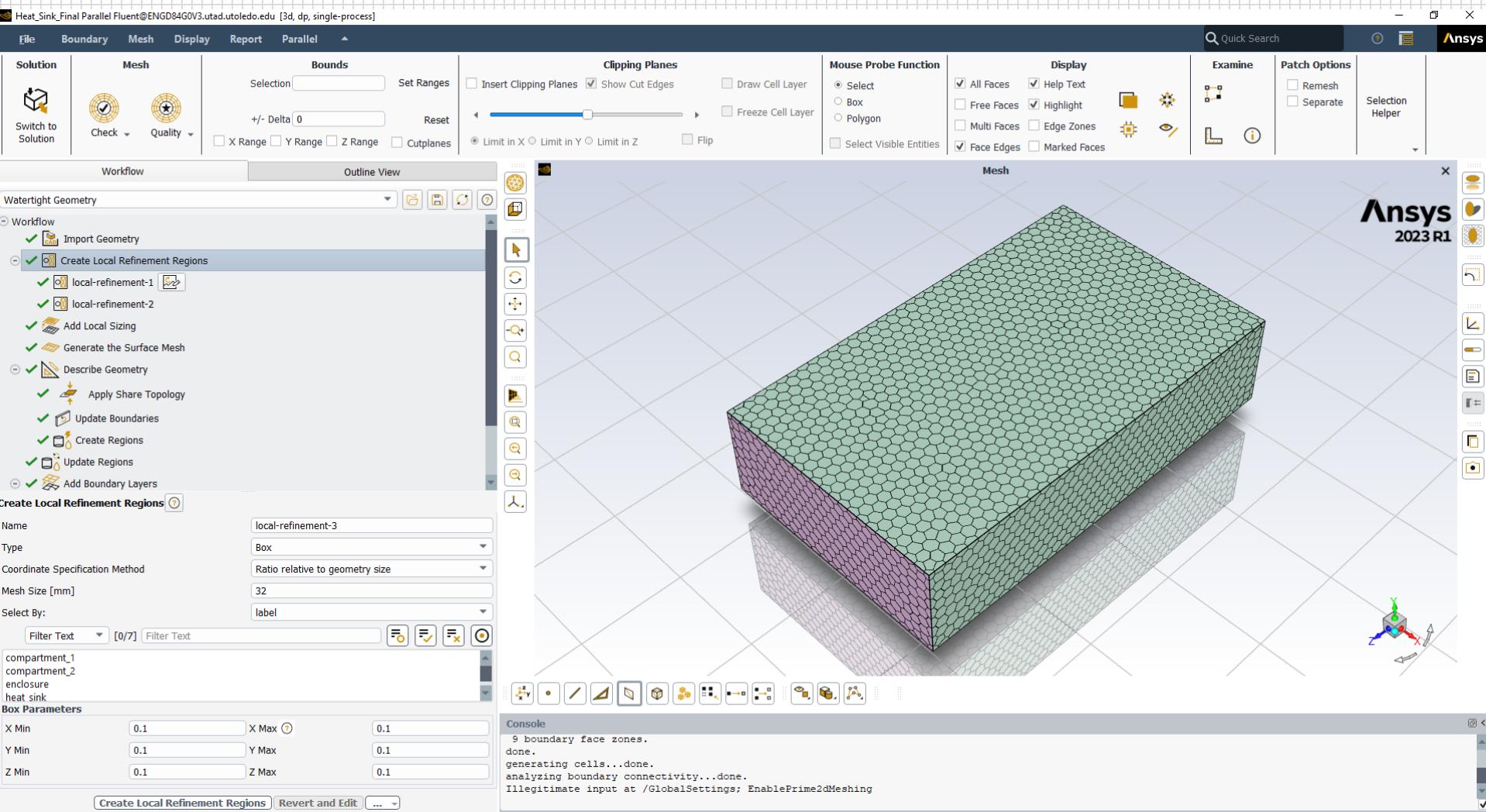
# Heat Sink Mesh: Sample Geometry



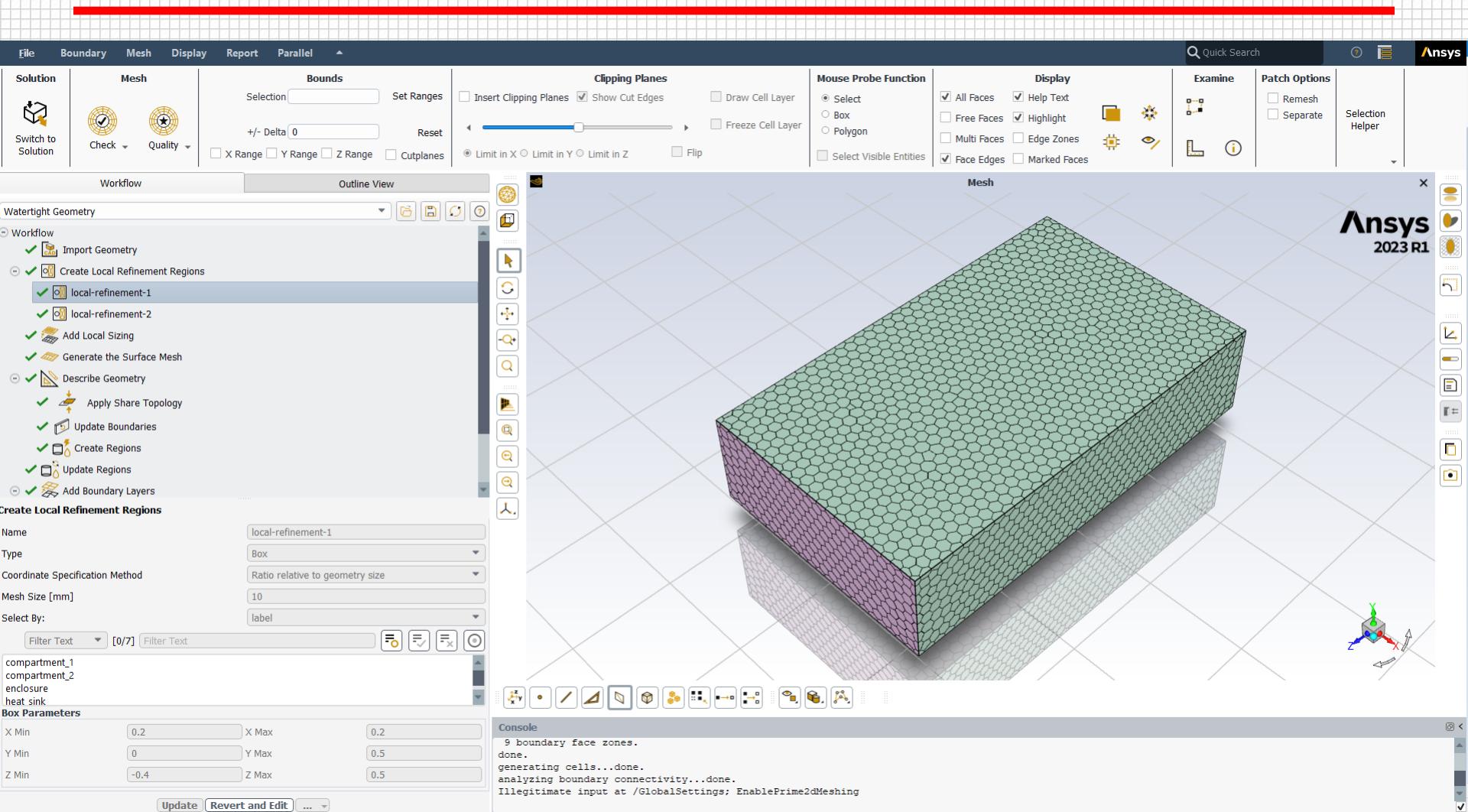
# Heat Sink Mesh: Import Geometry



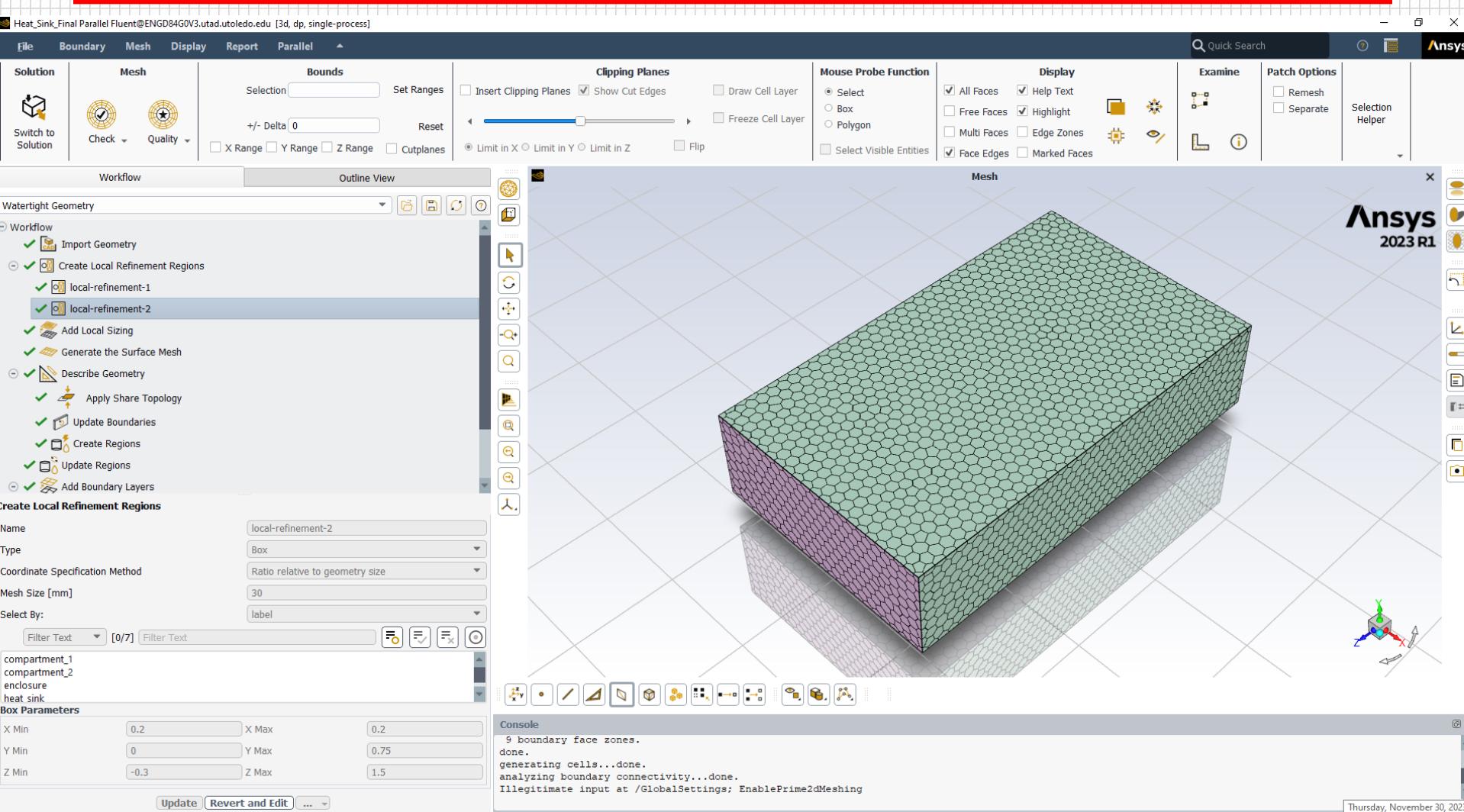
# Heat Sink Mesh: Local Refinements



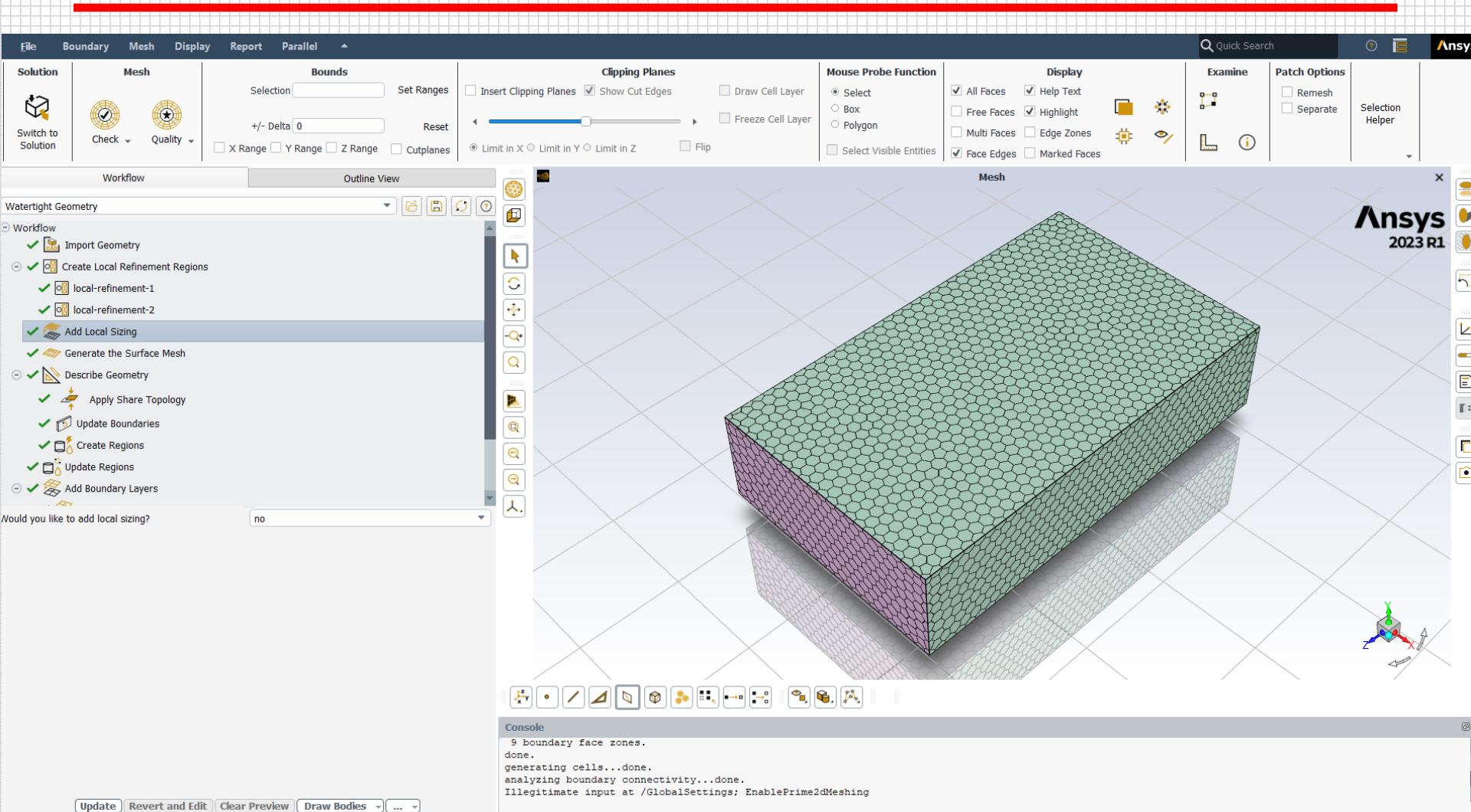
# Heat Sink Mesh: Local Refinements



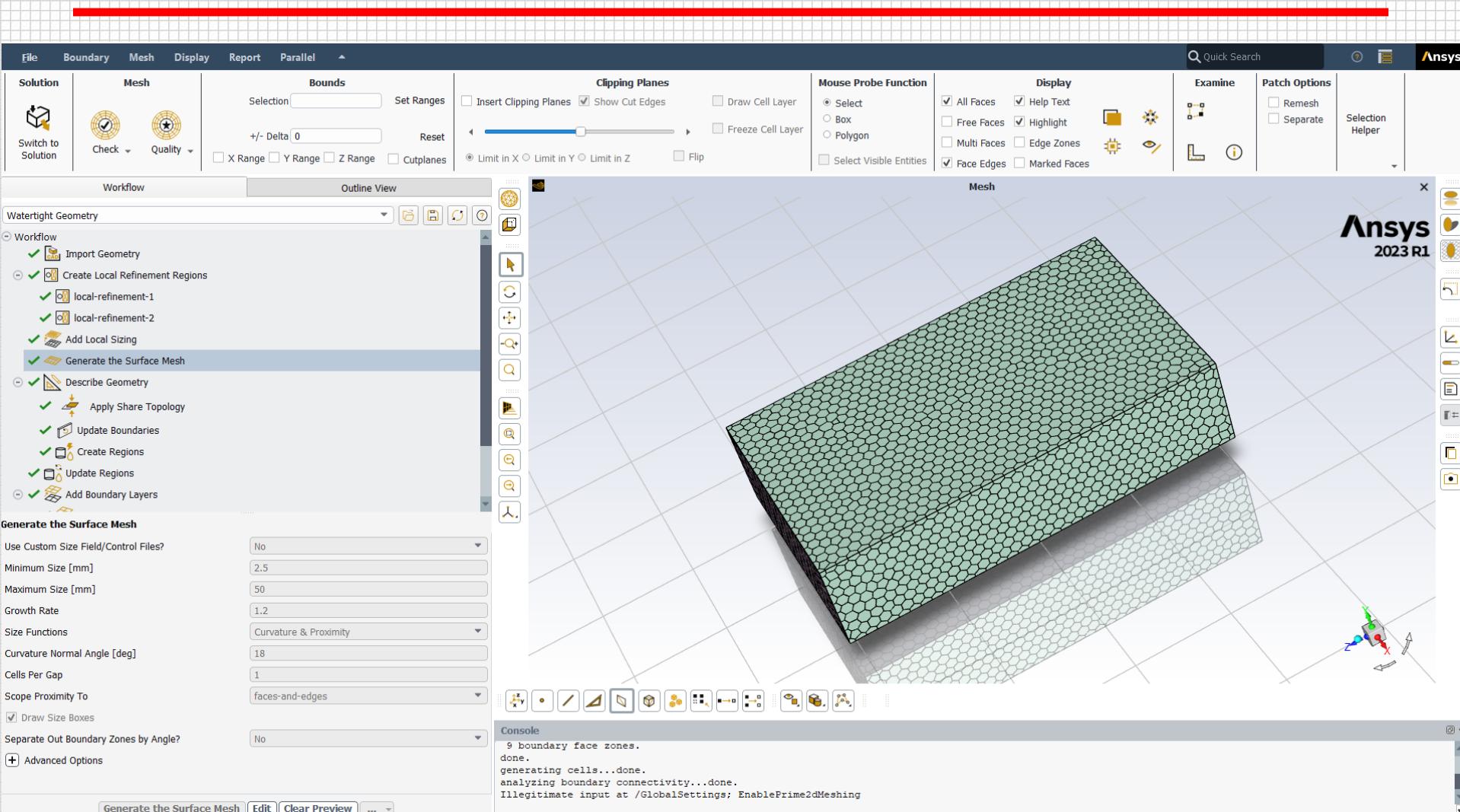
# Heat Sink Mesh: Local Refinements



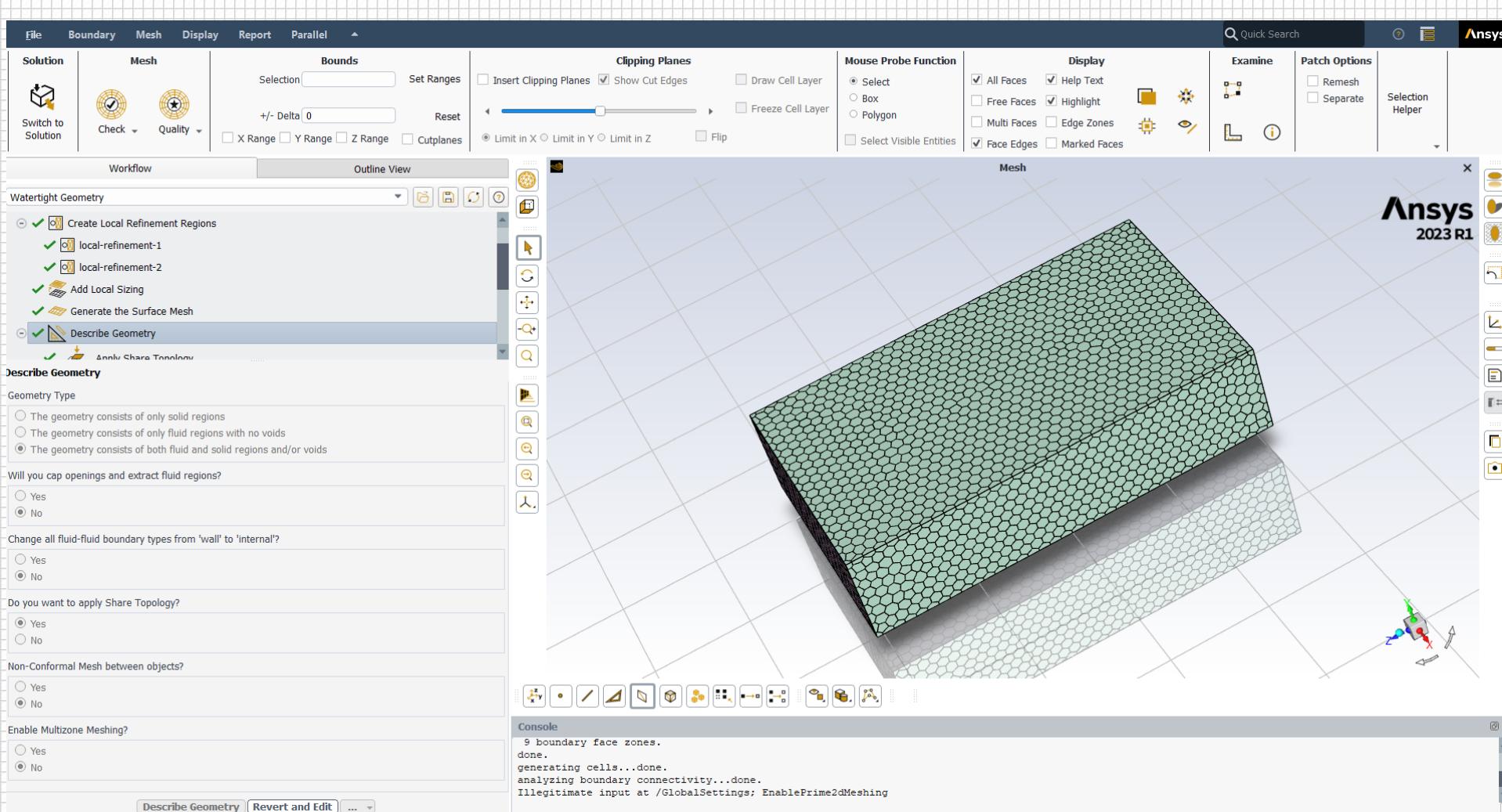
# Heat Sink Mesh: Local Sizing



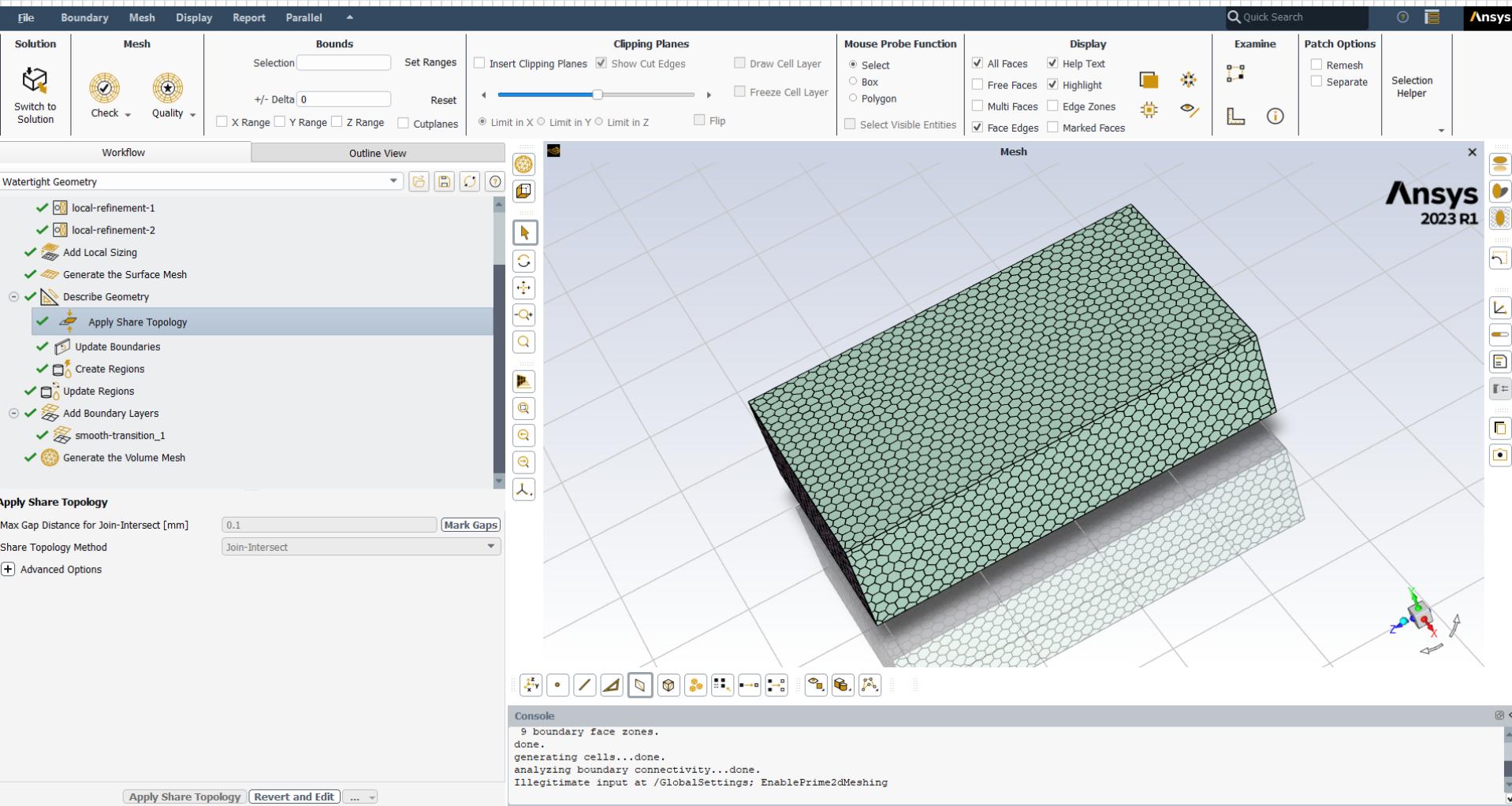
# Heat Sink Mesh: Surface Mesh



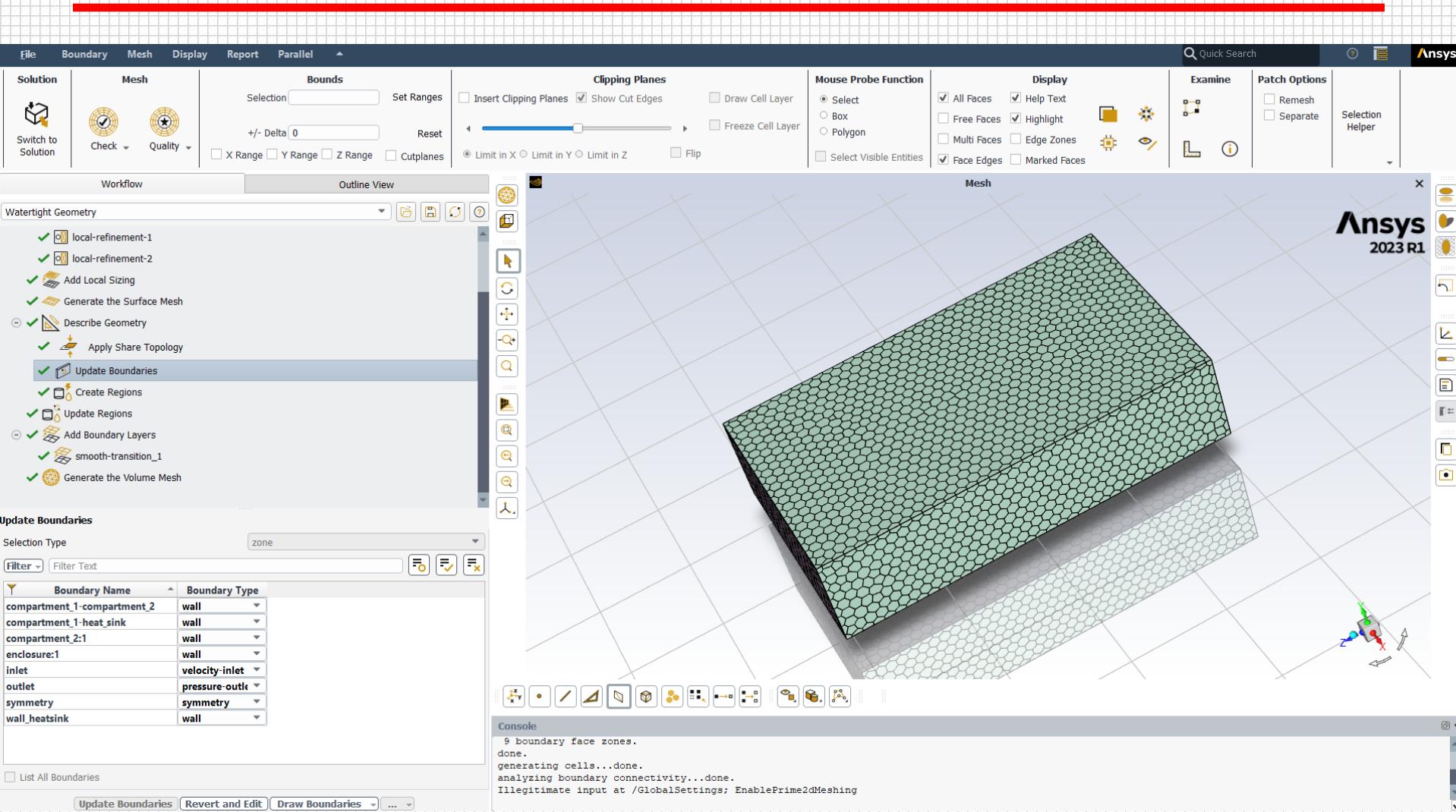
# Heat Sink Mesh: Describe Geometry



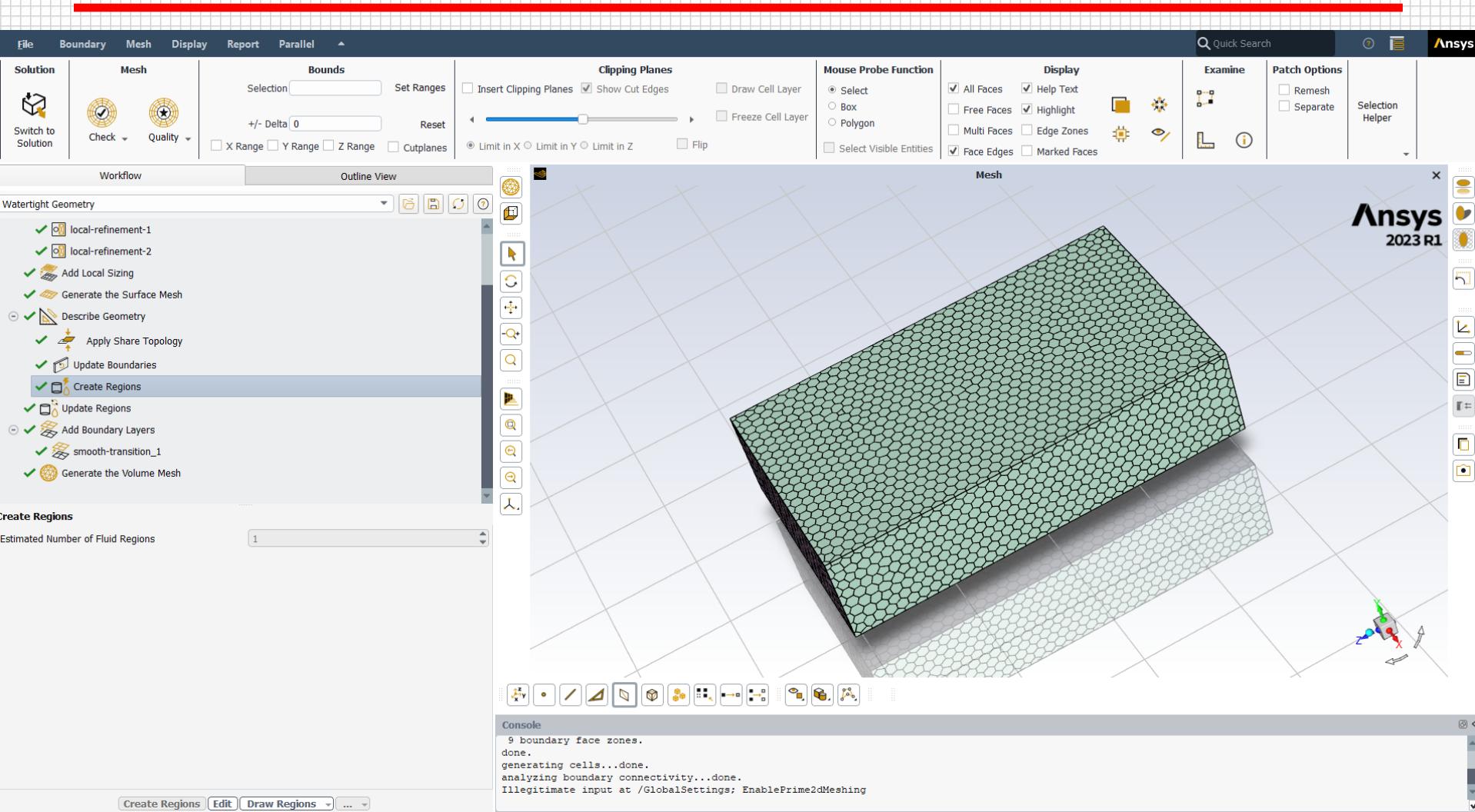
# Heat Sink Mesh: Apply Share Topology



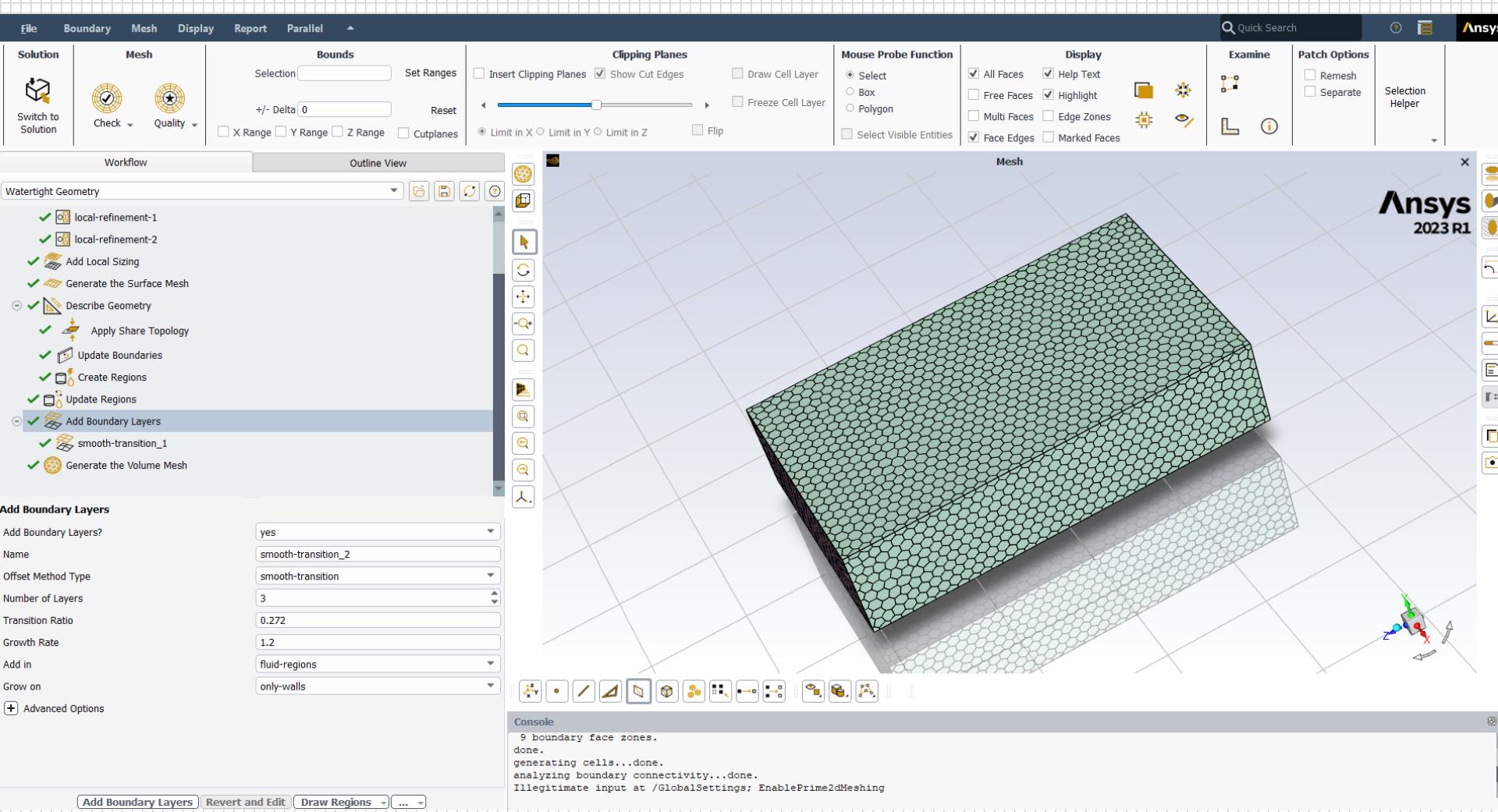
# Heat Sink Mesh: Update Boundaries



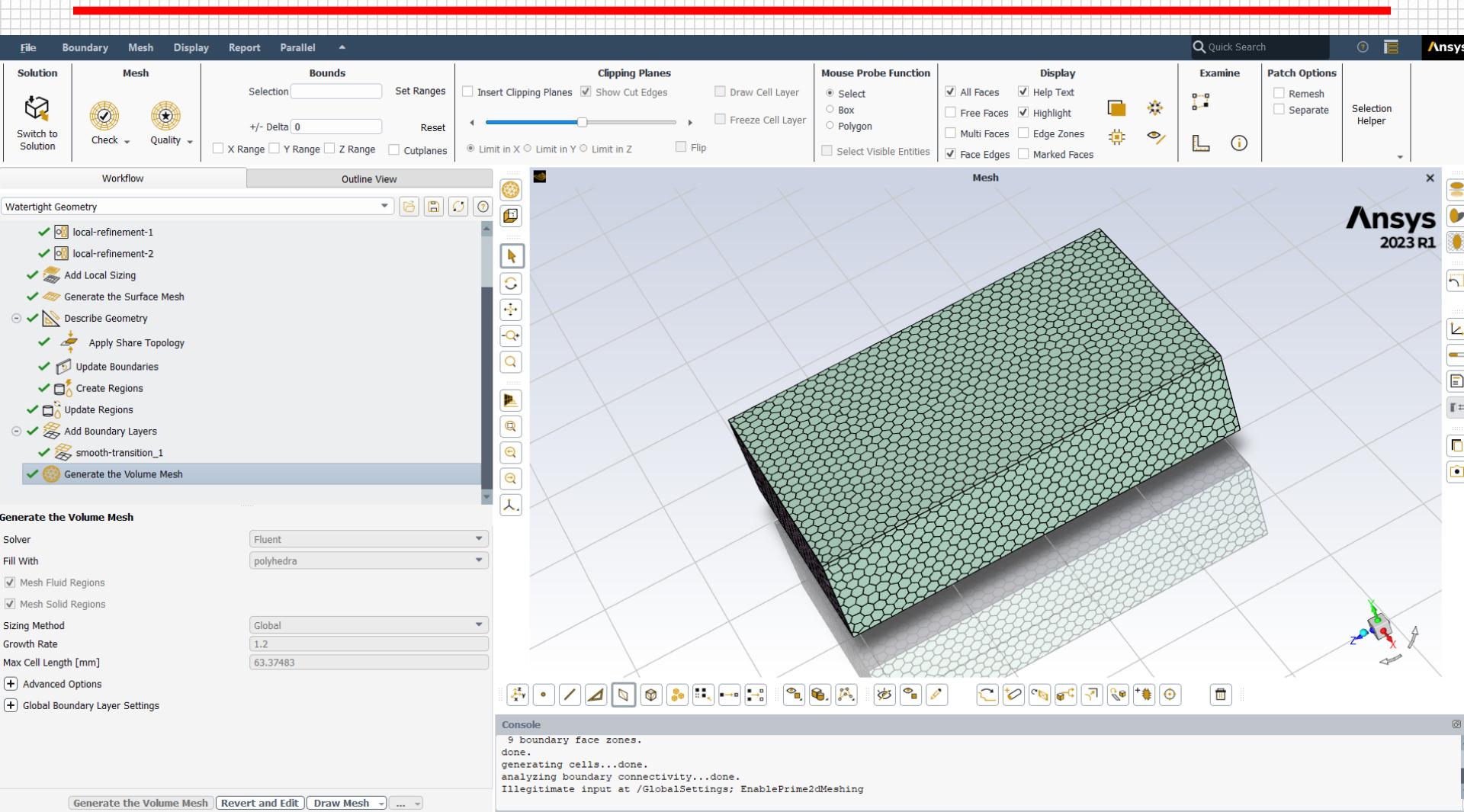
# Heat Sink Mesh: Create Regions



# Heat Sink Mesh: Add Boundary Layers



# Heat Sink Mesh: Generate the Volume Mesh



# Mesh Quality

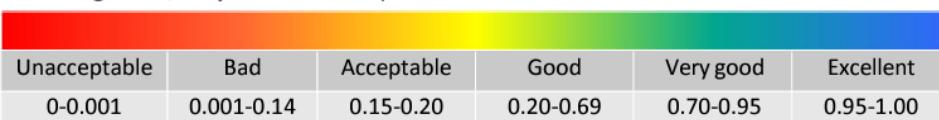
## Factors

Geometry issues  
Meshing parameters  
Meshing methods

Skewness mesh metrics spectrum



Orthogonal Quality mesh metrics spectrum



# Mesh Parameters and Quality

```
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 parameters

```
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.
```

## Mesh check

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

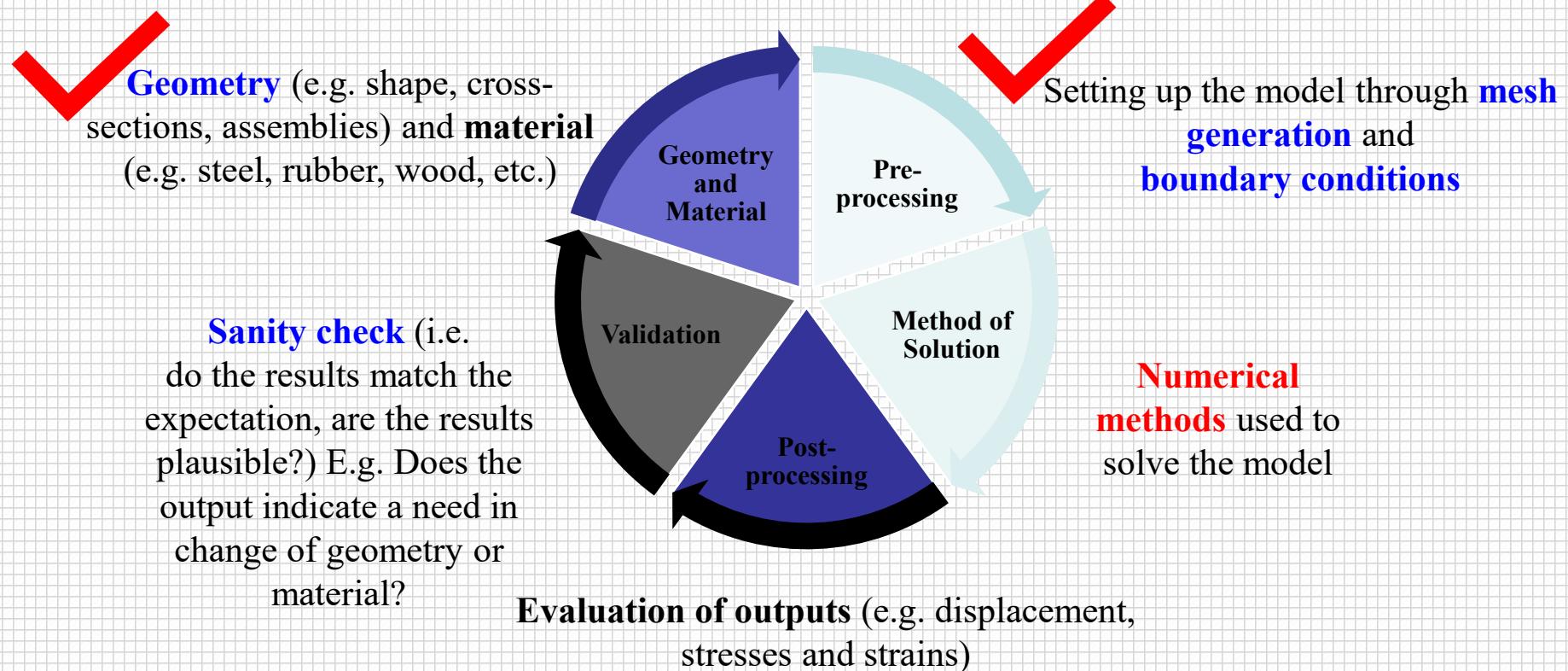
```
Marking Face in Range (0.53181627 0.58181627)...
```

```
-----  
Cluster Number Entity Count  
-----  
 1      1  
 2      1  
 3      1  
 4      1
```

## Surface mesh quality

# 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



# Day 2 Fluent Workshop: Heat Sink Design

---



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



# Topics

## (Day 1) Heat Sink

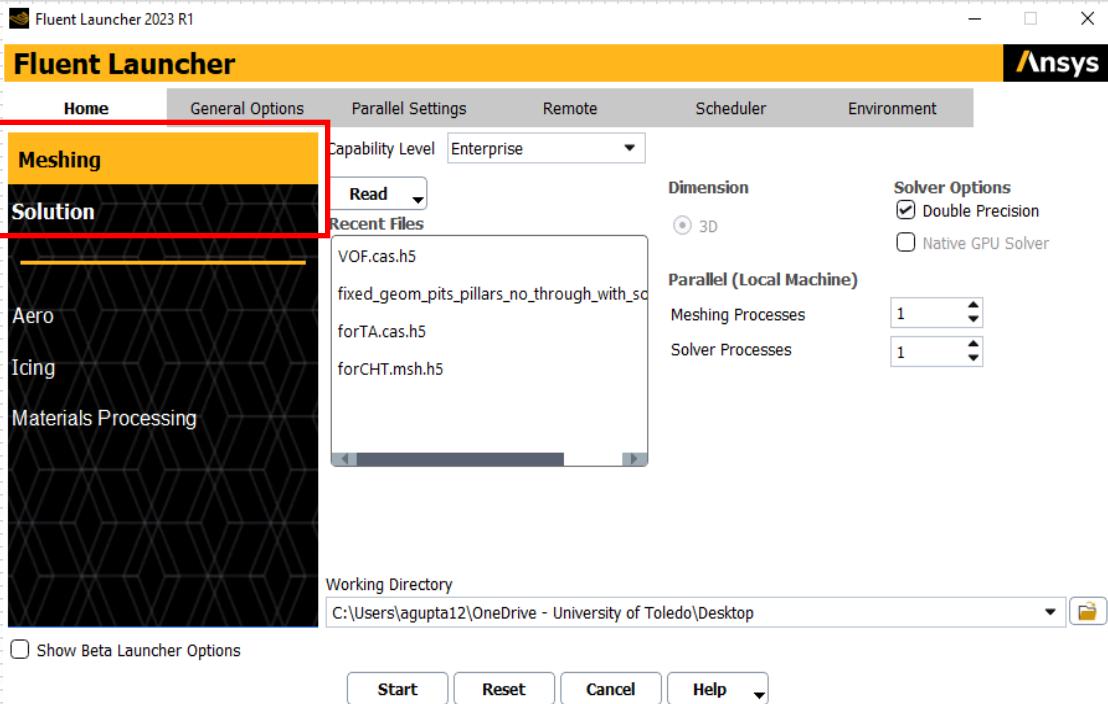
- Modes of Heat Transfer
- Geometries
- Electronics cooling- Class problem 3.8

## (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

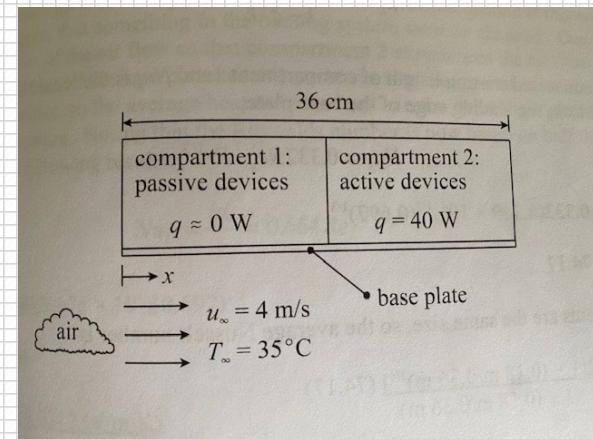
- **Materials:** fluid (air) , solid (copper )

- **Material property:**

Copper:  $\rho = 8928 \text{ kg/m}^3$ ,  $k = 387.6 \text{ W/m.K}$

Air:  $\rho = 1.225 \text{ kg/m}^3$ ,  $k = 0.0242 \text{ W/m.K}$

$$\mu = 1.7894 \times 10^{-5} \text{ kg/m.s}$$



- **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:  $q = 40 \text{ W}$ , 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

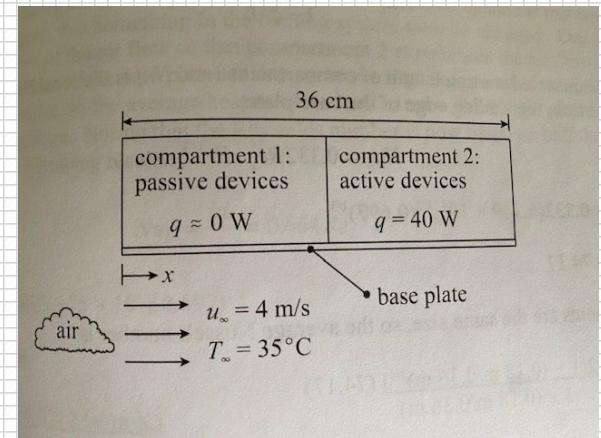
- **Materials:** fluid (air) , solid (aluminum)

- **Material property:**

Aluminum: :  $\rho = 2719 \text{ kg/m}^3$ ,  $k = 202.4 \text{ W/m.K}$

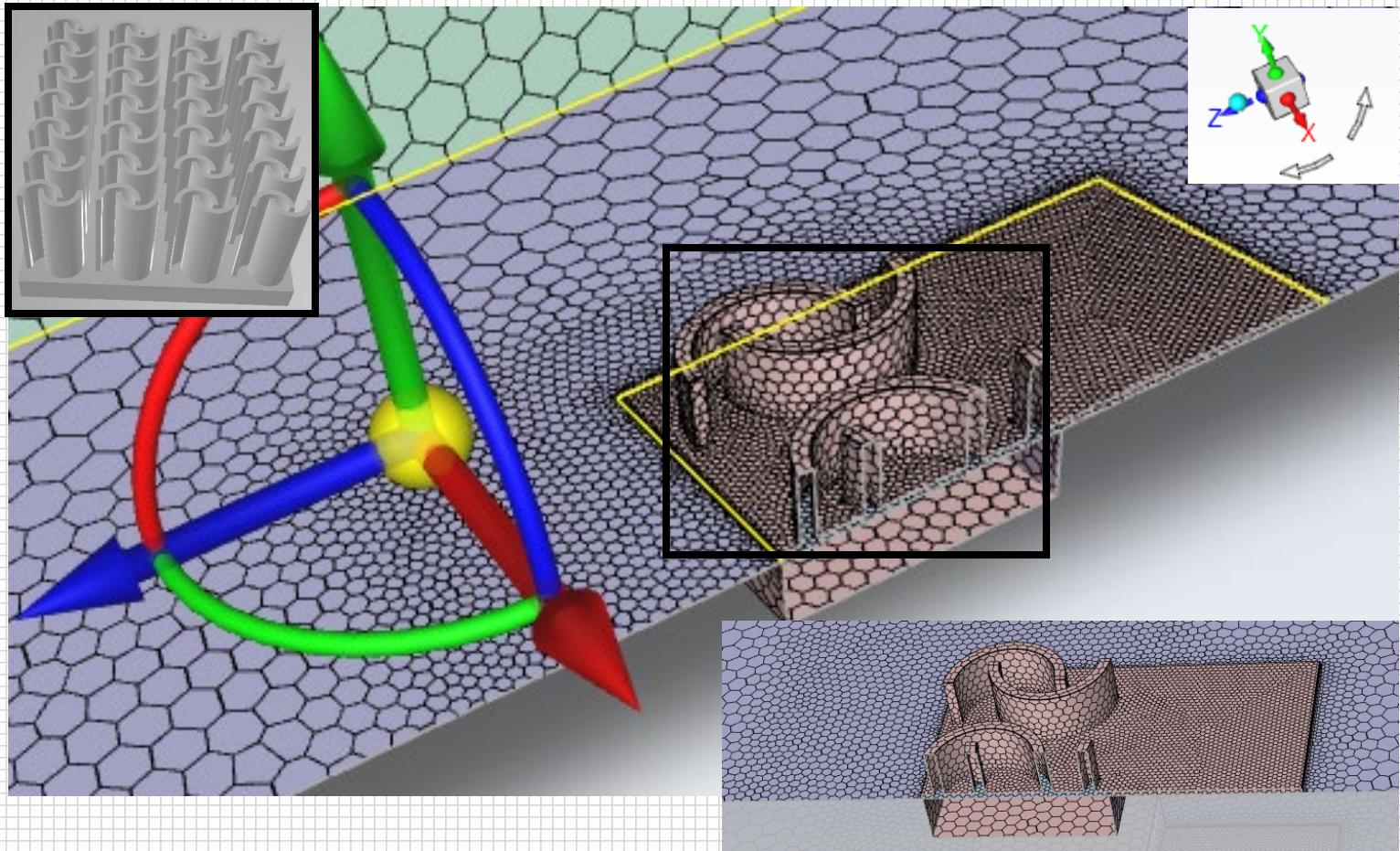
Air:  $\rho = 1.225 \text{ kg/m}^3$ ,  $k = 0.0242 \text{ W/m.K}$

$$\mu = 1.7894 \times 10^{-5} \text{ kg/m.s}$$

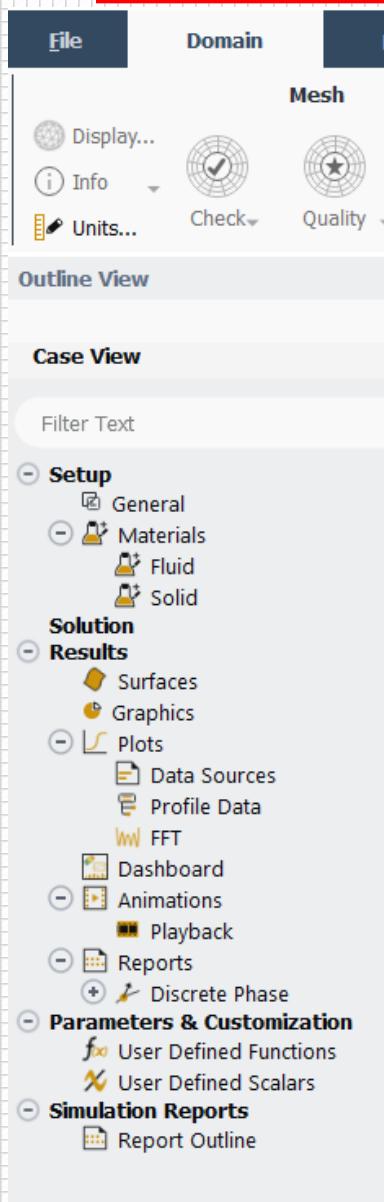


- **Physical models:** viscous and turbulent, Steady State using pressure-based coupled
- **Boundary conditions:** air inlet velocity = 4 m/s,  $T = 308 \text{ K}$ ,  
outlet (pressure outlet): open to atmosphere (Gauge pressure = 0 Pa)  
heating surface:  $q = 40 \text{ W}$ , fluid walls (wall): adiabatic

# 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: shear 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:  $k = 0.0312 \text{ W/m.K}$ ,  $v = 22.68 \times 10^{-6} \text{ m}^2/\text{s}$ ,  $\text{Pr} = 0.697$

# Solution: Workflow

The image displays five screenshots of a software interface, likely ANSYS Fluent, illustrating the workflow for solving a simulation.

- Top Screenshot:** Shows the main menu bar with tabs: File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, Design, Parametric, and a search bar. Under the Physics tab, there are sections for Mesh (Display, Info, Units, Check, Quality, Make Polyhedra), Zones (Combine, Delete, Append, Separate, Deactivate, Replace Mesh, Adjacency, Activate, Replace Zone), Interfaces (Mesh, Dynamic Mesh, Mixing Planes, Overset, Gap Model), Mesh Models (Turbo Models, Turbo Workflow, Turbo Create, Turbo Topology, Spectral Content, Periodic Instancing), and Turbomachinery (Adapt, Manual, Automatic, Controls, Manage, Create, Manage).
- Second Screenshot:** Shows the Physics tab selected. It includes sections for Solver (Operating Conditions, Reference Values), Models (Radiation, Multiphase, Structure, Energy, Heat Exchanger, Species, Acoustics, Viscous, Discrete Phase, More), Materials (Create/Edit), Zones (Cell Zones, Boundaries, Profiles), and Model Specific (Discrete Phase, DTRM Rays, Conduction Manager, PCB Model).
- Third Screenshot:** Shows the User-Defined tab selected. It includes sections for Field Functions (Custom, Units, Parameters, Named Expressions), User Defined (Function Hooks, Memory, 1D Coupling, Scalars, Fan Model), and Model Specific (Execute on Demand, Read Table).
- Fourth Screenshot:** Shows the Solution tab selected. It includes sections for Controls (Equations, Limits, Advanced), Reports (Residuals, Convergence, Definitions, File, Plot), Results (Initialization, Method (Hybrid, Standard), More Settings, Options, Patch, Reset Statistics, Reset DPM, t=0, Initialize), Design (Activities (Autosave, Create, Manage)), and Parametric (Run Calculation, Input Summary, Check Case, Update Dynamic Mesh, No. of Iterations, Calculate).
- Bottom Screenshot:** Shows the Graphics tab selected. It includes sections for Surface (Create, Manage), Graphics (Mesh, Pathlines, LICs, XY Plot, Data Sources, Interpolated Data, Histogram, FFT, Residuals, Profile Data), Plots (Reference Values, Surface Integrals, Zone Motion, Fluxes, Volume Integrals, Projected Areas, Forces, Heat Exchanger), Reports (Sweep Surface, Scene Animation, Turbo Post, Solution Playback), Animation (Discrete Phase, DTRM Graphics, PDF Table, S2S Information), and Model Specific (DTRM Rays, Conduction Manager, PCB Model).

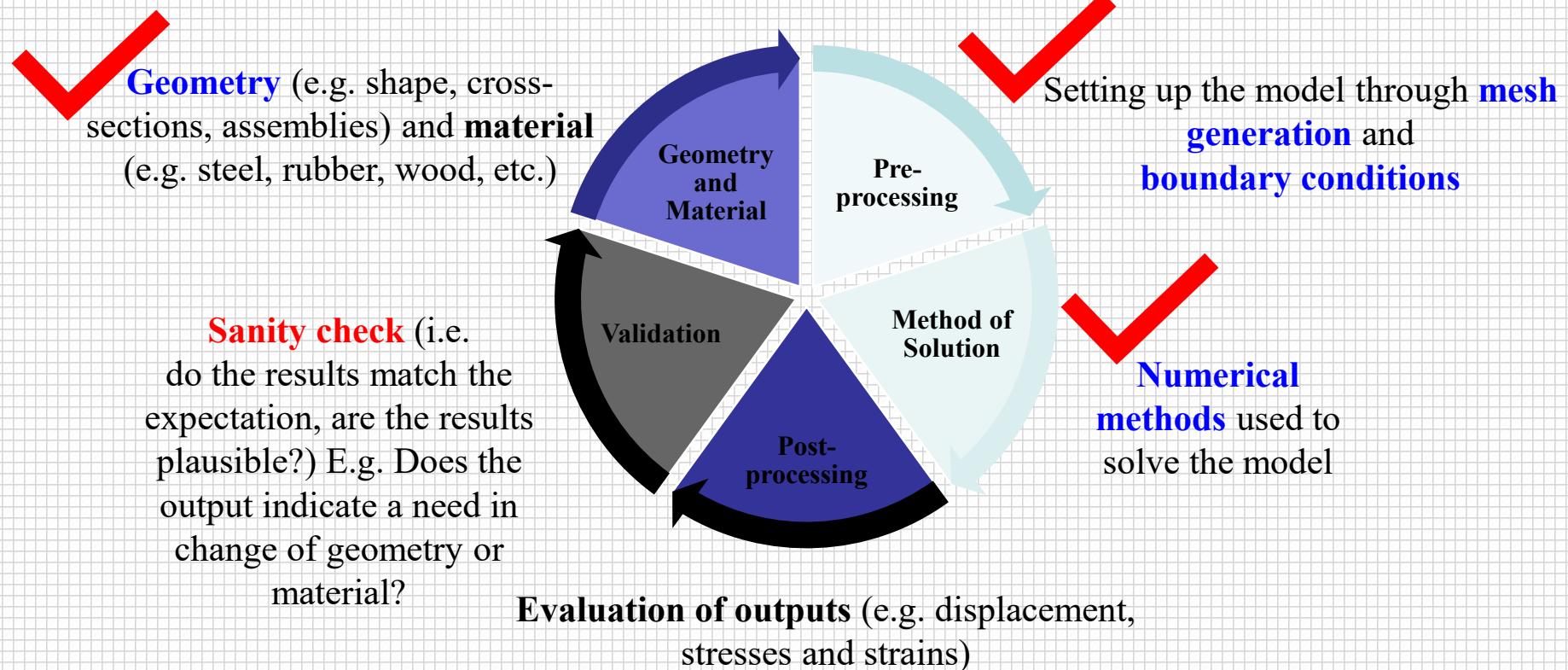
# Solution: Workflow

The image displays four screenshots of software toolbars, likely from a finite element analysis (FEA) or simulation application, illustrating various workflow options:

- Top Toolbar:** Shows tabs for File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, Design, and Parametric. The View tab is active, showing sub-options for Display (Views..., Headlight, Lighting), Graphics (Lights..., Compose..., Colormap..., Material Editor..., Annotate...), and Graphics Toolbars (Mesh Display, View, Copy, Pointer, Visibility, Object Selection/Display). A checkmark icon is present.
- Second Toolbar:** Shows tabs for File, Domain, Physics, User-Defined, Solution, Results, View, and Parallel. The Parallel tab is active, showing sub-options for General (Auto Partition..., Thread Control..., Partition/Load Balance...), Network (Latency, Bandwidth, Connectivity...), Timer (Usage, Reset), and System (CPU Info).
- Third Toolbar:** Shows tabs for File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, and Design. The Design tab is active, showing sub-options for Geometry (Parameterize and Explore..., Observable...), Methods (Methods..., Solver Controls..., Monitors...), Calculate (Calculate...), Gradient-Based (Postprocess Options..., Reporting..., Design Tool...), and Parameter-Based (Gradient-Based Optimizer..., Optimizer...).
- Bottom Toolbar:** Shows tabs for File, Domain, Physics, User-Defined, Solution, Results, View, Parallel, Design, and Parametric. The Parametric tab is active, showing sub-options for Study (Initialize, Duplicate), Design Point (Add Design Point), Import (Import, Export), Simulation Report (Design Point Report Settings, Current Design Point Report, Parametric Report), Update Options (Update Current, Update All), Save Journals (Save Journals), Update Status (Refresh Status), Comparison Plot (Show Summary..., Create New), and Parallel (Update Status, Comparison Plot).

# 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



# Electronics Cooling using Heat Sink (Example 3.8)

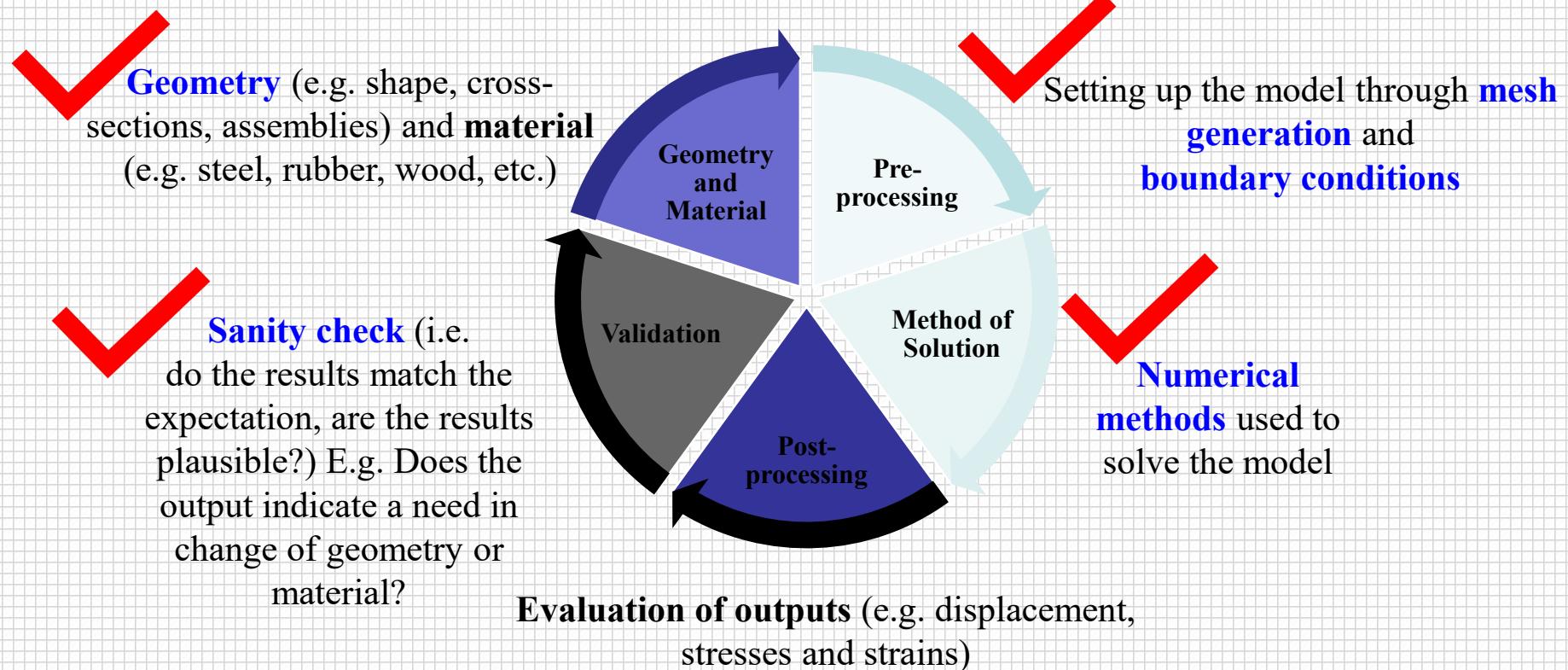
---

## Results:

Configuration	Avg Temp (°C)	Max Temp (°C)	Al Volume (cm³)	% 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



# 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:  $\rho = 8928 \text{ kg/m}^3$ ,  $k = 387.6 \text{ W/m.K}$   
Air:  $\rho = 1.225 \text{ kg/m}^3$ ,  $k = 0.0242 \text{ W/m.K}$   
 $\mu = 1.7894 \times 10^{-5} \text{ kg/m.s}$
- **Materials:** fluid (air) , solid (aluminum)
- **Material property:**  
Aluminum: :  $\rho = 2719 \text{ kg/m}^3$ ,  $k = 202.4 \text{ W/m.K}$   
Air:  $\rho = 1.225 \text{ kg/m}^3$ ,  $k = 0.0242 \text{ W/m.K}$   
 $\mu = 1.7894 \times 10^{-5} \text{ kg/m.s}$