

Project presentation

Heat Transfer Innovation Project

Group-3

To access the COMSOL files go to this link:

<https://drive.google.com/drive/folders/1bybIEUBj3gwaaU2JXCeCgL4BwRutHF6?usp=sharing>

UNDERSTANDING HEAT TRANSFER THROUGH SIMULATIONS

- Heat transfer —by conduction and convection —is a key topic in many engineering fields.
- However, it often feels abstract when taught only through formulas and textbook diagrams.
- Aim of the project: To use COMSOL Multiphysics to simulate heat transfer in 2D and 3D.
- These simulations help in clearly seeing how heat spreads over time and space.
- This approach makes it easier for students to understand heat flow by using interactive visuals instead of just theory.

INNOVATION & EDUCATIONAL IMPACT

- Despite not being formally introduced in coursework, **COMSOL was self-explored** to enhance learning.
- Simulations designed from scratch by students without prior formal training.
- 2D and 3D simulations offer an intuitive understanding of:
 - Temperature evolution over time.
 - Flow effects on thermal gradients.
- Key innovation: This resource can serve as a valuable visual aid for future batches to understand heat transfer more effectively.
- Potential to integrate into lab/demo sessions to complement theoretical courses.

PROCESS

01

Planning the Simulations

- Selected basic 2D and 3D shapes to study conduction and convection.
- Identified real-life-inspired heat transfer scenarios.

02

Learning and Setting Up COMSOL

- Explored COMSOL on our own to understand its interface and tools.
- Created geometries, assigned materials, and defined boundary conditions.

03

Running and Observing Simulations

- Used COMSOL's heat transfer modules to simulate conduction and convection.
- Visualized how temperature changes with time and space in different setups.

Internal Convection in a Pipe (Without Obstruction)

Geometry and boundary conditions

Packages Used:

- **Conjugate Heat Transfer:**

Simulates both conduction in the solid (Copper Pipe) and convection in the fluid (Water) and pipe surface to surrounding, capturing the interaction between the heated Copper Pipe and Water Flow.

- **Laminar Flow:**

Models the Waterflow as smooth and orderly (laminar), appropriate for low to moderate velocities and ensuring accurate heat transfer predictions in this regime.

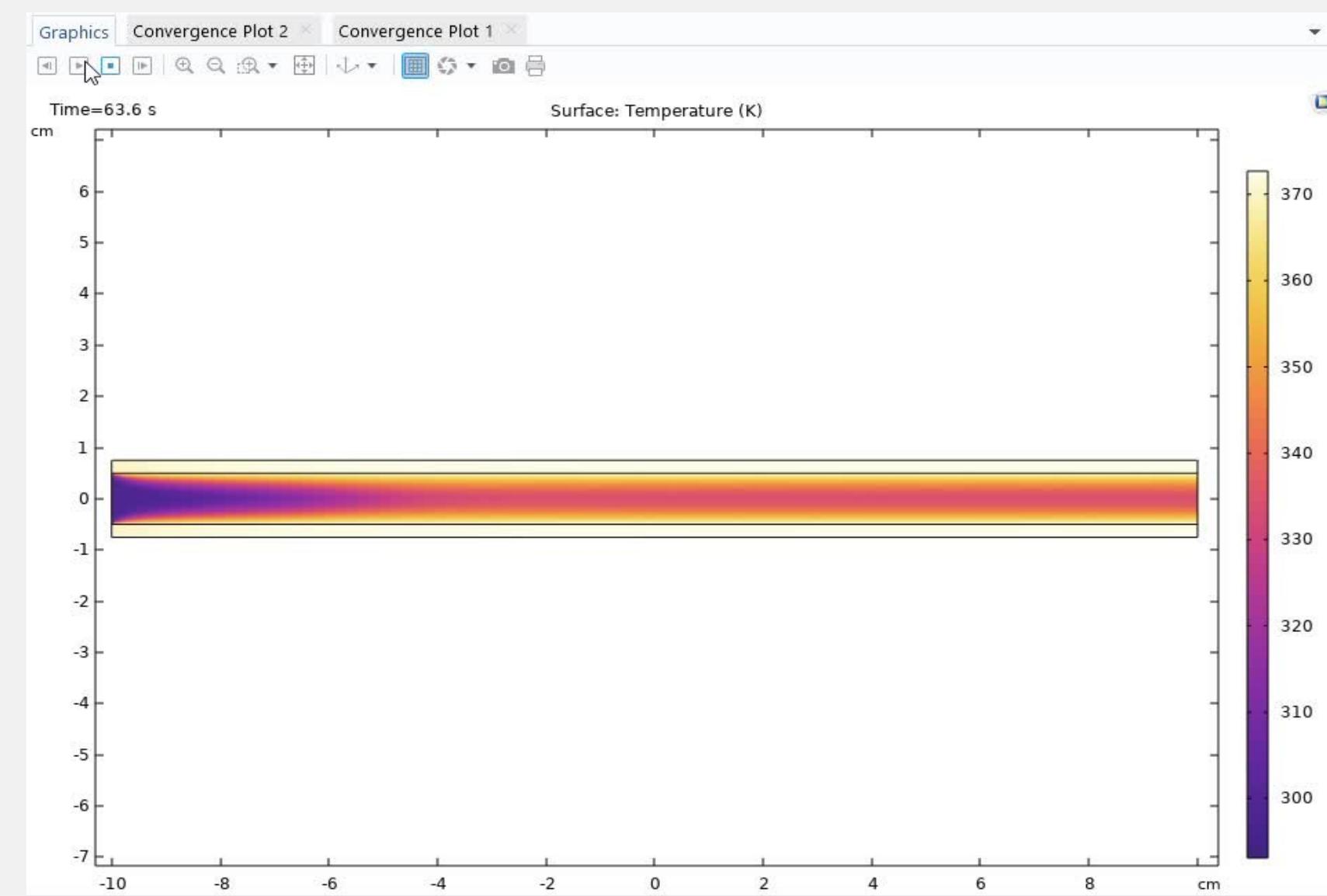
Boundary Conditions:

- Copper Pipe surrounded by steam at 373.15K (This will be the reason of Heat Transfer)
- Left Boundary: Water inlet at specified velocity.
- Right Boundary: Open (outlet) for free Water exit.
- Inner Boundaries: No-slip condition (Water velocity = 0).

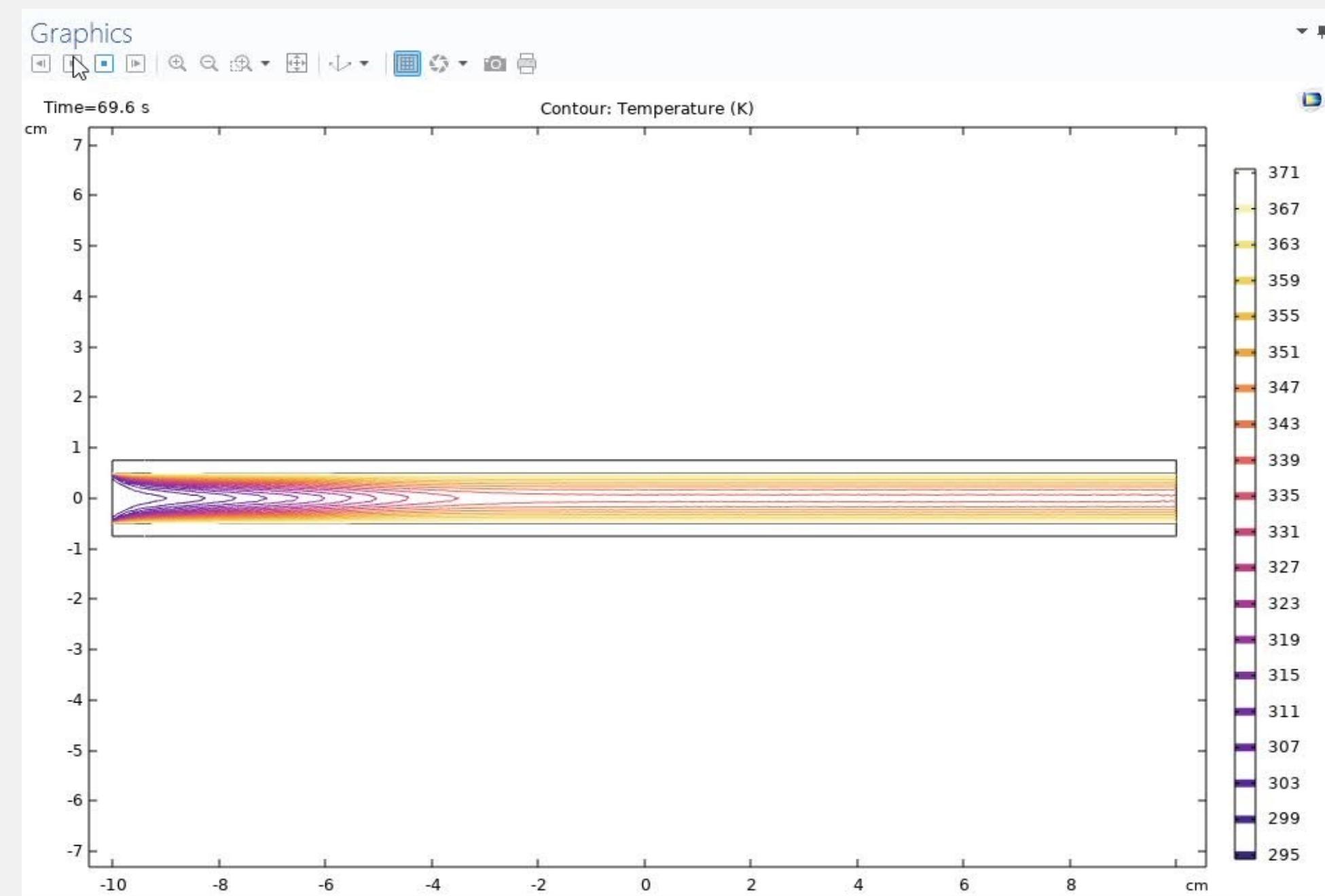
Materials:

- Solid: Copper Pipe.
- Fluid: Water.

Temperature Variation



Contour Plot



Explanation

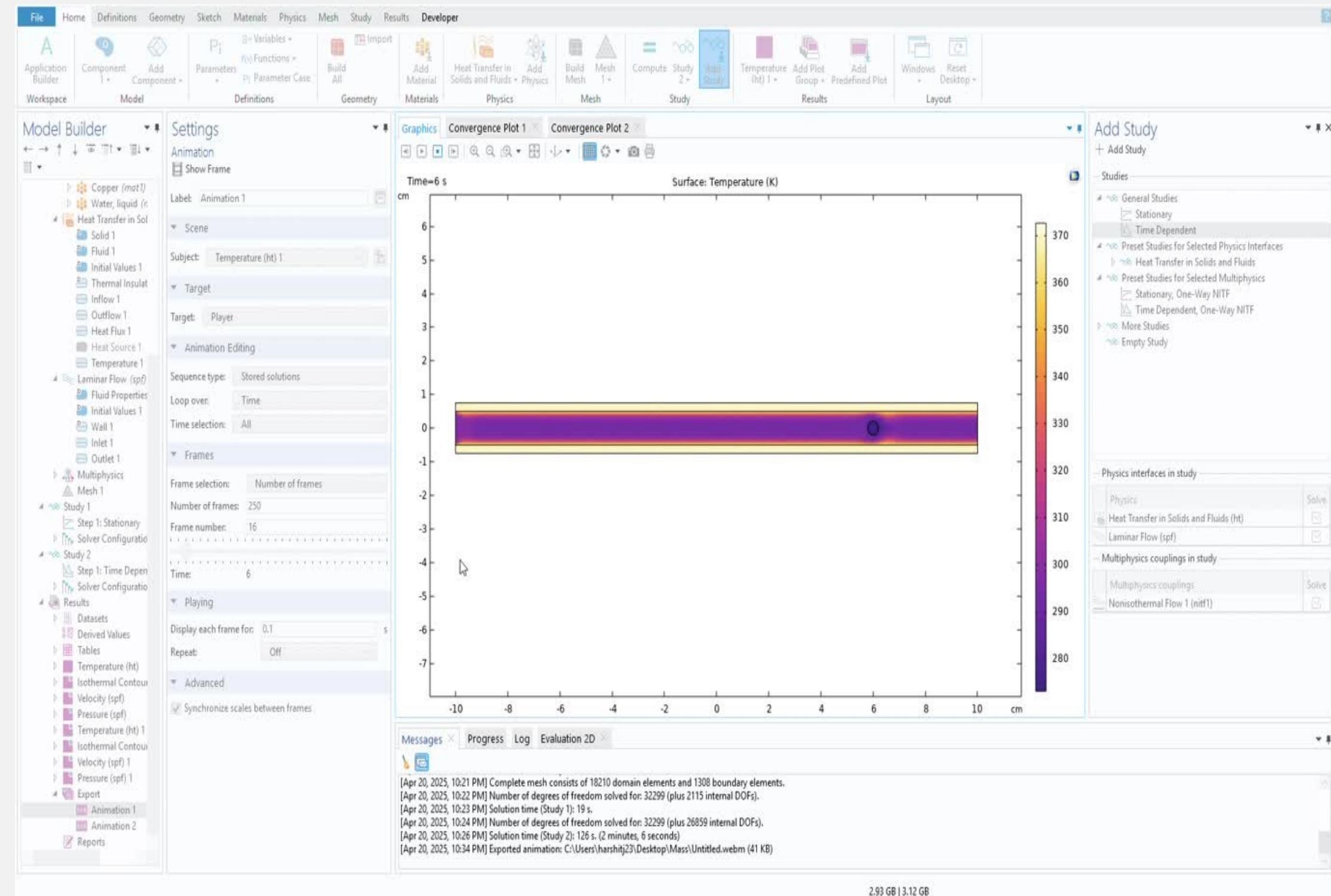
- Heat from surrounding steam (373.15 K) conducts through the copper pipe and convects into the cooler water inside.
- As water flows, a thermal boundary layer forms and thickens downstream, gradually raising the water temperature.
- Due to laminar flow, the heating profile remains smooth and steady.
- Temperature contours show progressive heating, with the outlet water significantly hotter, confirming efficient energy transfer and steady-state behavior.

Internal Convection in a Pipe (With Obstruction)

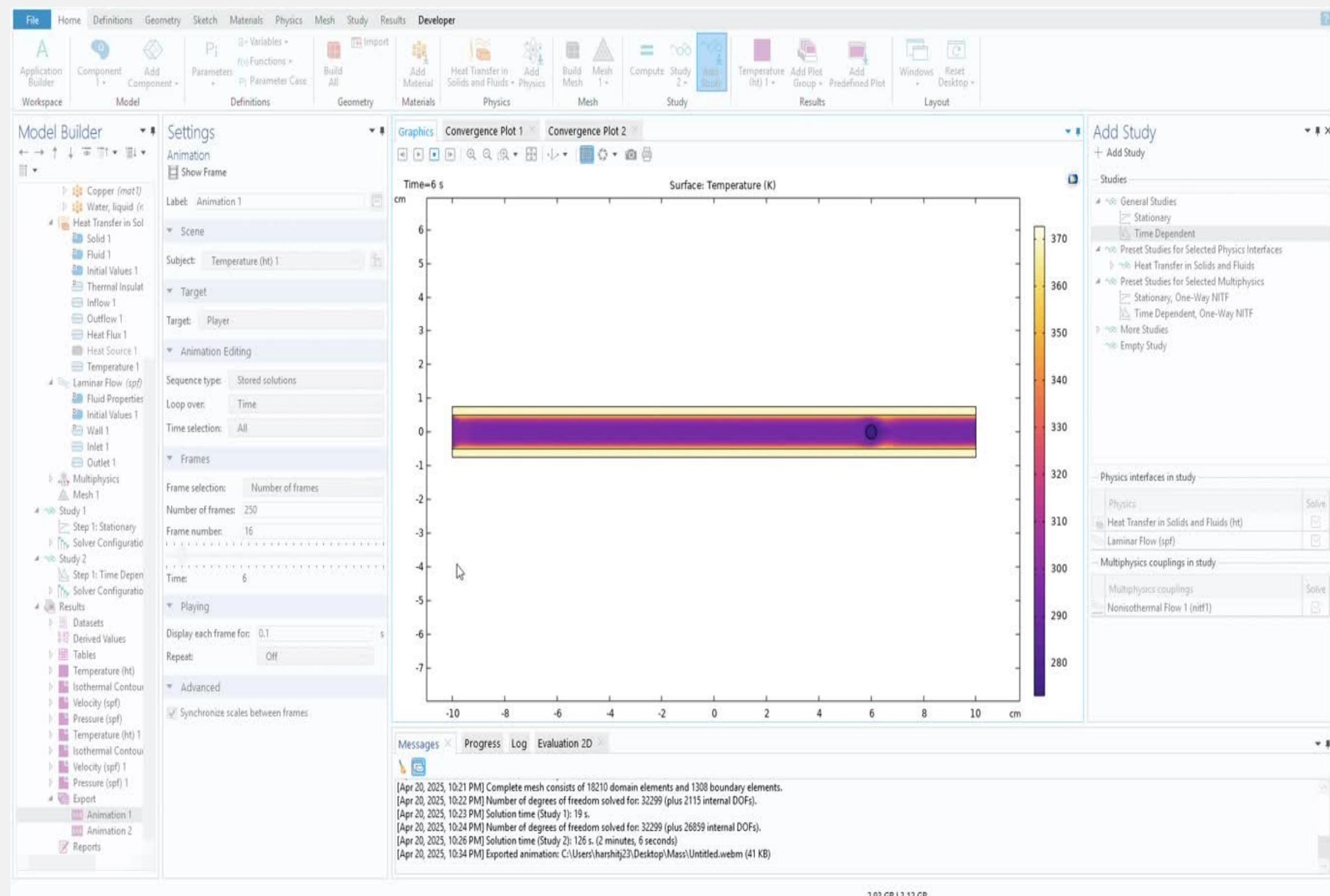
Geometry and Boundary Conditions

- A cylindrical pipe of Outer radius 0.75 cm and inner radius of 0.5 cm
- Used a sphere and a triangle to obstruct the flow
- Fluid (Water) Conditions are same as in the previous one.
- The profile is shown as it develops over time.

Temperature Variation (Sphere)



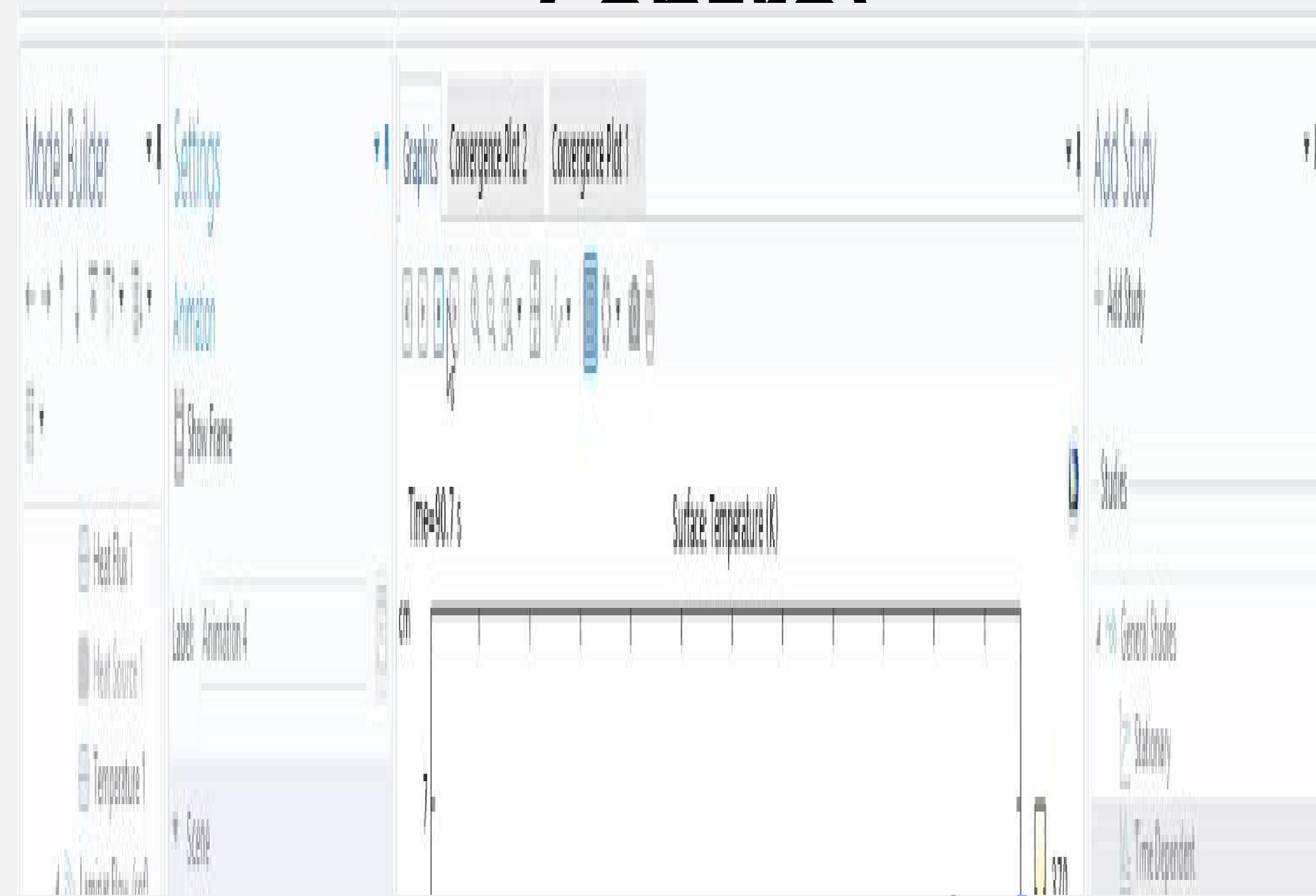
Contour Plots (Sphere)



Temperature Variation (Triangle at Centre)



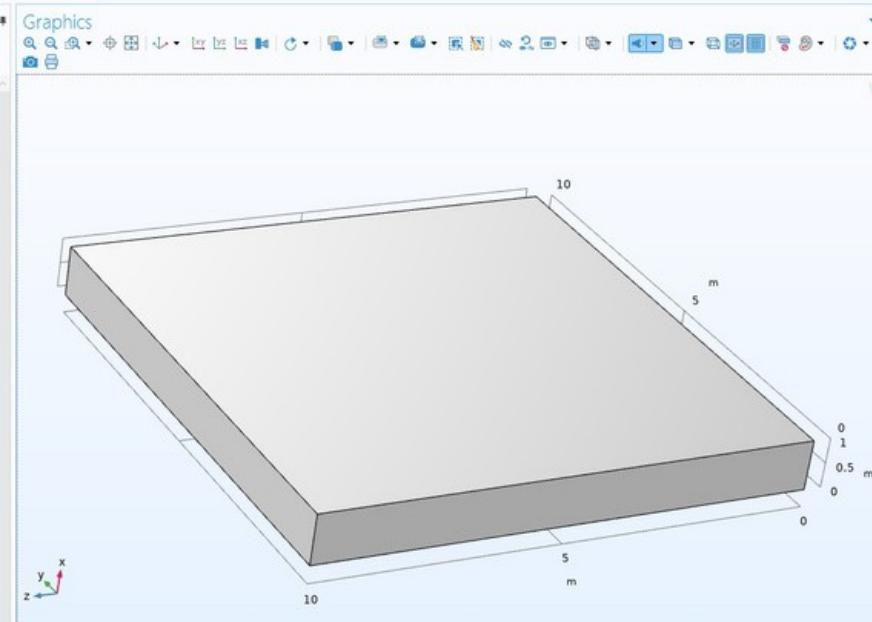
Contour Plots (Triangle At Center)



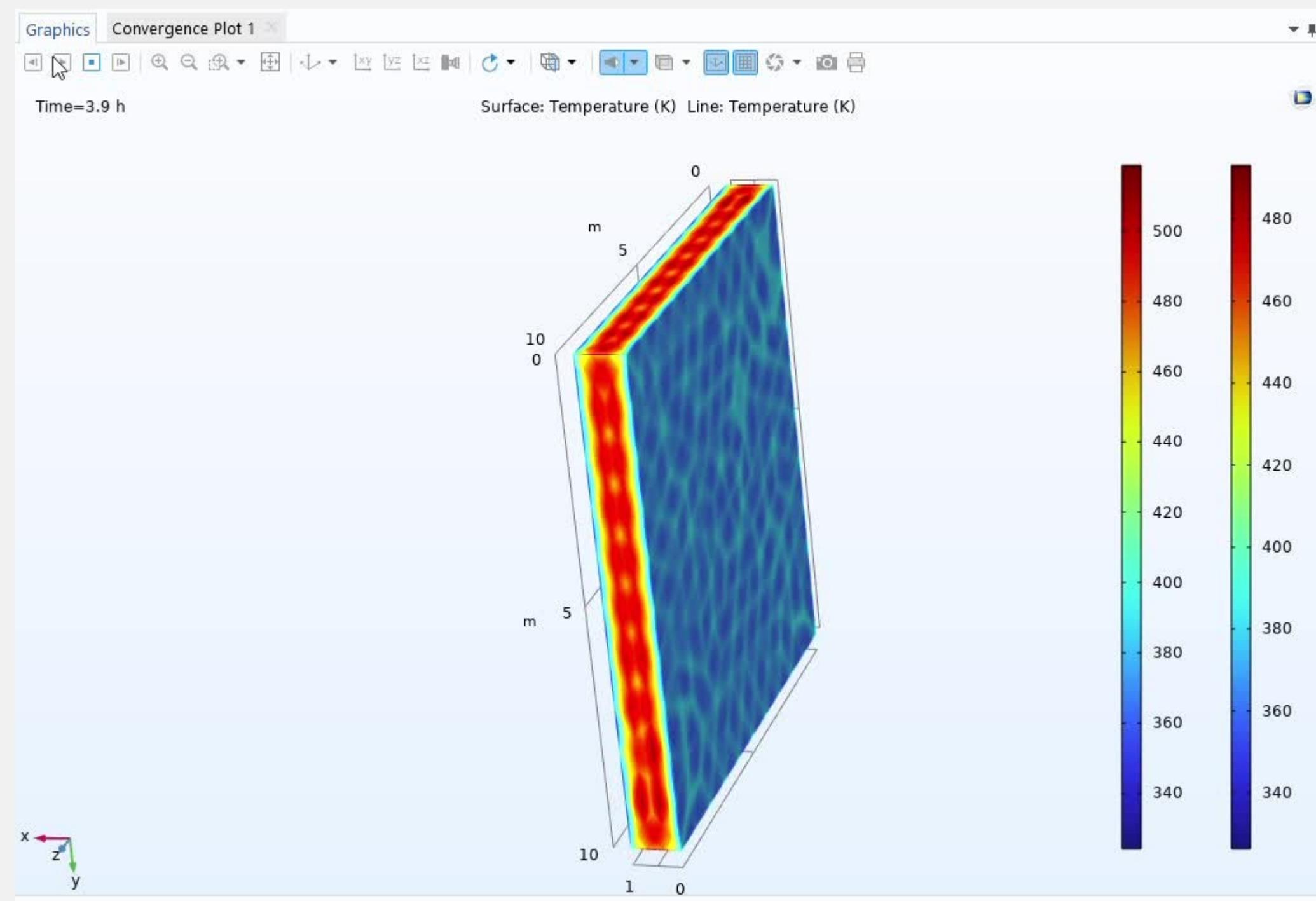
Conduction Through A Plane Wall

Geometry and Boundary Conditions

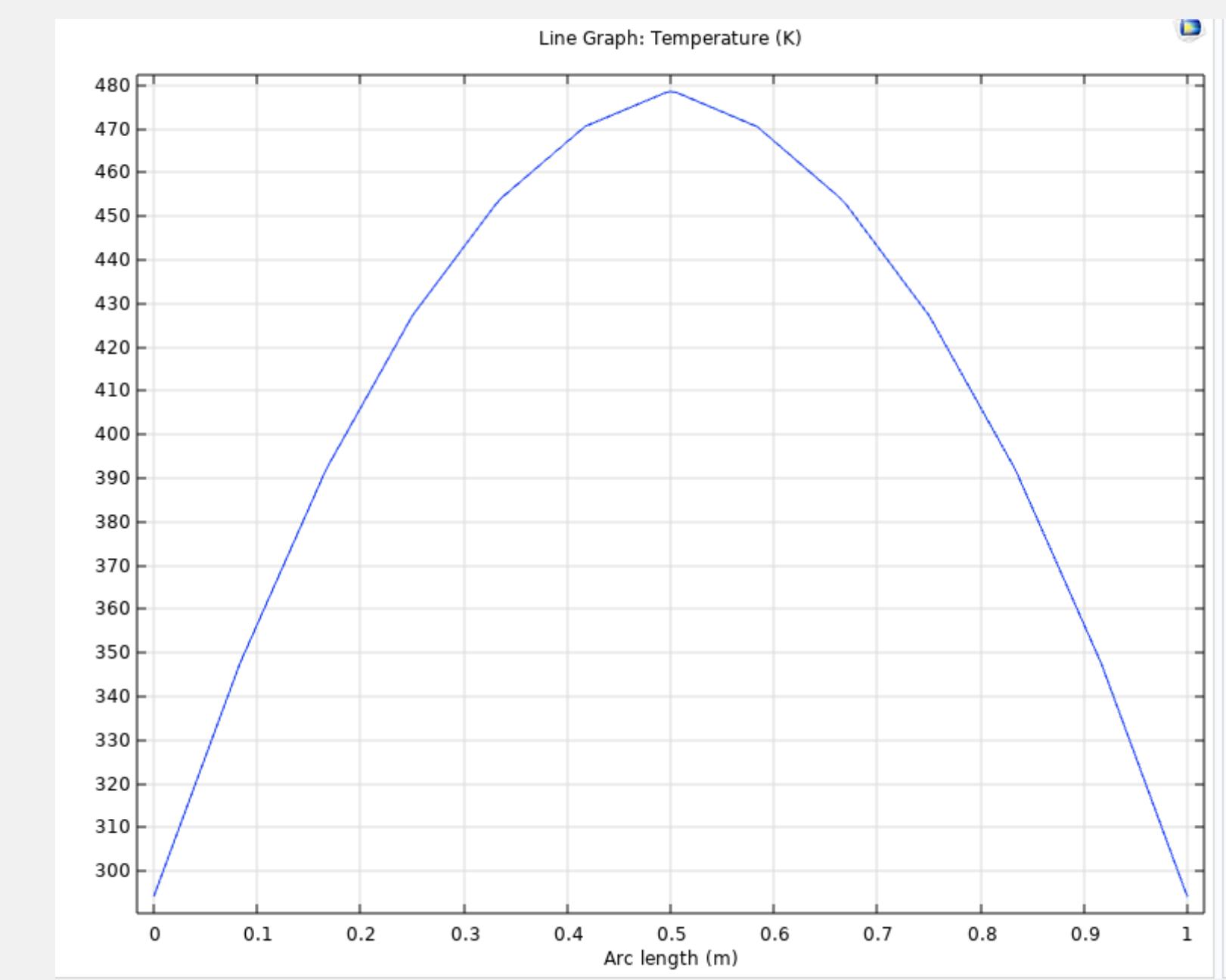
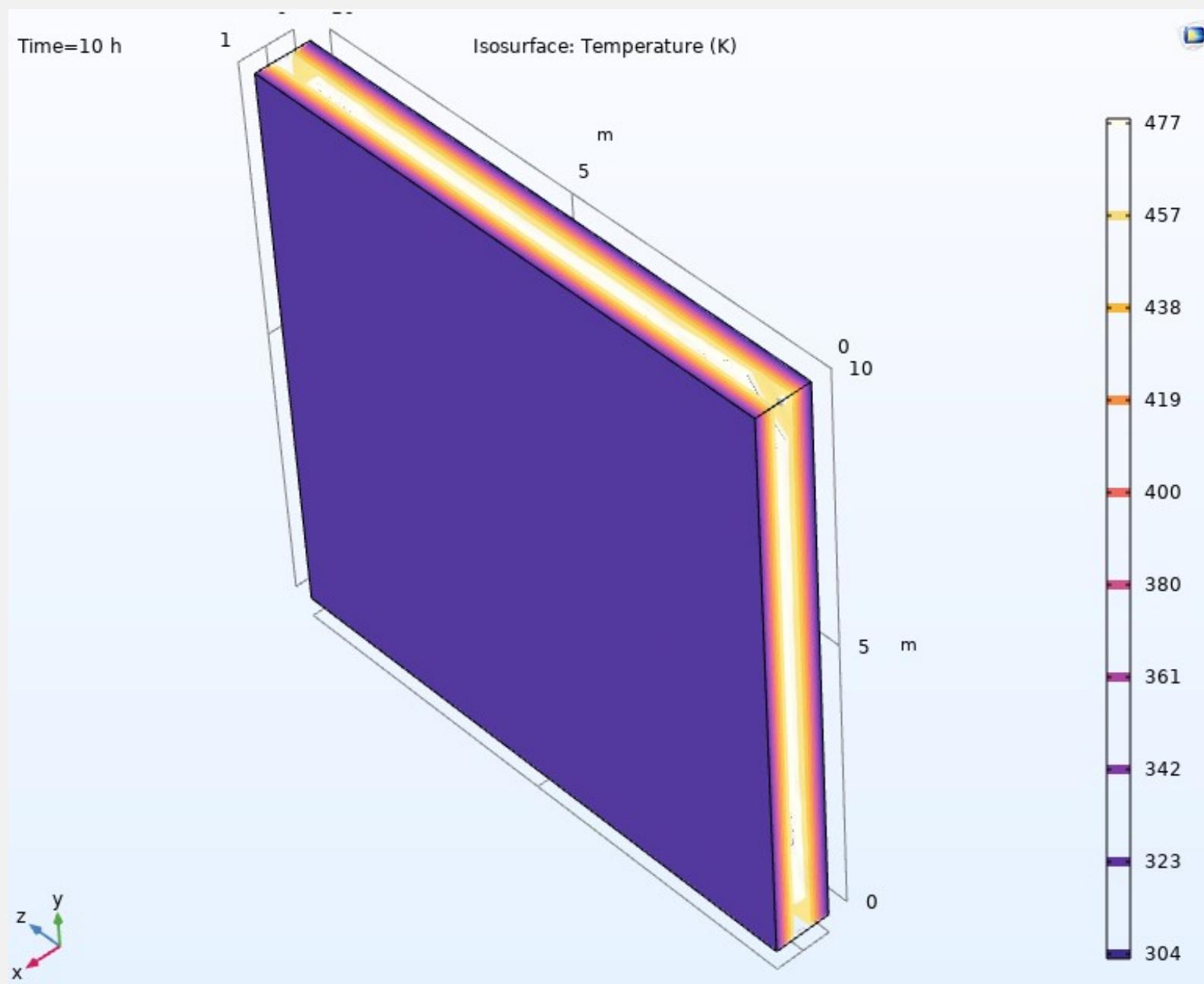
- Used Heat Transfer In Solids Package
- A concrete wall of length=10m, height=10m and width= 1m
- Wall is at 493K intial Temperature.
- We took a constant $h=40\text{W/m}^2\text{K}$
- The profile is shown as it develops over time.



Temperature Profile Development

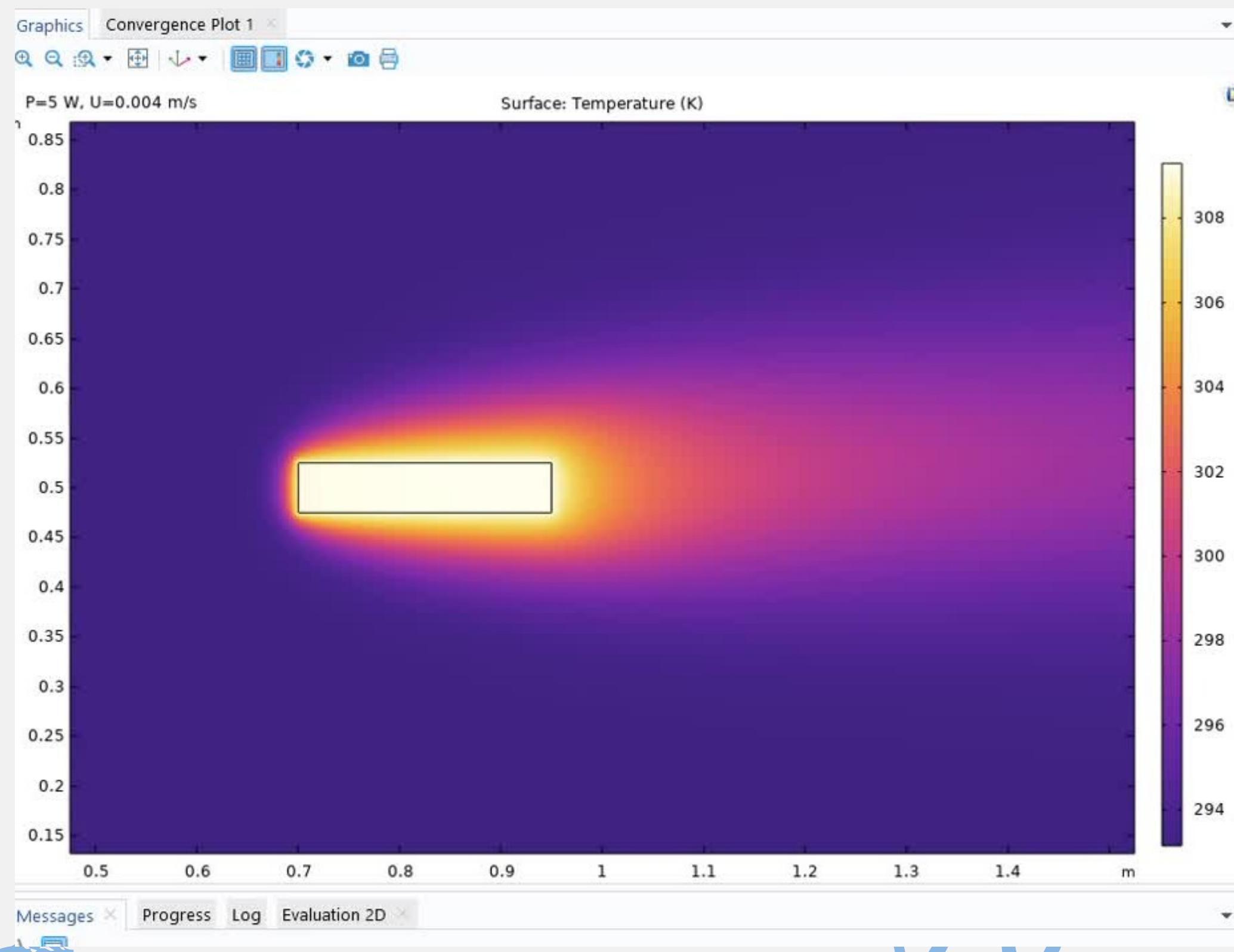


Profiles after 10 hours

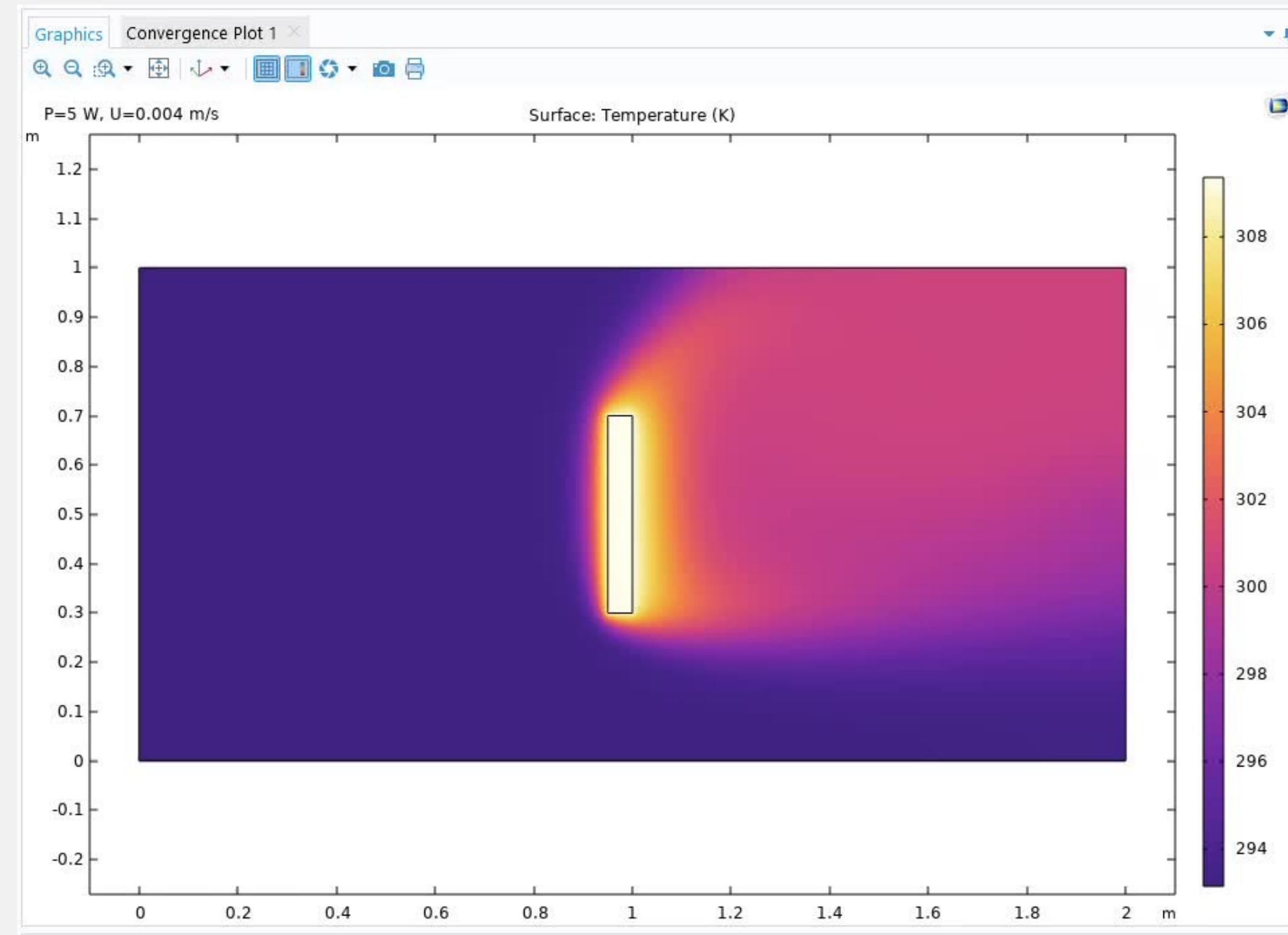


Convective Heat Transfer from a Heated Plate: Influence of Velocity and Orientation

Heat Transfer by Convection at different Air velocities(Horizontal Plate)



Heat Transfer by Convection at different Air velocities(Vertical Plate)



Geometry and boundary conditions

Packages Used:

- **Conjugate Heat Transfer:**

Simulates both conduction in the solid (structured steel) and convection in the fluid (air), capturing the interaction between the heated steel rectangle and the surrounding airflow.

- **Laminar Flow:**

Models the airflow as smooth and orderly (laminar), appropriate for low to moderate velocities and ensuring accurate heat transfer predictions in this regime.

Boundary Conditions:

- **Rectangle:** Constant heat source of 5 units.
- **Left Boundary:** Air inlet at specified velocity.
- **Right & Top Boundaries:** Open (outlet) for free air exit.
- **Bottom Boundary:** No-slip condition (air velocity = 0).

Materials:

- **Solid:** Structured steel.
- **Fluid:** Air.

Insights

Velocity and Boundary Layer Behavior:

- **Increasing Air Velocity:** As air velocity increases, the boundary layer becomes thinner, reducing the insulating effect and allowing more efficient heat transfer from the plate to the air. However, in the case of the vertical plate, there is not a well-defined boundary layer or "blanket" like in horizontal configurations.

Flow Dynamics on Vertical Plate (Turbulent Flow):

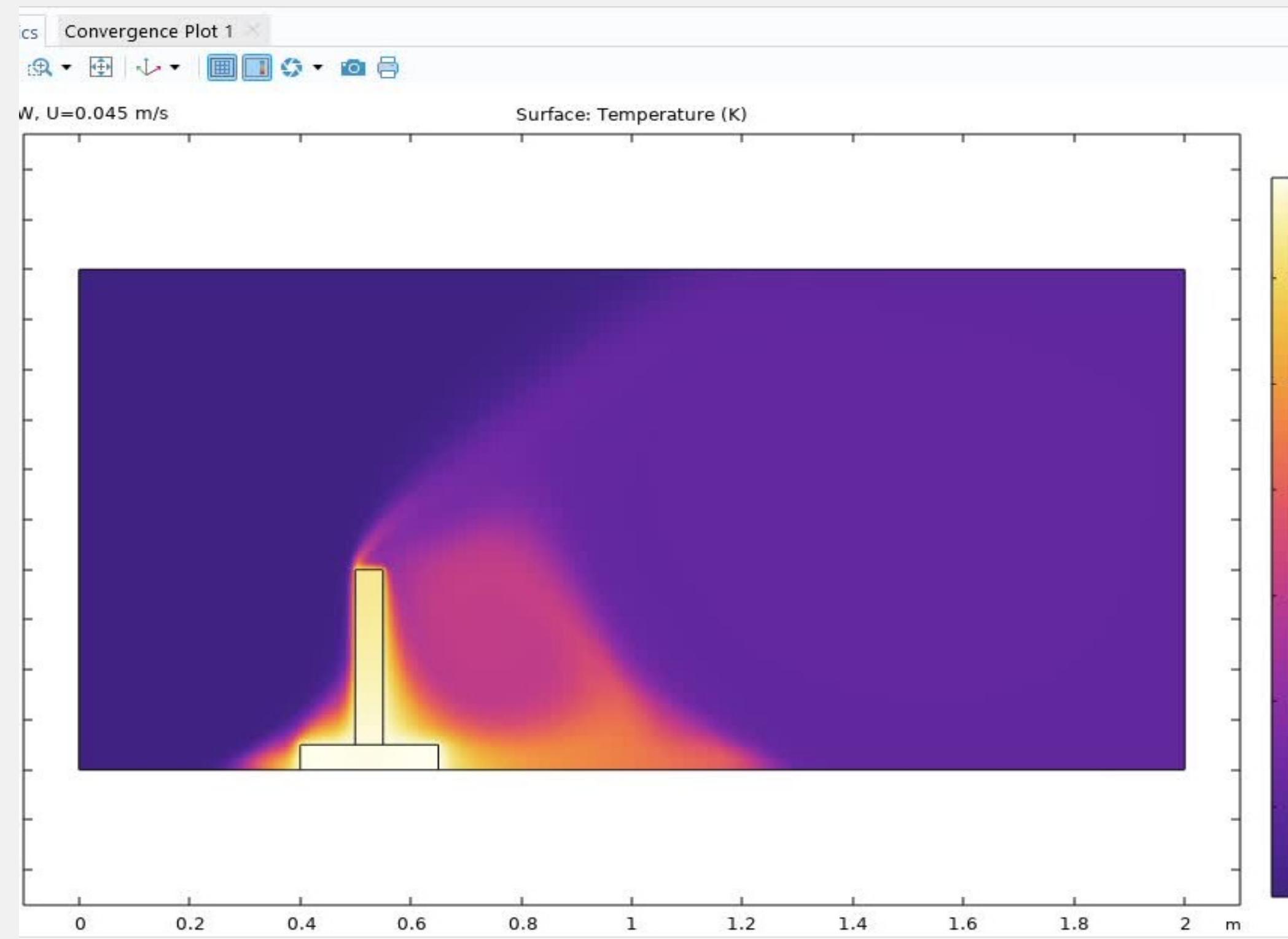
- **No Clear Boundary Layer:** For vertical plates, the airflow doesn't form a typical boundary layer. Rather, after the air passes over the plate, the flow becomes turbulent almost immediately. This turbulence accelerates the heat dissipation process, as the mixing of hot and cold air increases, improving thermal exchange with the plate.

Impact of Air Density on Vertical Plate Cooling:

- **Hot Air Rising:** After the air heats up and becomes less dense, it naturally rises. This results in the direction of the airflow changing after crossing the plate. The hot air tends to move upward, which creates a vertical convection current. This helps carry the heat away from the surface more efficiently, aiding in the cooling of the plate.

Convection Through Finned Surfaces(2D)

Heat Transfer by Convection at different Air velocities(Rectangular Fin)



Geometry and boundary conditions

Packages Used:

- **Conjugate Heat Transfer:**

Simulates both conduction in the solid (structured steel) and convection in the fluid (air), capturing the interaction between the heated steel rectangle and the surrounding airflow.

- **Laminar Flow:**

Models the airflow as smooth and orderly (laminar), appropriate for low to moderate velocities and ensuring accurate heat transfer predictions in this regime.

Boundary Conditions:

- Rectangle 1:(Base) Constant heat source of 5 units.
- Rectangle 2:(Fin) Connected to Base and of same material
- Left Boundary: Air inlet at specified velocity.
- Right & Top Boundaries: Open (outlet) for free air exit.
- Bottom Boundary: No-slip condition (air velocity = 0).

Materials:

- Solid: Structured steel.
- Fluid: Air.

Insights

Fins as Passive Heat Extractors

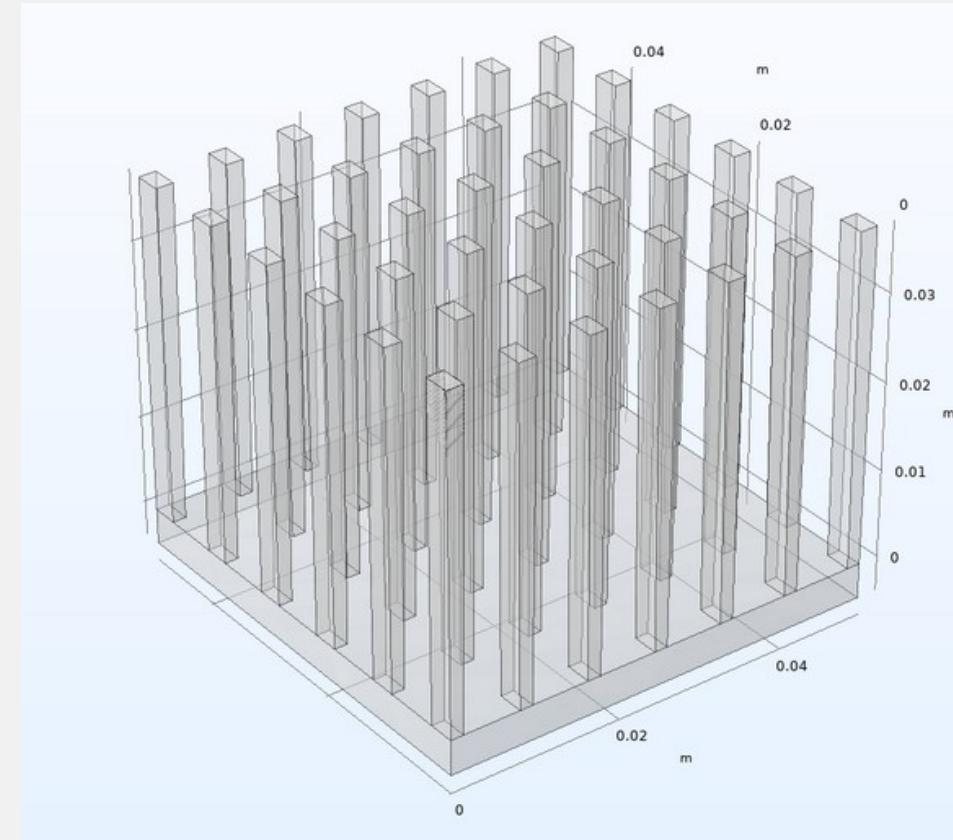
- The vertical protrusion (fin) behaves like a traditional cooling fin:
 - It conducts heat from the horizontal base upward.
 - It provides increased surface area for convective heat loss.
- This is particularly effective because, as previously observed:
 - Vertical surfaces lose heat more efficiently via convection than horizontal ones.
 - The fin structure essentially leverages natural convection and geometry to enhance cooling passively.

Flow Regime and Heat Accumulation Behind the Fin

- At low air velocities, the flow stays laminar, and this creates a stagnant region behind the fin (a recirculation or wake zone).
 - This causes localized heat accumulation, as the air doesn't move fast enough to carry the heat away.
 - We can visualize this as a "thermal shadow" behind the fin.
- As air velocity increases, the flow starts to:
 - Become transitional or turbulent downstream.
 - Which reduces or eliminates the stagnant hot zone behind the fin.
 - The heat is dragged along with it, improving the overall cooling performance.

Convection Through Finned Surfaces(3D)

Geometry of Fins

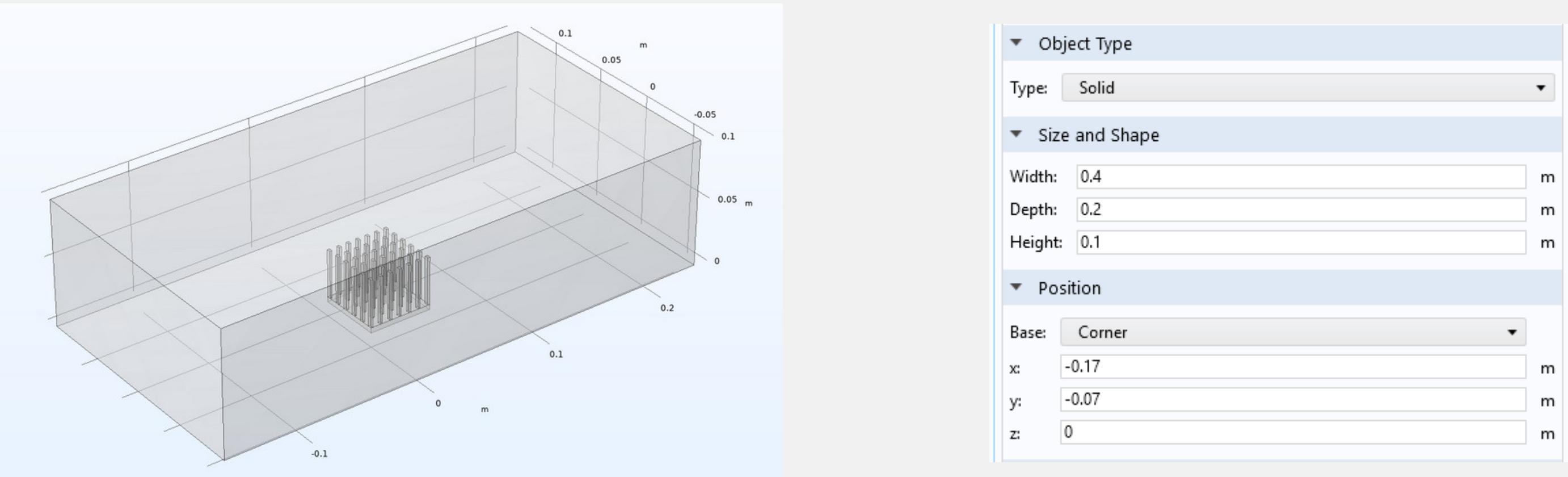


Built-in Comsol Model

Used Aluminium as Fin Material

Name	Expression	Value	Description
fin_type	1	1	Fin type: 1-Pin fins, 2-Dissimilar bor...
X_base	50[mm]	0.05 m	Base dimension in x direction
Y_base	50[mm]	0.05 m	Base dimension in y direction
Z_base	4[mm]	0.004 m	Base dimension in z direction
n_fins_x	7	7	Amount of fins in x direction
n_fins_y	6	6	Amount of fins in y direction
X_fins_bottom	1.5[mm]	0.0015 m	Fin dimension in x direction, bottom
X_fins_top	2[mm]	0.002 m	Fin dimension in x direction, top
Y_fins_bottom	3[mm]	0.003 m	Fin dimension in y direction, bottom
Y_fins_top	3[mm]	0.003 m	Fin dimension in y direction, top
Y_fins_bottom_2	6[mm]	0.006 m	Fin dimension in y direction, botto...
Y_fins_top_2	2[mm]	0.002 m	Fin dimension in y direction, top, se...
Z_fins	38[mm]	0.038 m	Fin height
o_x	0[mm]	0 m	Border offset in x direction
o_y	0[mm]	0 m	Border offset in y direction
fillet_top	0	0	Fillet on top: 1-Enabled, 0-Disabled
fillet_bottom	0	0	Fillet on bottom: 1-Enabled, 0-Disab...
notch	0	0	Notch: 1-Enabled, 0-Disabled
notch_width	5[mm]	0.005 m	Notch width
notch_height	5[mm]	0.005 m	Notch height
step	0	0	Step: 1-Enabled, 0-Disabled
step_height	3[mm]	0.003 m	Step height
step_width	1	1	Number of filled gaps from middle t...
chamfer	0	0	Chamfer: 1-Enabled, 0-Disabled
chamfer_width	5[mm]	0.005 m	Chamfer width
chamfer_height	5[mm]	0.005 m	Chamfer height
shell	0	0	0=3D fins, 1=shell fins

Geometry of Domain

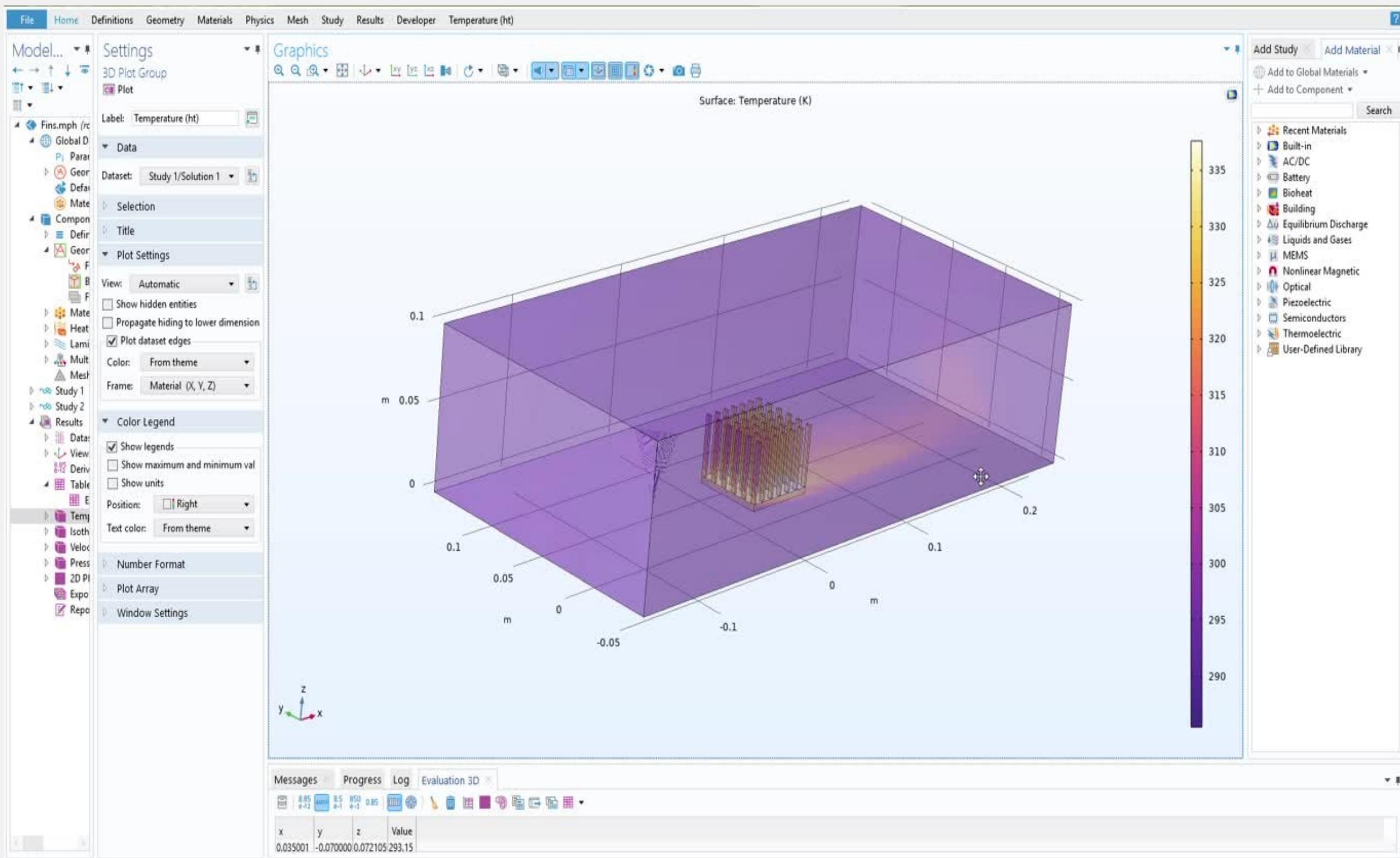


Used Air as Our Domain

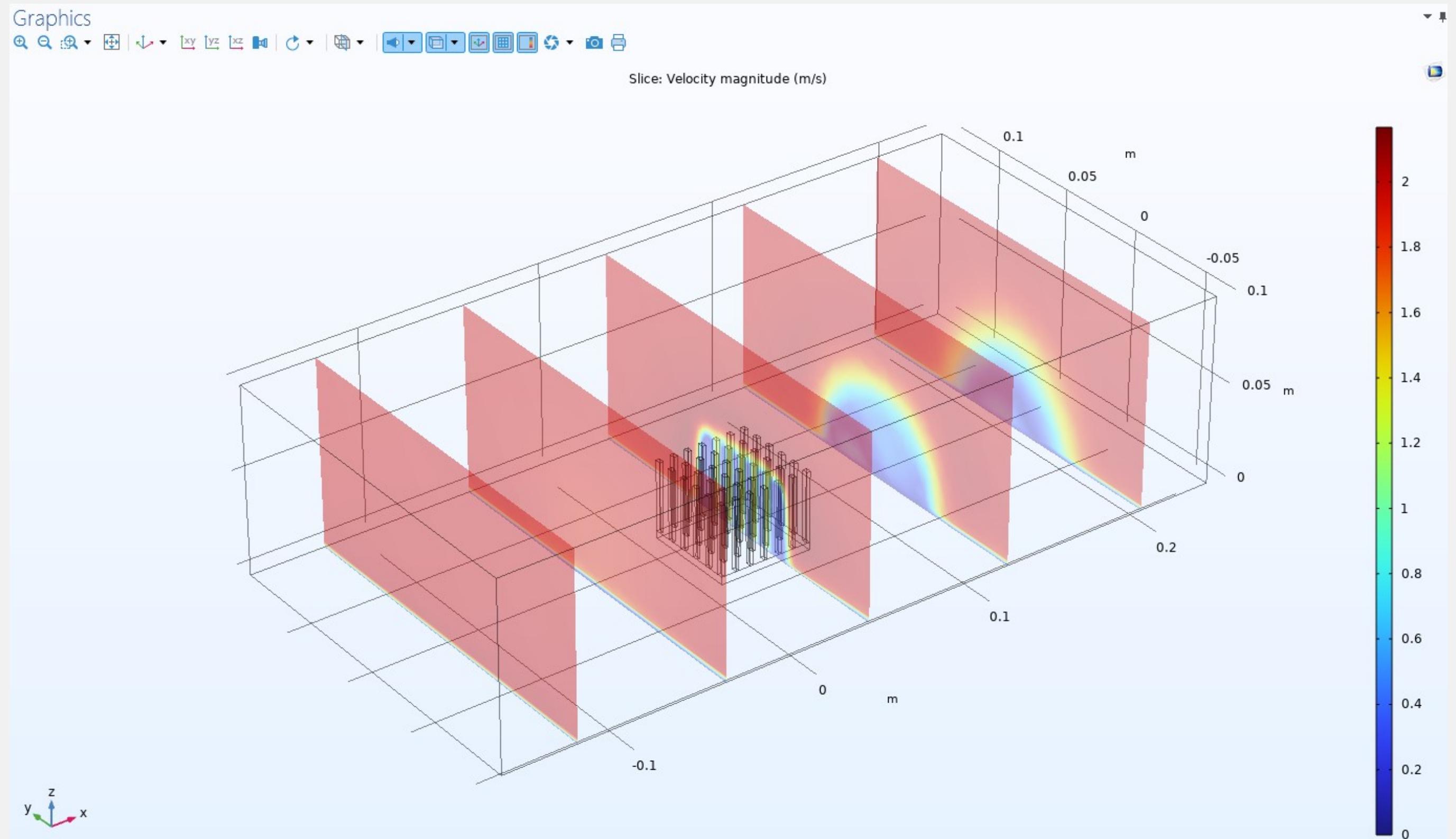
Boundary Conditions

- Used Conjugate Heat Transfer Laminar Flow Package.
- Heat is generated at the bottom of the plate. A constant heat flux of 40W is provided.
- Air is entering at room temperature (293 K) from the front face at a velocity of 2 m/s.
- Calculated results are at STEADY STATE.

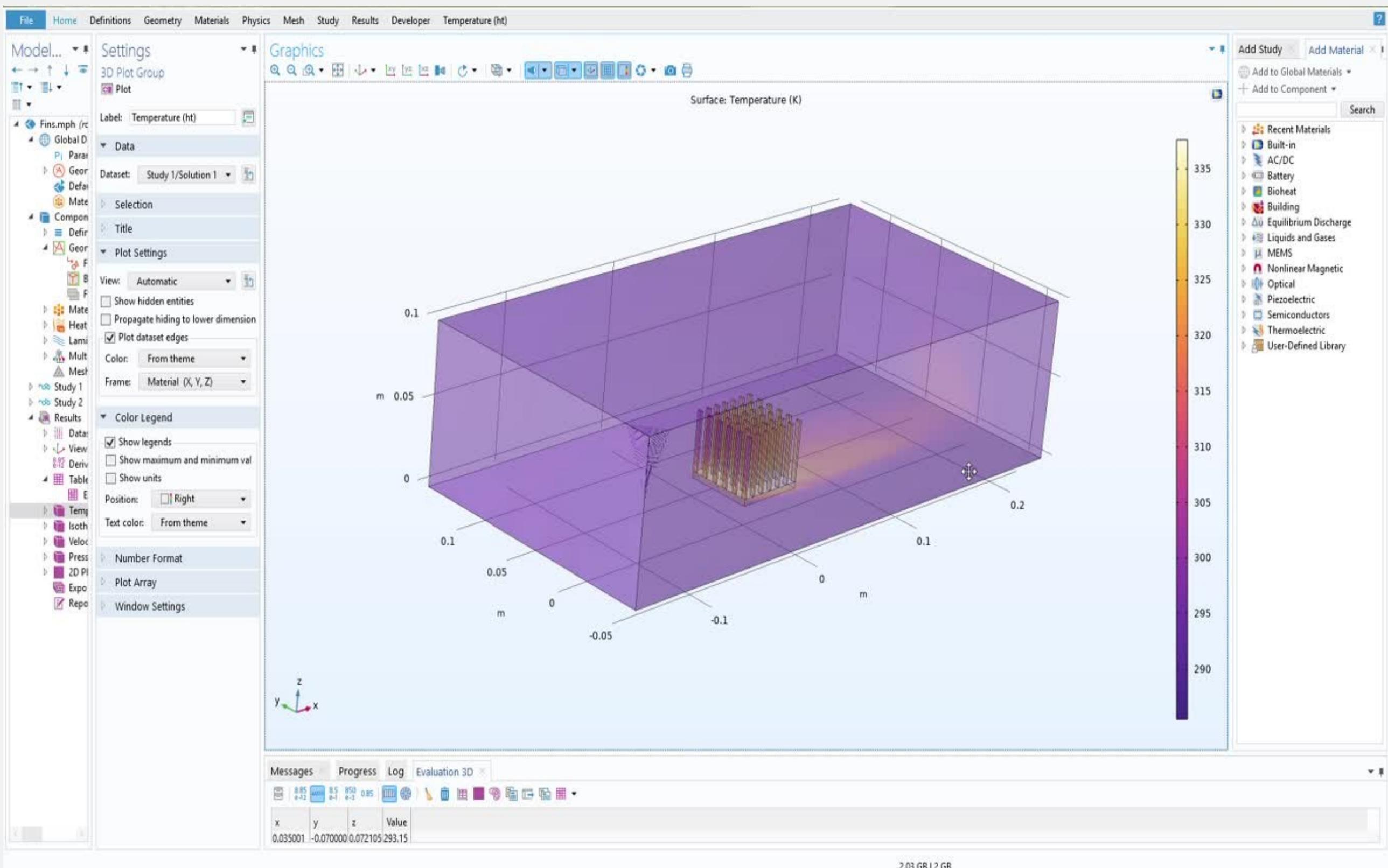
Temperature Profile



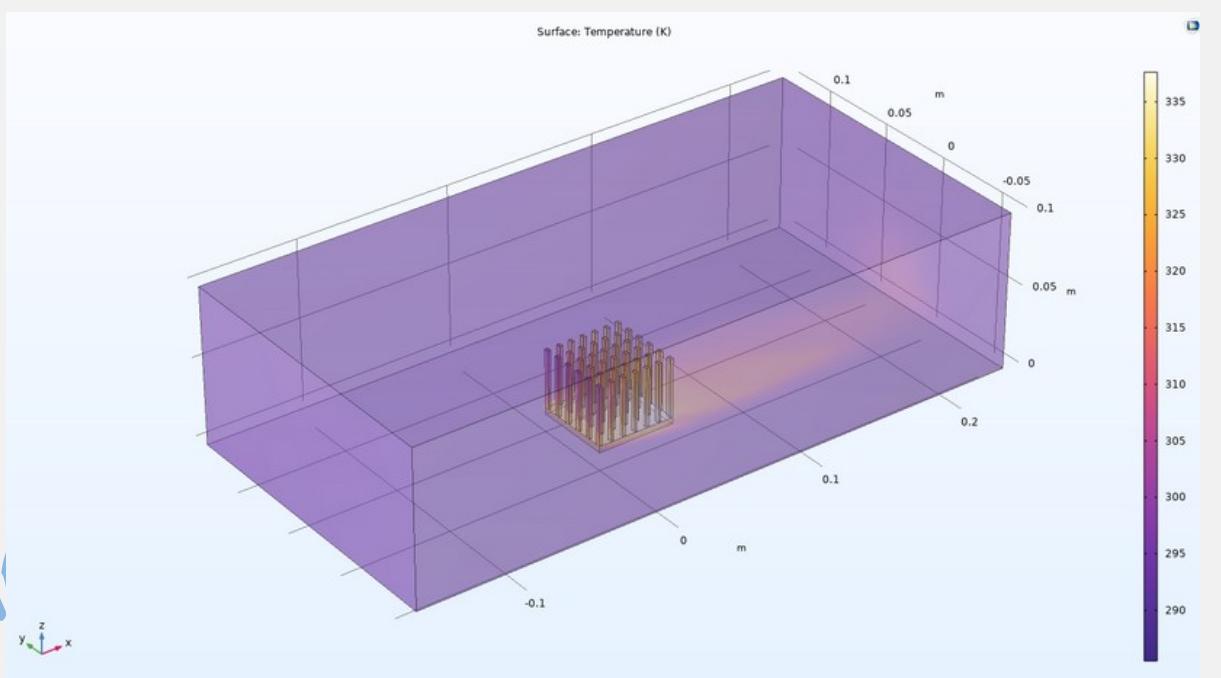
Velocity Profile



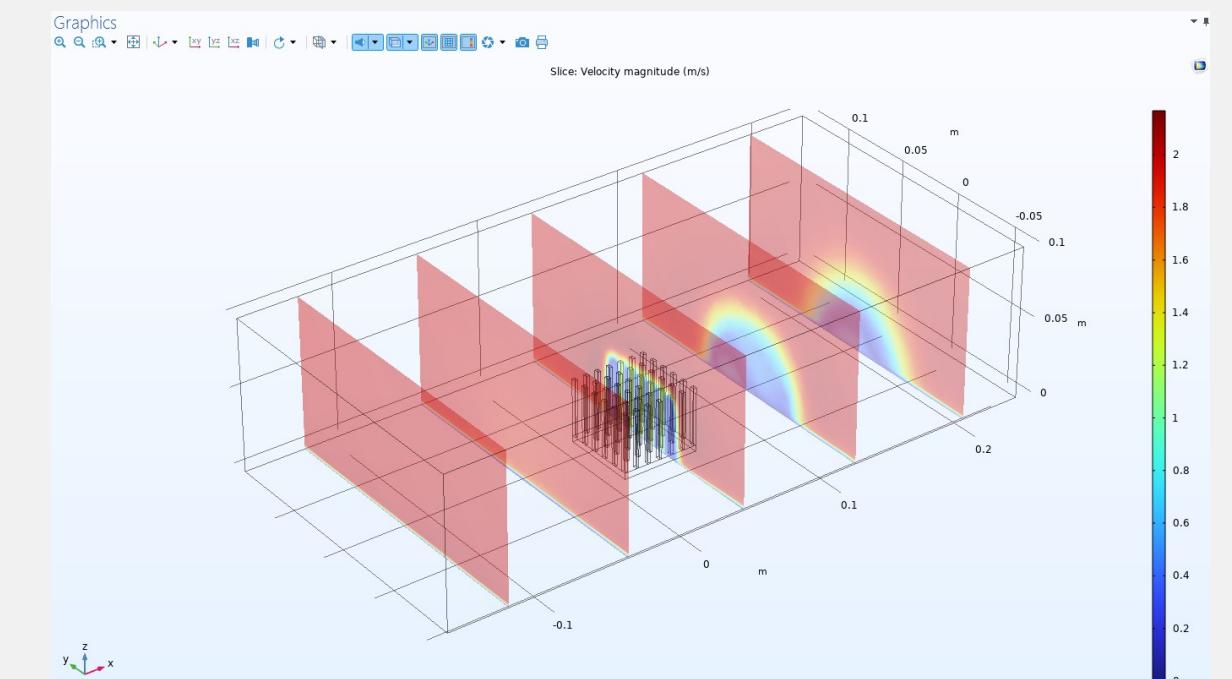
Pressure Variations



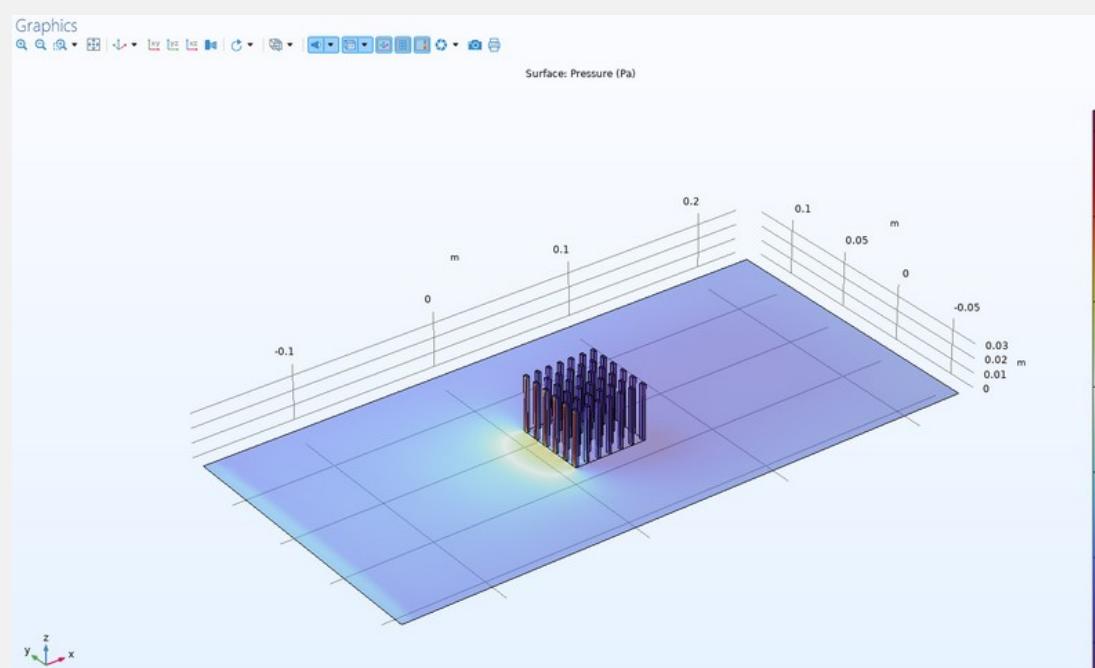
All Profiles



Temperature

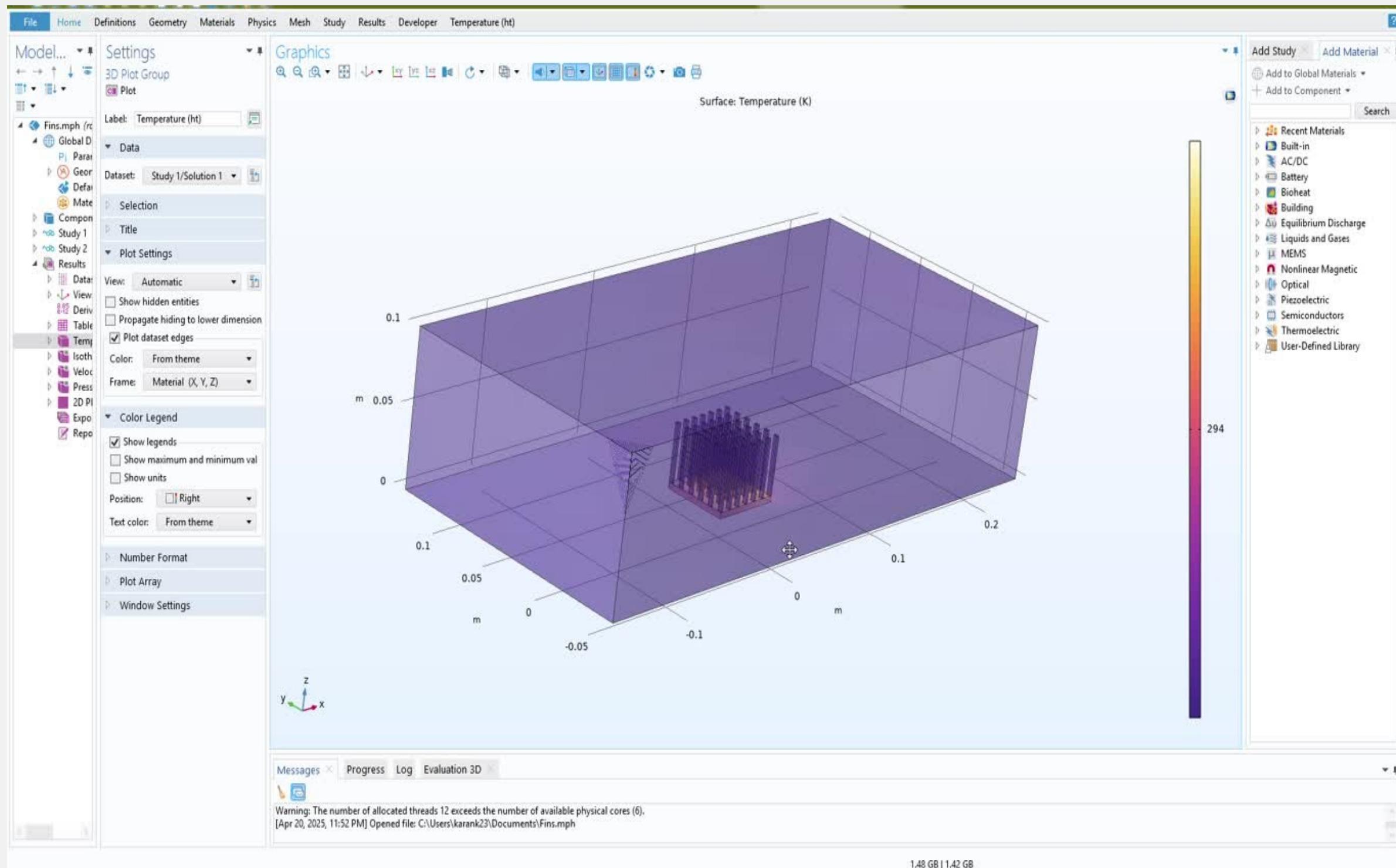


Velocity



Pressure

Water As Domain



Comparison

Almost all the fins were cooled to room temperature when water was used as our fluid domain.

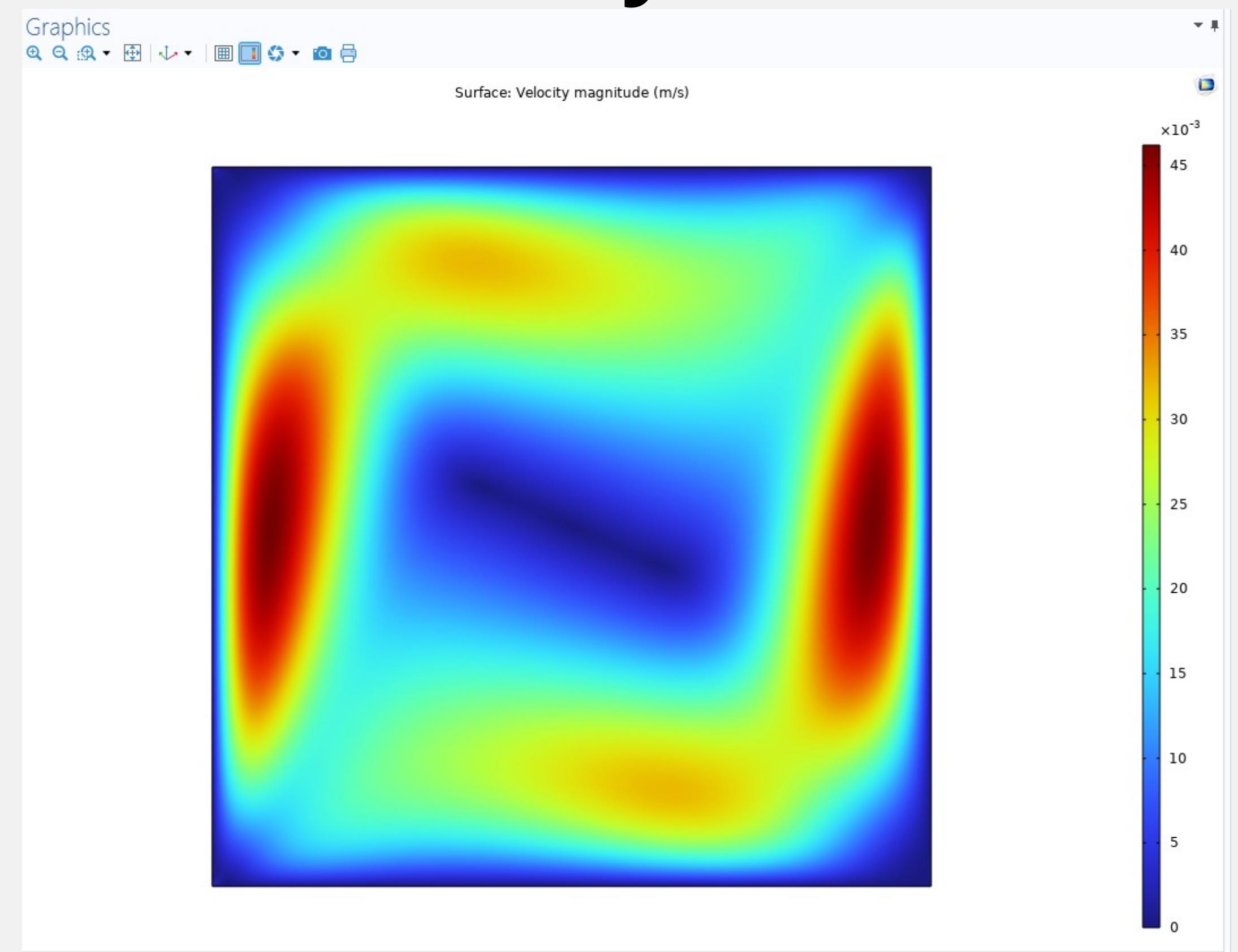
It conveys the effectiveness of water as a coolant in comparison to air.

Natural Convection

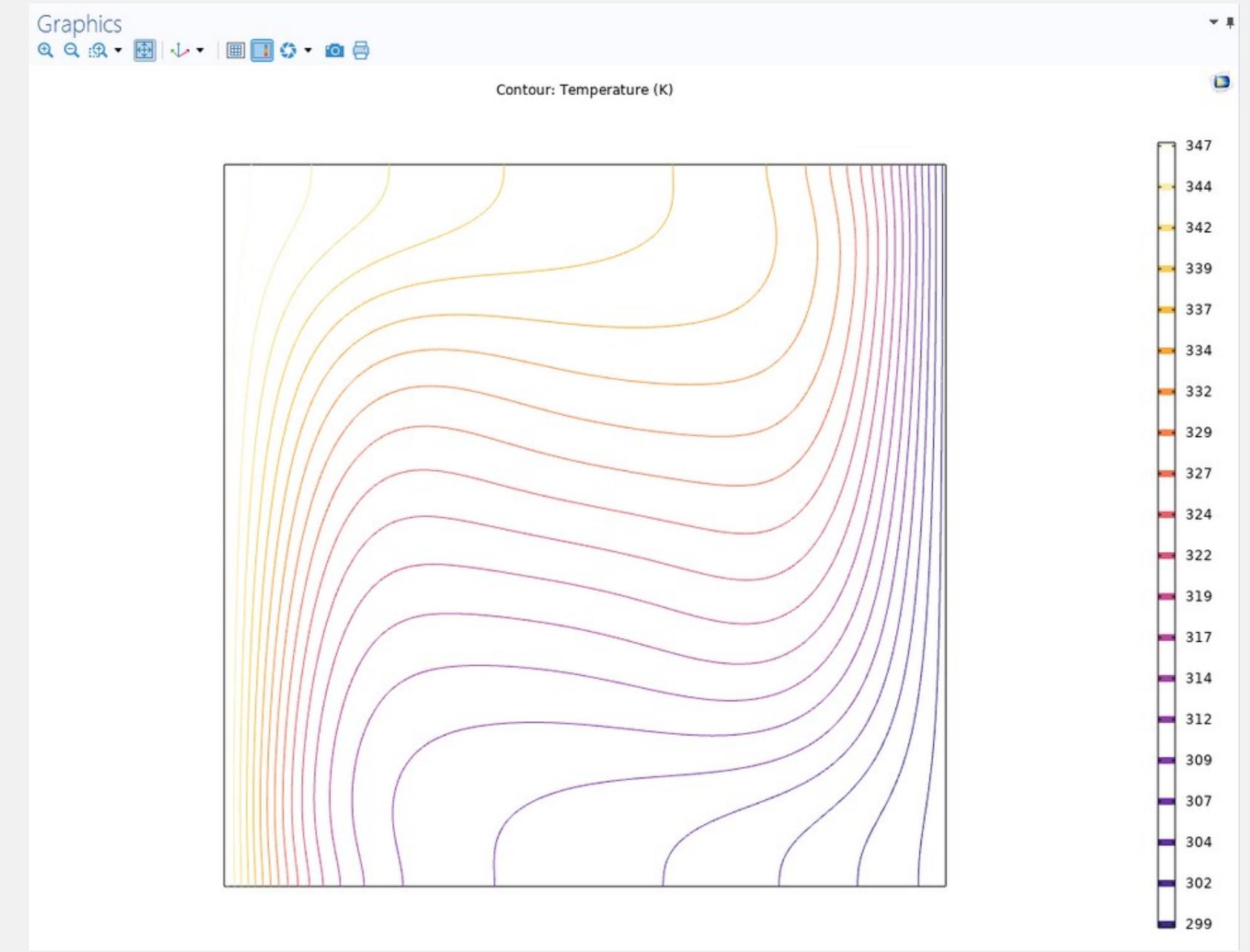
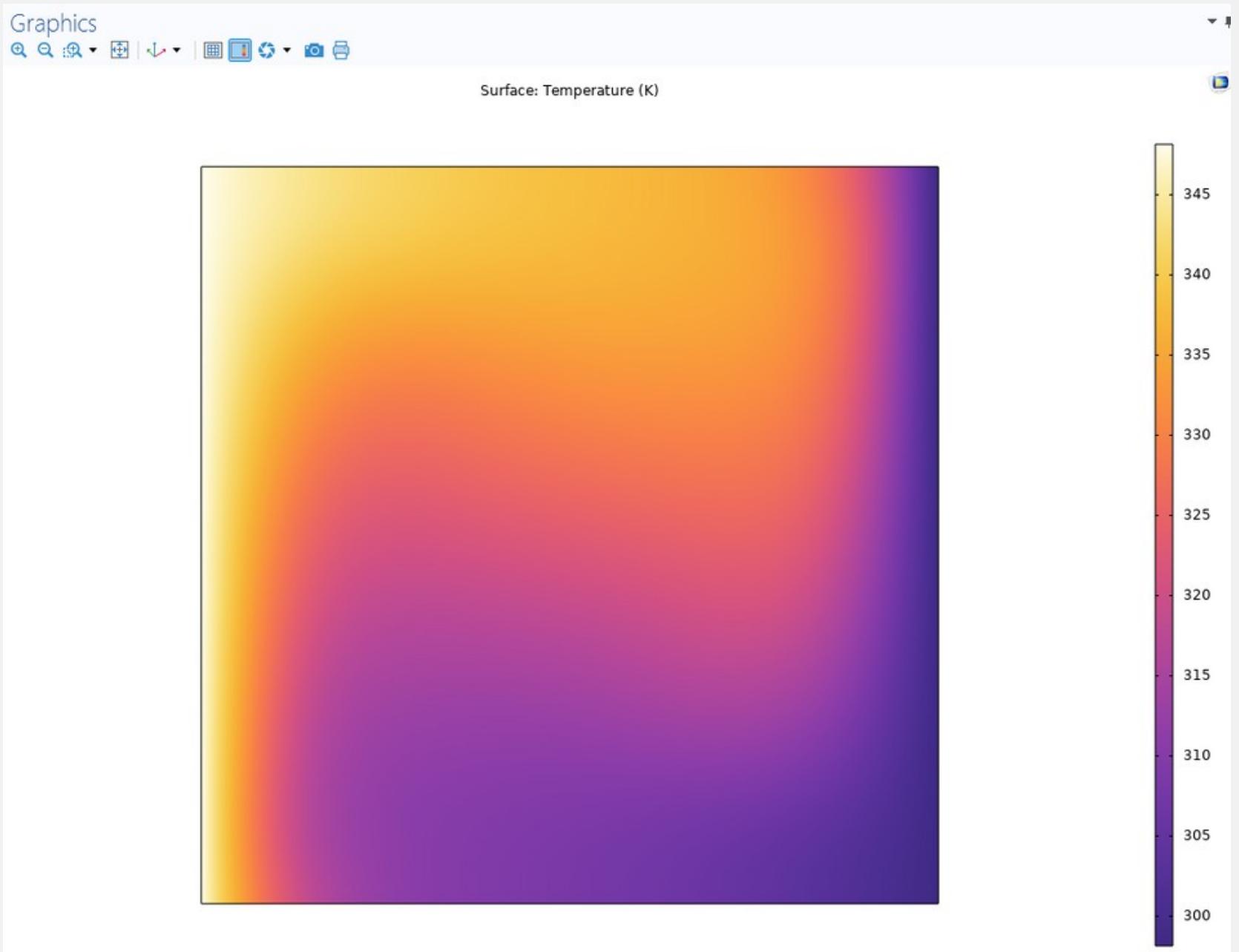
Geometry and Boundary Conditions

- Used Heat Transfer In Fluids (Laminar Flow) Package.
- We assumed an air domain of width 2cm.
- Left side of the domain was assigned a constant temperature of 75 degree C whereas the right side was kept at ambient 25 degree C.
- Gravity acts downwards and boussinesq approximation was valid.
- Calculated results shows the STEADY STATE profile.

Velocity Profile



Temperature Profile



Explanation

- The domain has hot and cold walls creating natural convection, where heat differences drive the fluid motion.
- Warm air becomes lighter and rises, while cool air sinks, creating circular flow loops inside the box.
- The Boussinesq approximation simplifies fluid flow calculations when density changes are small but important — like in natural convection.
- Density is constant everywhere except in the buoyancy term of the Navier-Stokes equations.

**Thank you
very much!**

Group-3