

Natural convection in a square cavity

1. Purpose

The Purpose of CFD Lab 6 is to simulate natural convection in a two-dimensional square box on a mesh consisting of quadrilateral elements.

Students will validate **Nusselt number** obtained by simulation using empirical data, analyze the differences between calculations and experimental results, and present results in CFD Lab report.

2. Simulation Design

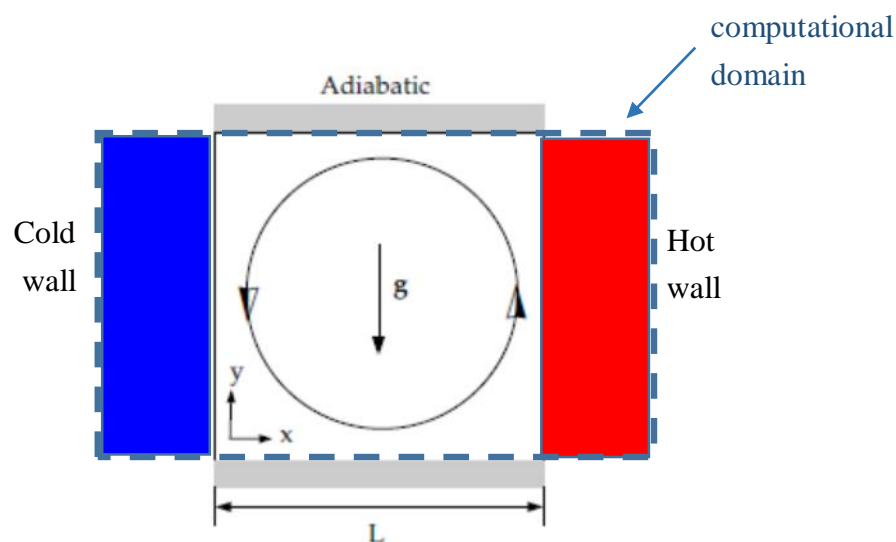
The problem to be considered is shown schematically in Figure 1. A square box of side L has a hot right wall, a cold left wall, and adiabatic top and bottom walls. Gravity acts downwards. A buoyant flow develops because of thermally-induced density gradients. The medium contained in the box is air. The working fluid has a Prandtl number of approximately 0.74, and the Rayleigh number based on L is in the range of $5 \cdot 10^3$ – $5 \cdot 10^5$. This means the flow is laminar. The Boussinesq assumption is used to model buoyancy.

$$T_0 = (T_h + T_c)/2.$$

$$L = 0.01, 0.02, 0.04 \text{ m.}$$

$$T_{\text{hot}} = 400 \text{ K}$$

$$T_{\text{cold}} = 300 \text{ K}$$



In addition to air convection inside a cavity, the heat conducted through the walls is simulated. It means that the conjugated heat transfer is considered.

The effect of Rayleigh and Prandtl number on flow and heat transfer is studied.

For this purpose the length value L and Prandtl number value are varied. The values are provided by the teacher.

A structured conformal mesh should be created for walls and fluid part.

3. Project Schematic in Ansys Workbench

3.1. Start ANSYS Workbench.

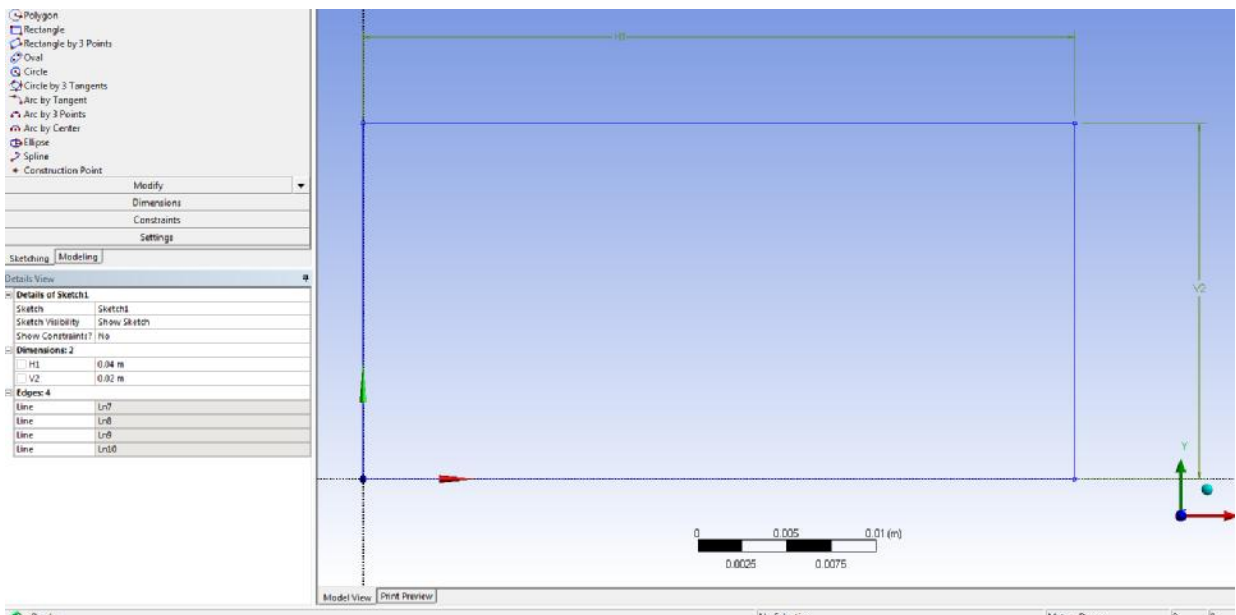
3.2. Create a scheme of your project in Workbench Project Schematic Window.

3.3. **File > Save As.** Save the workbench file.

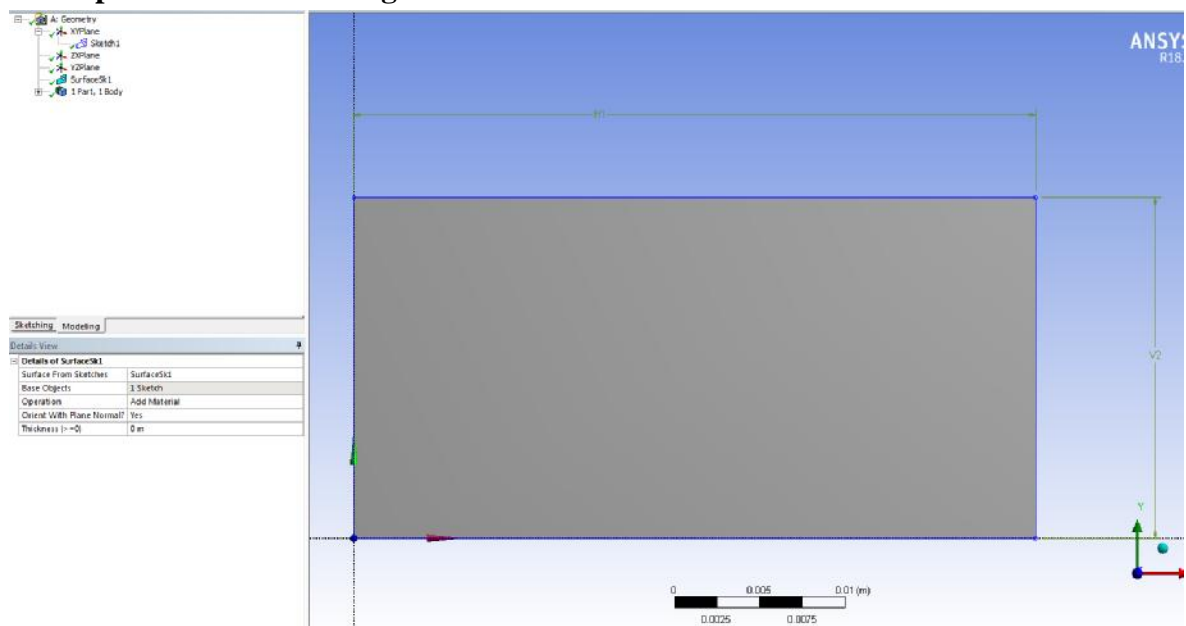
4. Geometry

4.1. Start **Design Modeler** to create the computational area.

4.2. **Sketching>Draw>Rectangle.** Create a rectangular area of given dimensions.

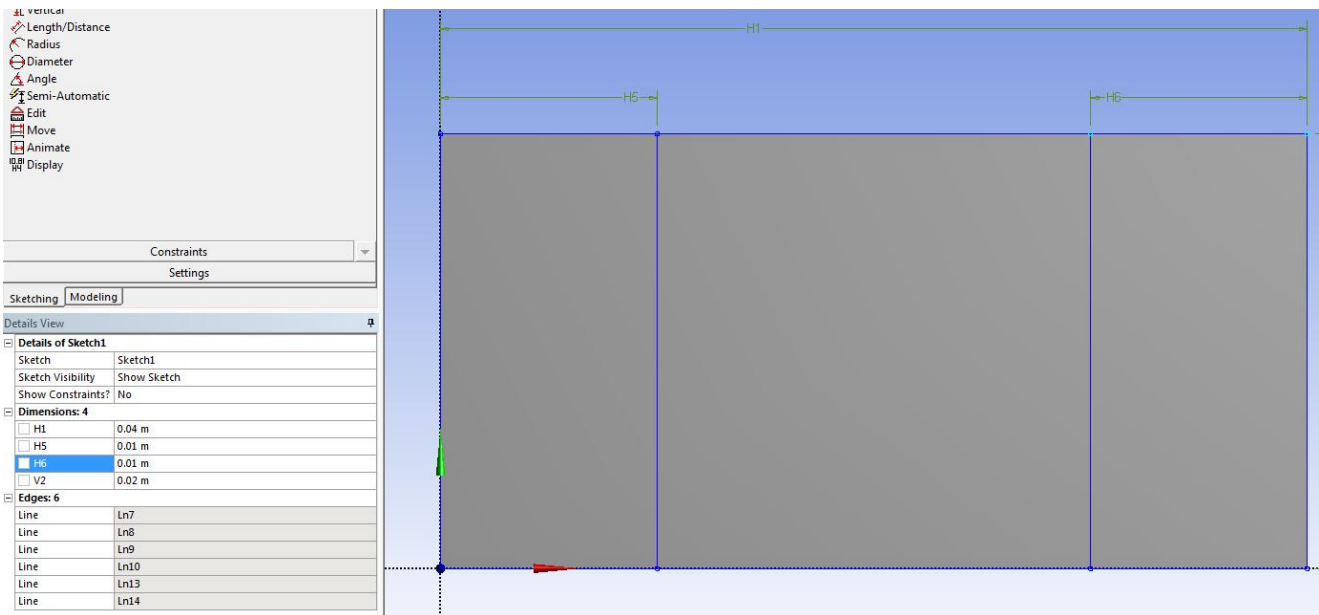


4.3. **Concept>Surfaces from Edges.** Create a surface.

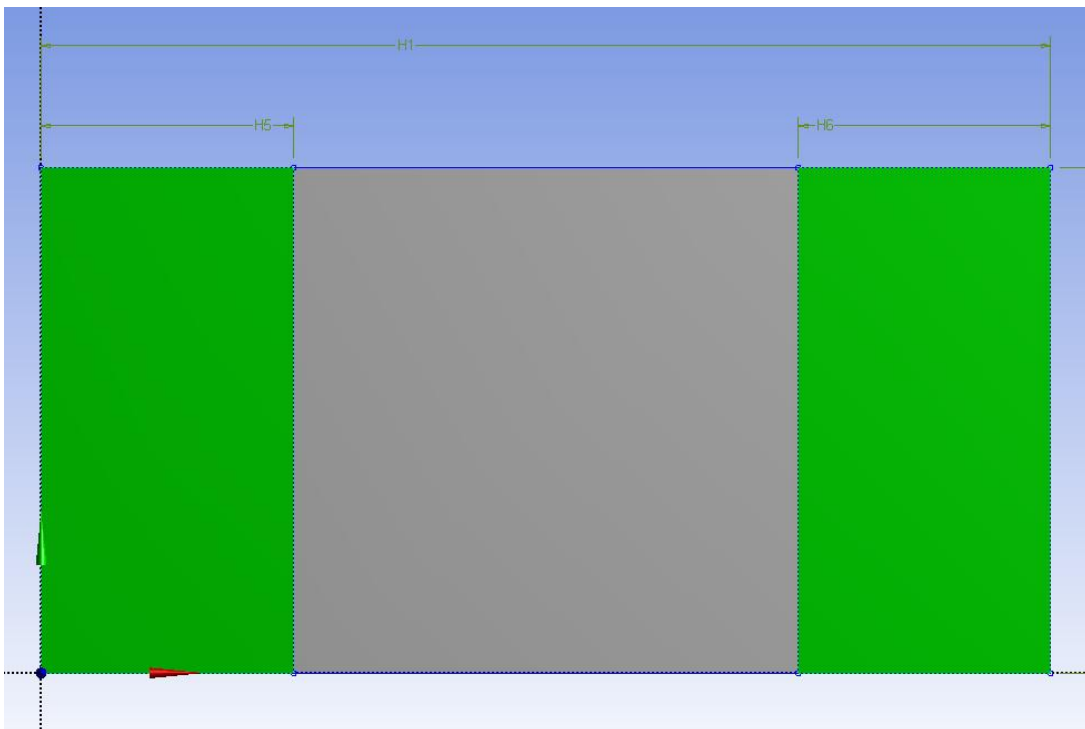


4.4. **Sketching>Draw>Line.** Create two vertical lines.

4.5. **Sketching>Dimensions.** Set the horizontal dimensions.



4.6. **Tools>Face Split.** Choose as **Target Face** the generated Surface. Choose as **Tool Geometry** the lines. Click **Generate**.



4.7. Close **Design Modeler**.

5. Meshing

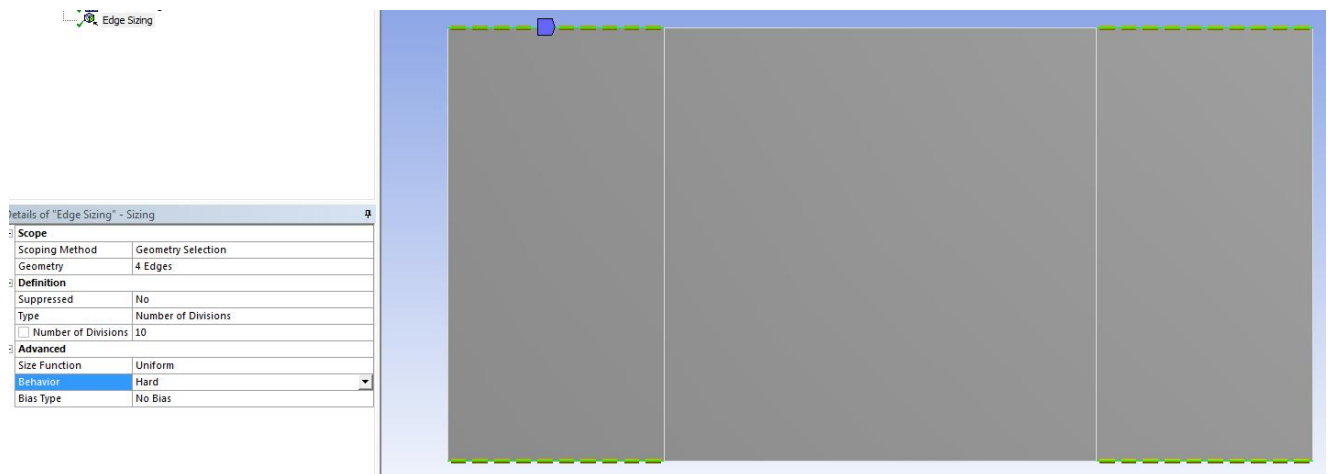
5.1. Start **Meshing**.

5.2. Details of “Mesh”-> **Physic Reference->CFD**

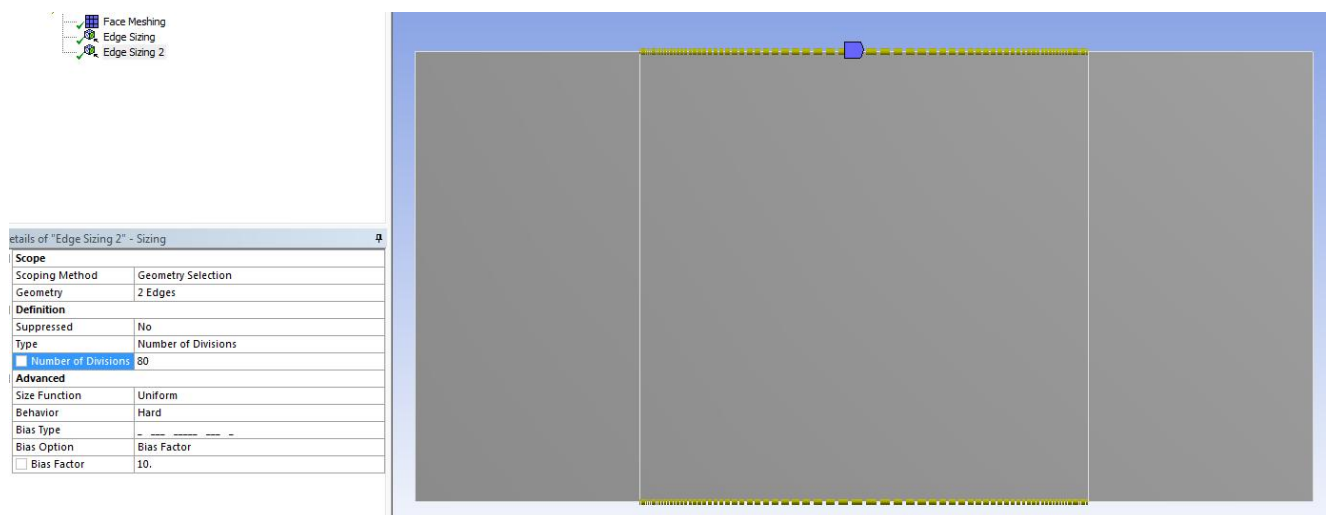
5.3. **Mesh>Insert > Face Meshing**. Choose three Faces in **Geometry**.

Create a nonuniform structured mesh. The grid nodes in the cavity should be concentrated near the walls. An inflation is not necessary for the solid parts.

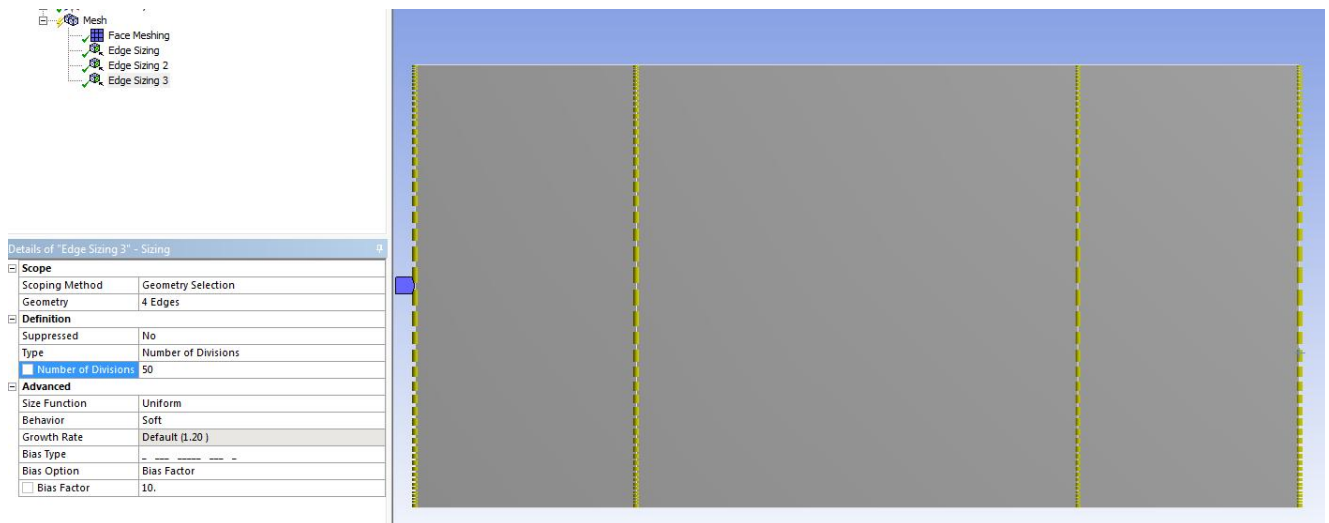
5.4. **Mesh>Insert > Sizing**. Choose four horizontal edges. Set the number of divisions without Bias. An example is shown below. But you can choose another number of divisions.



5.5. **Mesh>Insert > Sizing**. Choose two central horizontal edges. Set the number of divisions with a Bias factor providing concentrations near the wall. An example is shown below. But you can choose another number of divisions.



5.6. **Mesh>Insert > Sizing.** Choose four vertical edges. Set the number of divisions with a Bias factor providing concentrations near the wall. An example is shown below. But you can choose another number of divisions.

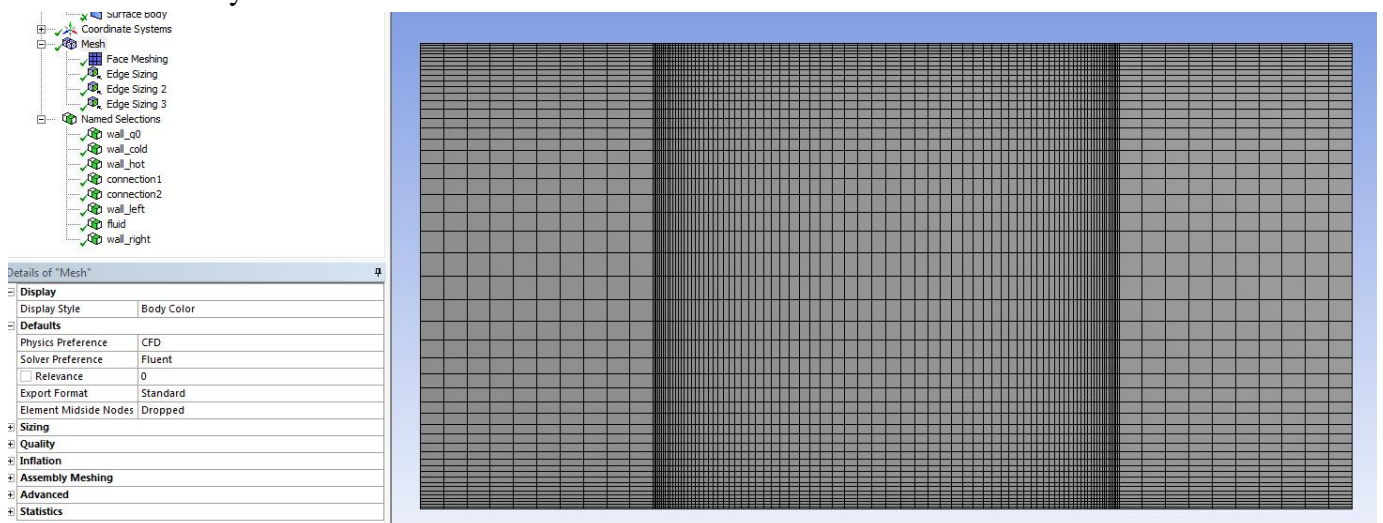


Note! We create conformal grid, therefore the nodes should be distributed equally for faces corresponding to solid parts and fluid parts. In general, we do not need so many divisions for solid bodies.

5.7. Click **Generate Mesh.**

5.8. Create the following **Named Selections**:

- for the left cold surface of the wall “wall_cold”,
- for the right hot surface of the wall “wall_hot”,
- for the top and bottom adiabatic surfaces (6 edges in total) “walls_q0”,
- for connections between air filled cavity and walls “connection1” and “connection2”.
- for solid walls: “wall_left”, “wall_right”.
- For cavity: “fluid”

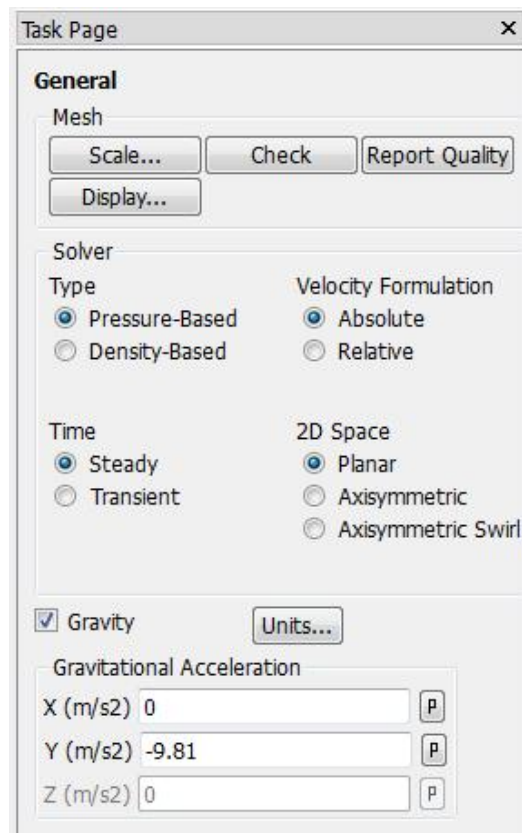


5.9. Close **Meshing.**

5.10. **File > Save Project.** Save the project and close the window. **Update Mesh** on Workbench if necessary.

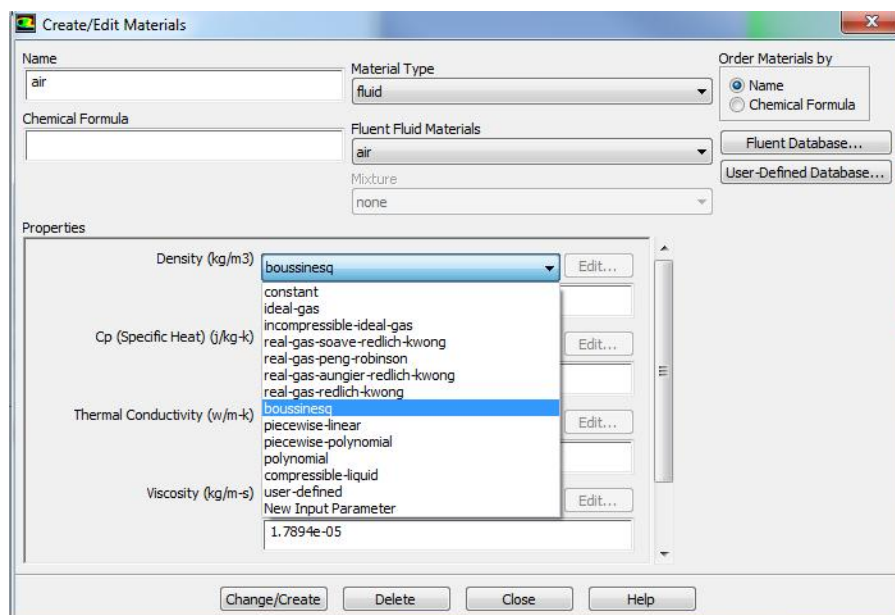
6. Solving in Fluent

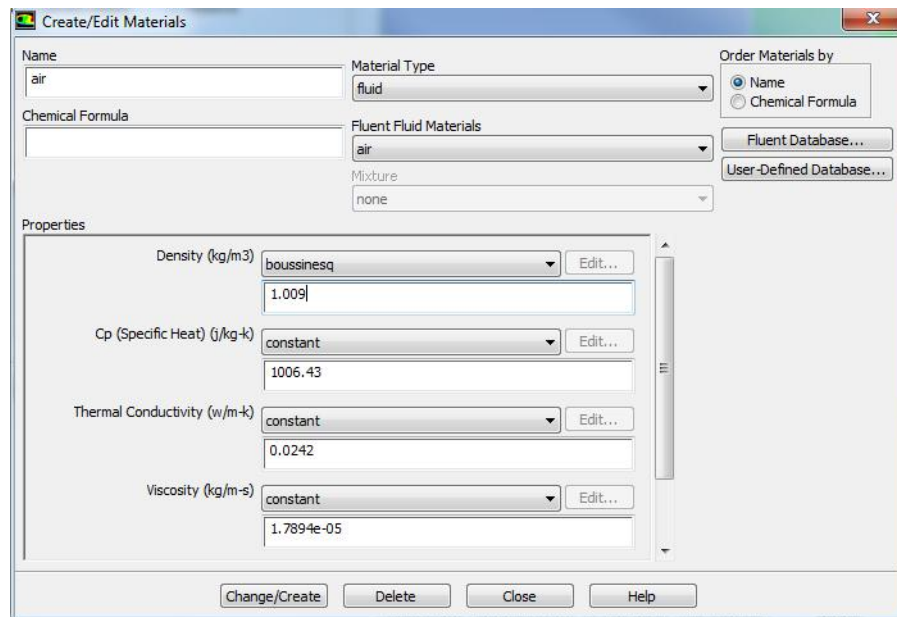
- 6.1. Run **Fluent**.
- 6.2. Add the effect of gravity to the model.



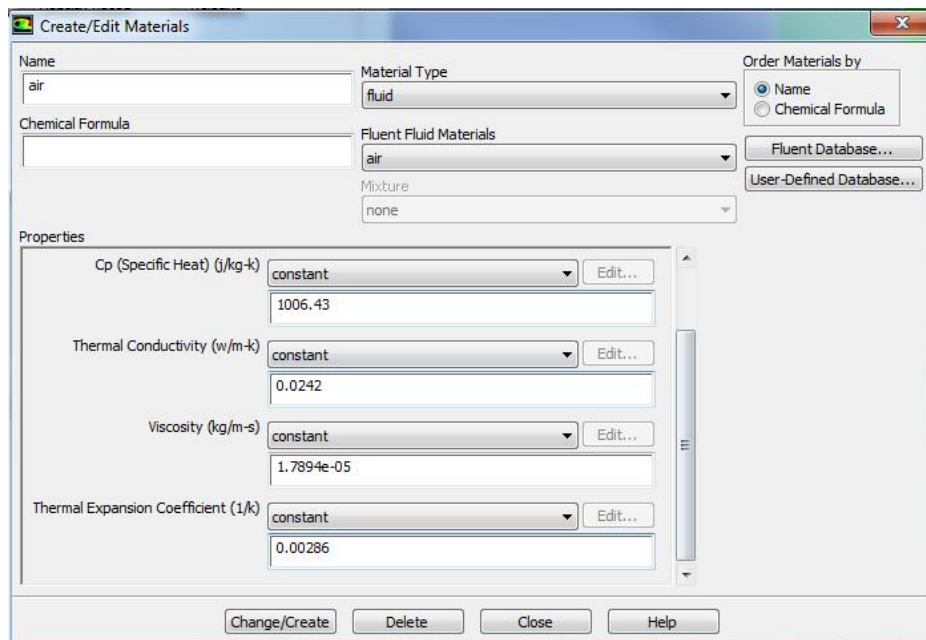
Be sure that the following options are chosen in **Setup**.

- 6.3. **Energy > On**
- 6.4. **Model > Viscous > Laminar**
- 6.5. **Materials > Fluid**. Select **boussinesq** from the drop-down list for **Density** and then enter the value corresponding to mean temperature. On the picture below an example for air is given.

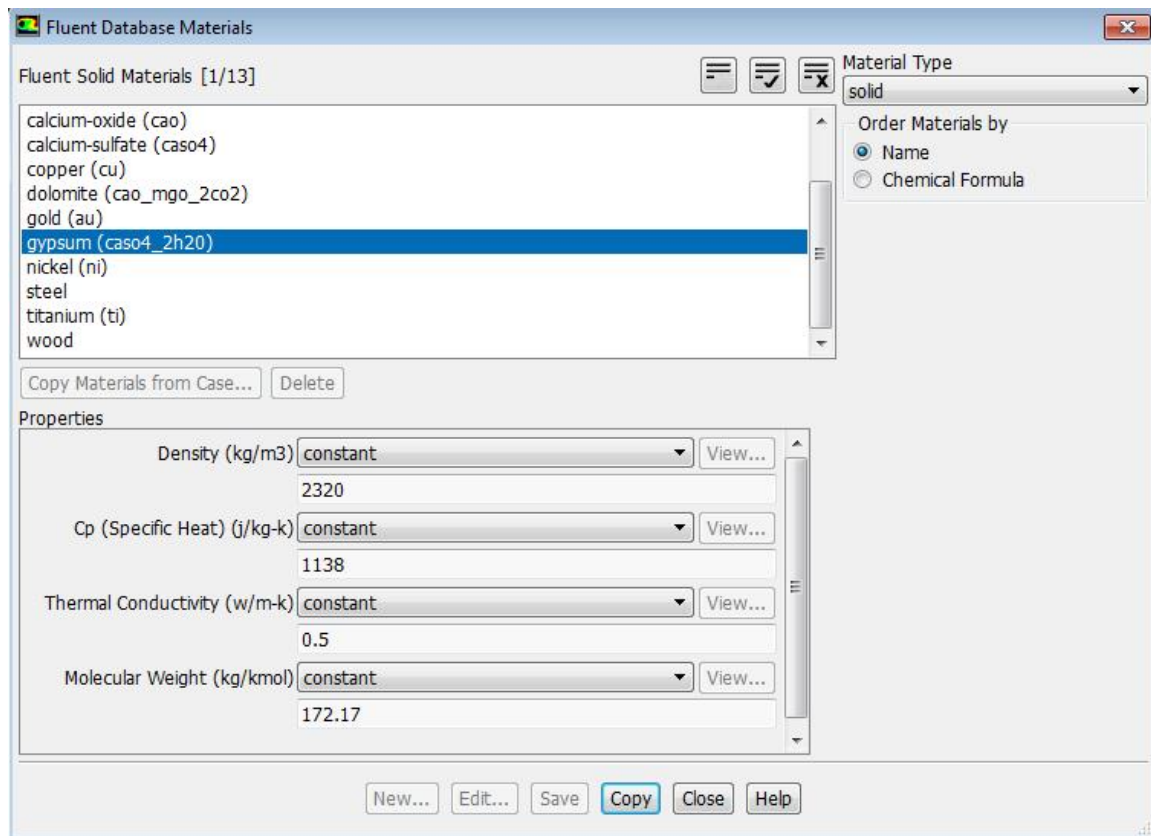




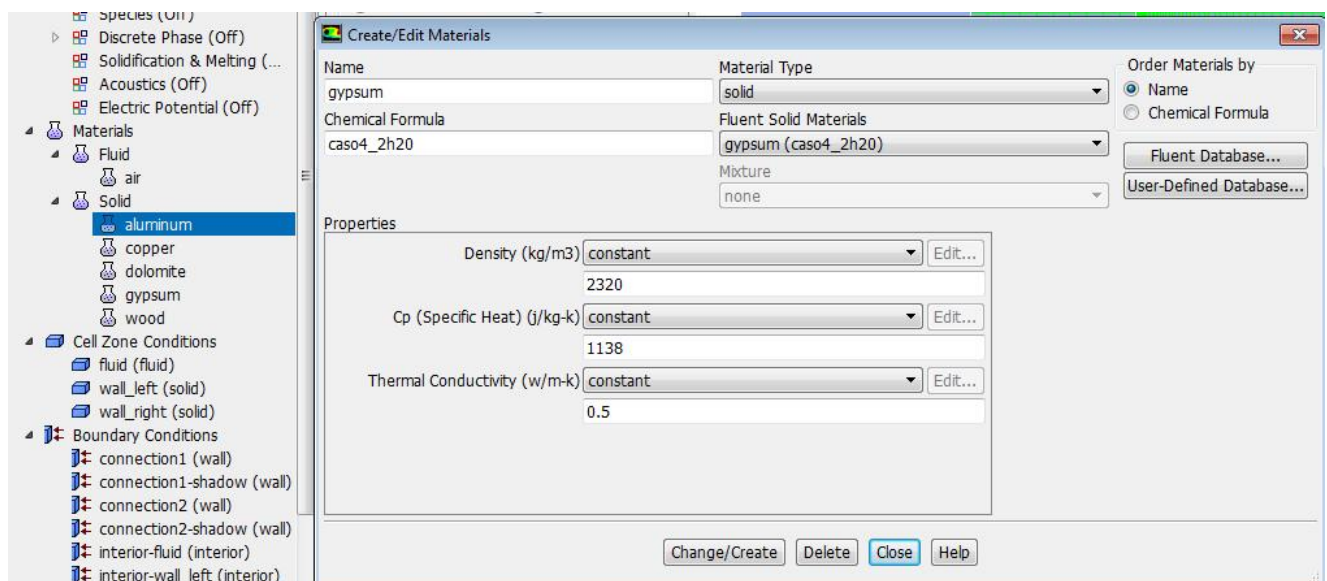
6.6. Change the **Thermal Expansion Coefficient**. Click **Change/Create**.



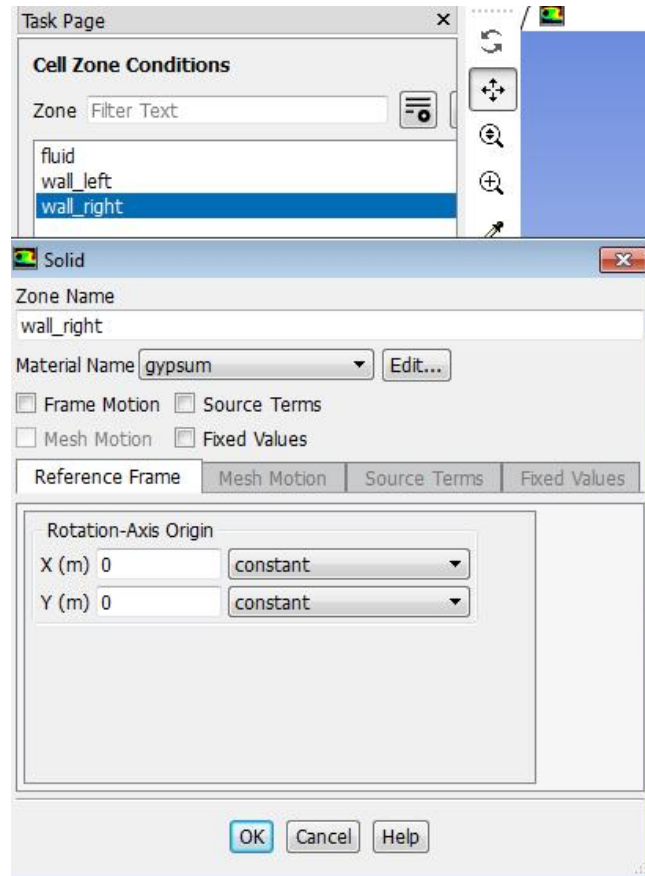
6.7. **Materials>Solid>Fluent Database**. Choose solid materials with different thermal conductivity from the list. Click **Copy**.



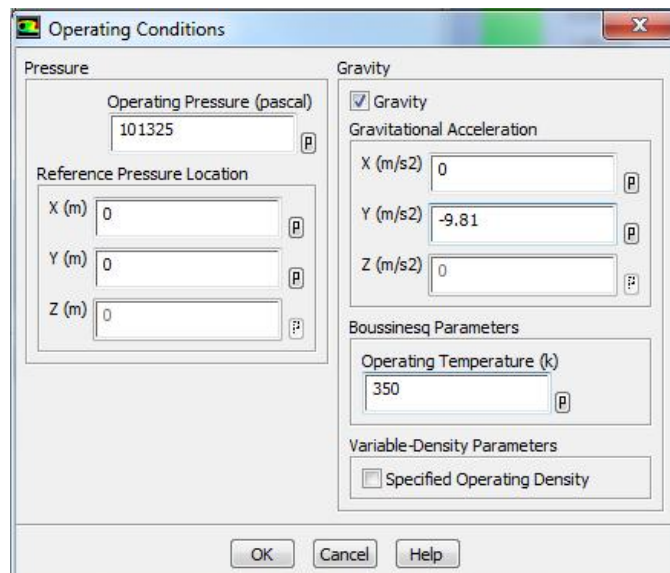
You will see chosen materials in the list of Solids.



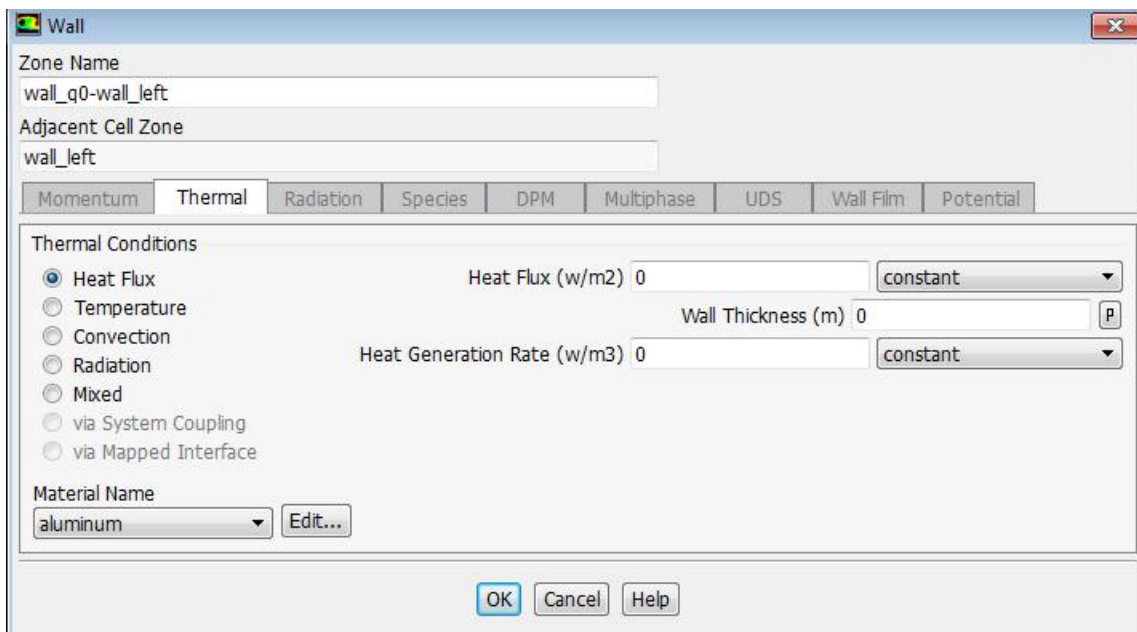
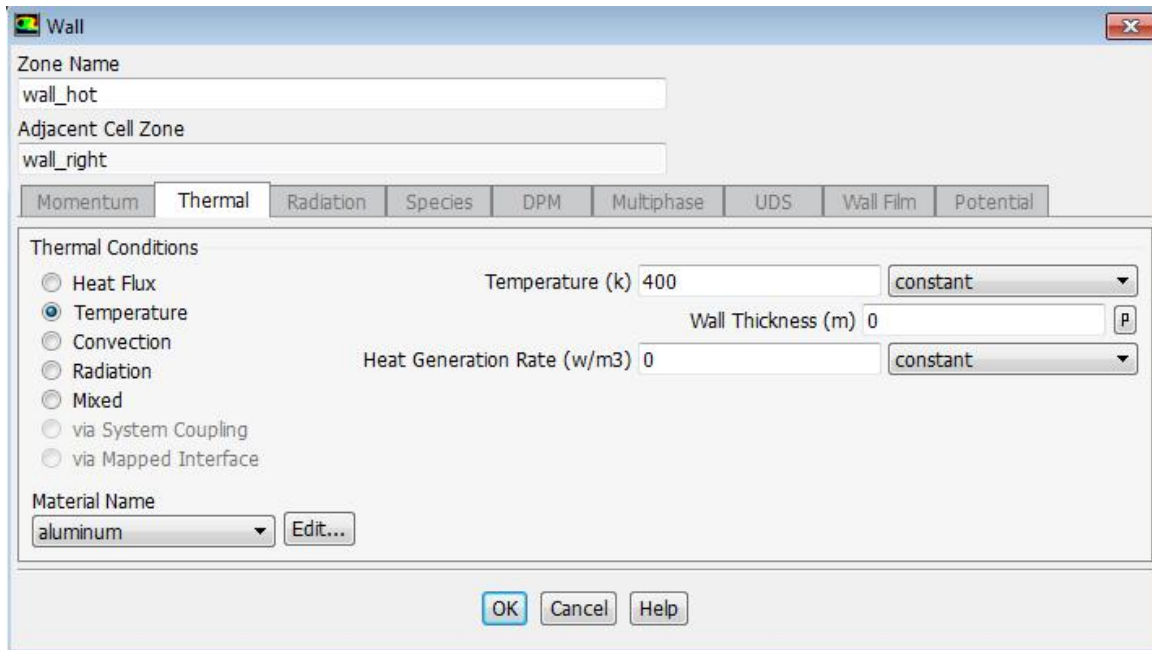
6.8. **Cell Zone Conditions.** Choose solid materials for the left and for the right walls.



6.9. **Boundary Conditions> Operating Conditions.** Check the operating temperature. It should be equal to the mean temperature of the walls.



6.10. **Setup>Boundary Conditions.** Set the boundary conditions. The temperature for the left wall should be set equal to 300K, the temperature for the right wall – 400K, for adiabatic walls we need the condition “heat flux=0”. No boundary conditions are required for connection and shadow boundaries.



6.11. **Solution > Run calculation.** Change number of iterations to 1000 and click Calculate.

6.12. **File > Save Project.**

Perform the next calculations corresponding to given length values or Prandlt number values.

7. Results post-processing

Pictures to be created: for two different Prandtl numbers and two Rayleigh numbers:

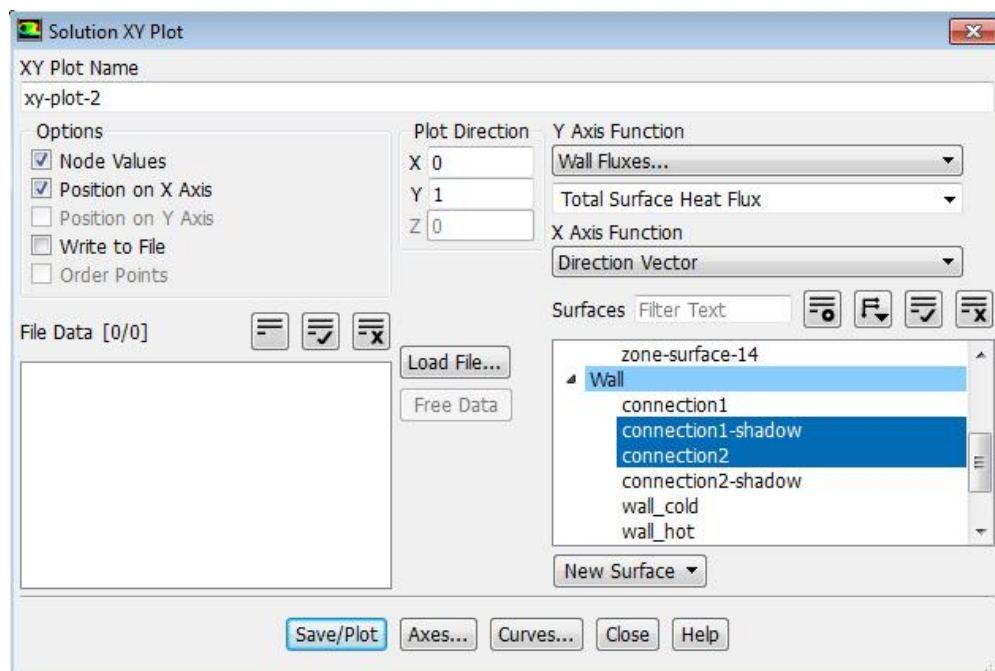
- Temperature fields
- Velocity vector fields
- Stream function fields
- Surface heat flux distribution along the cooled and heated surfaces of the cavity.

Data to be calculated

- Rayleigh number
- Average Nusselt number
- Average heat transfer coefficient

To create the XY Plots of surface heat transfer distribution on the vertical walls of the cavity do the following

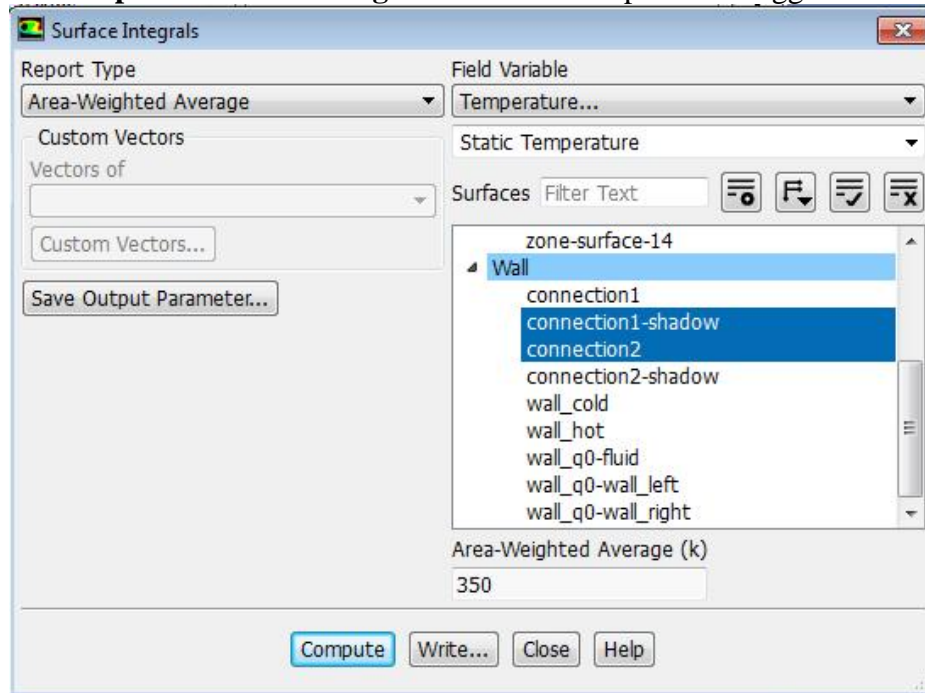
7.1.1. Results> Plots > XY Plot. Choose the options as suggested below.



7.2. Equations

To calculate the Rayleigh number corresponding to the calculation do the following:

7.2.1. Results>Reports>Surface Integrals. Choose the options as suggested below.



7.2.2. Use the temperatures printed in **Console** to define the temperature drop between the heated and cooled walls of the cavity.

7.2.3. Calculate Rayleigh number using the formula

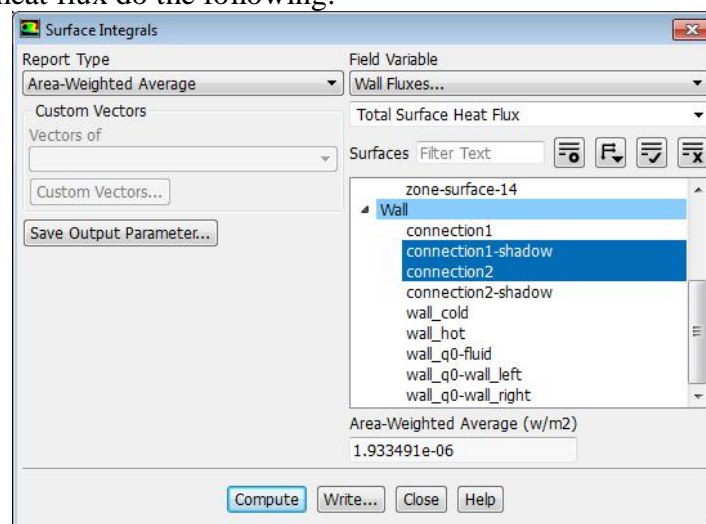
$$Ra = \frac{g\beta\Delta TL^3}{\nu^2} \cdot Pr$$

Here β can be calculated as $\beta = \frac{1}{0.5 \cdot (T_{hot} + T_{cold})}$

7.2.4. The Nusselt number corresponding to **Fluent simulation** depends on temperature drop and average heat flux:

$$Nu = \frac{\frac{q}{\Delta T} \cdot L}{\lambda}$$

To obtain the average heat flux do the following:



7.2.5. The Nusselt number corresponding to **empirical equation** depend on Grashof and Prandtl number:

- For air flow the average Nusselt number can be calculated by empirical equation

$$Nu = 0.231 (Gr \cdot Pr)^{0.25},$$

here Grashof number is

$$Gr = \frac{g\beta\Delta TL^3}{\nu^2}.$$

- For the other liquids the average Nusselt number can be calculated by empirical equation

$$Nu = 0.18 \cdot \left(\frac{Gr \cdot Pr^2}{0.2 + Pr} \right)^{0.28}.$$

Boussinesq model in Fluent

13.2.4.2. The Boussinesq Model

For many natural-convection flows, you can get faster convergence with the Boussinesq model than you can get by setting up the problem with fluid density as a function of temperature. This model treats density as a constant value in all solved equations, except for the buoyancy term in the momentum equation:

$$(\rho - \rho_0)g \approx -\rho_0\beta(T - T_0)g \quad (13-2)$$

where ρ_0 is the (constant) density of the flow, T_0 is the operating temperature, and β is the thermal expansion coefficient. Equation 13-2 is obtained by using the Boussinesq approximation $\rho = \rho_0(1 - \beta\Delta T)$ to eliminate ρ from the buoyancy term. This approximation is accurate as long as changes in actual density are small; specifically, the Boussinesq approximation is valid when $\beta(T - T_0) \ll 1$.

13.2.4.3. Limitations of the Boussinesq Model

The Boussinesq model should not be used if the temperature differences in the domain are large. In addition, it cannot be used with species calculations, combustion, or reacting flows.

13.2.4.4. Steps in Solving Buoyancy-Driven Flow Problems

The procedure for including buoyancy forces in the simulation of mixed or natural convection flows is described below.

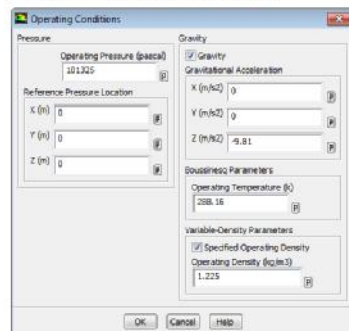
1. To activate the calculation of heat transfer, enable the energy equation by right-clicking **Energy** in the tree (under **Setup/Models**) and clicking **On** in the menu that opens (Figure 13.1: Enabling the Energy Equation).

Setup → **Models** → **Energy** **On**

2. Define the operating conditions in the **Operating Conditions** dialog box (Figure 13.3: The Operating Conditions Dialog Box).

Setup → **Cell Zone Conditions** → **Operating Conditions...**

Figure 13.3: The Operating Conditions Dialog Box



- a. Enable the **Gravity** option under **Gravity**.
- b. Enter the appropriate values in the **X**, **Y**, and (for 3D) **Z** fields for **Gravitational Acceleration** for each Cartesian coordinate direction. (Note that the default gravitational acceleration in ANSYS Fluent is zero.)
- c. If you are using the incompressible ideal gas law, check that the **Operating Pressure** is set to an appropriate (nonzero) value.
- d. Depending on whether or not you use the Boussinesq approximation, specify the appropriate parameters described below:

- If you are not using the Boussinesq model, the inputs are as follows:
 - If necessary, enable the **Specified Operating Density** option in the **Operating Conditions** dialog box, and enter a value for the **Operating Density**. See below for details.
 - Define the fluid density as a function of temperature as described in [Defining Properties Using Temperature-Dependent Functions](#) and [Density](#).

Setup → **Materials**

- If you are using the Boussinesq model (described in [The Boussinesq Model](#)) the inputs are as follows:
 - Enter the **Operating Temperature** (T_0 in Equation 13-2) in the **Operating Conditions** dialog box.
 - Select **boussinesq** in the drop-down list for **Density** in the **Create/Edit Materials** dialog box as described in [Defining Properties Using Temperature-Dependent Functions](#) and [Density](#), and enter a constant value.
 - Also in the **Create/Edit Materials** dialog box, enter an appropriate value for the **Thermal Expansion Coefficient** (β in Equation 13-2) for the fluid material.

Note that if your model involves multiple fluid materials you can choose whether or not to use the Boussinesq model for each material. As a result, you may have some materials using the Boussinesq model and others not. In such cases, you will need to set all the parameters described above in this step.

3. Define the boundary conditions. The boundary condition dialog boxes can be opened by right-clicking the boundary name in the tree (under **Setup/Boundary Conditions**) and clicking **Edit...** in the menu that opens; alternatively, you can open them from the **Boundary Conditions** task page:

Setup → **Boundary Conditions**

The boundary pressures that you specify at pressure inlet and outlet boundaries are the redefined pressures as given by Equation 13-3. In general you should enter equal pressures, p' , at the inlet and exit boundaries of your ANSYS Fluent model if there are no externally imposed pressure gradients.

4. Set the parameters that control the solution, using the **Solution Methods** task page.

Solution → **Solution Methods**

- a. Select **Body Force Weighted** or **Second Order** in the drop-down list for **Pressure** under **Spatial Discretization** in the **Solution Methods** task page.
- b. If you are using the pressure-based solver, selecting **PRLSTO1** for **Pressure** under **Spatial Discretization** is recommended.
- c. Add cells near the walls to resolve boundary layers, if necessary.

See also [Solving Heat Transfer Problems](#) for information on setting up heat transfer calculations.