Hands-on session 5 – Natural convection

Abstract

In this session, the transient analysis of room heating with a conventional heater is studied.. Evaluate the heat transfer rate of the heater and estimate the temperature increase in the room after 2 minutes.

Goal

The aim of this hands-on session is to solve the transient case of natural convection. The user will get familiar with the following aspects of the software and of a CFD study:

1. set-up the buoyancy effects;
2. set-up a transient analysis;
3. create the animation of transient case.

Hands-on session 5 – Natural convection 1
1 Introduction 3
2 Tasks 3
3 Tips 4
3.1 Gravity 4
3.2 Boussinesq approximation 4
3.3 Transient analysis 6
3.4 Create the animation 8

1 Introduction

The transient analysis of room heating with a conventional heater mounted below the window is studied. The case simulates the natural convection inside a room (T_{init} = 20°C) during the cold days, where the warm air coming from the heater (T_{heater} = 60°C) heats up the room and blocks the cold air coming from the window (T_{window} = 0°C). The case is simplified to 2D to reduce the simulation time. The room walls have a temperature of 20°C, whereas the wall furthest from the window has a lower temperature of 15°C. Evaluate the heat transfer rate of the heater and estimate the temperature increase of air in the room after 2 minutes.

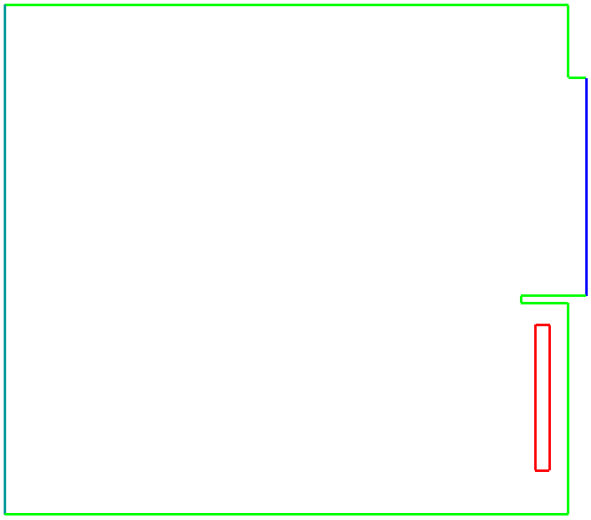


Figure 1 - The 2D geometry of analysed room with a heater.

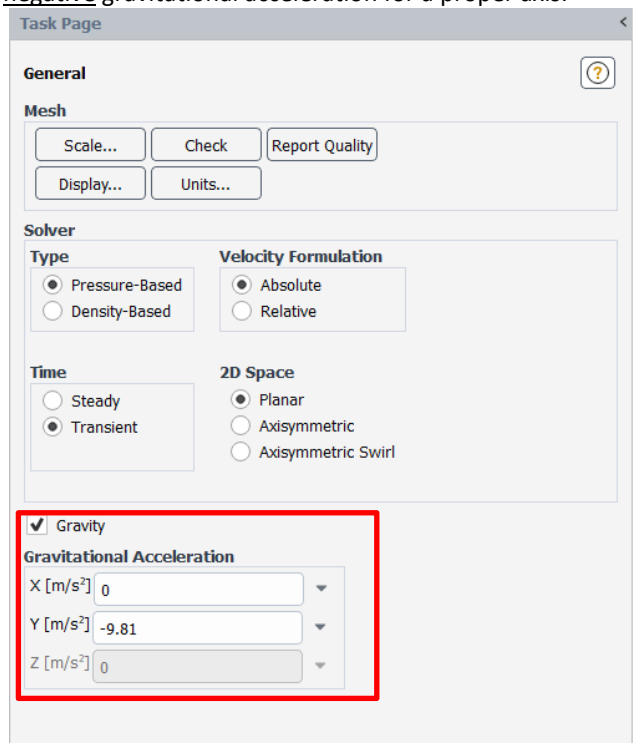
2 Tasks

1. Set up the case
2. Choose appropriate viscous model
3. Turn on the buoyancy effects and input the important parameters
4. Choose the transient solver and set up the time step and number of iterations
5. Set up the animation
6. Run the simulation and evaluate the heat transfer rate and average room temperature

3 Tips

3.1 Gravity

In order to account the natural convection, the buoyancy effects have to be turned on in the solver. First, the gravitational acceleration has to be set up in the General tab by turning on the Gravity option and introducing the negative gravitational acceleration for a proper axis.



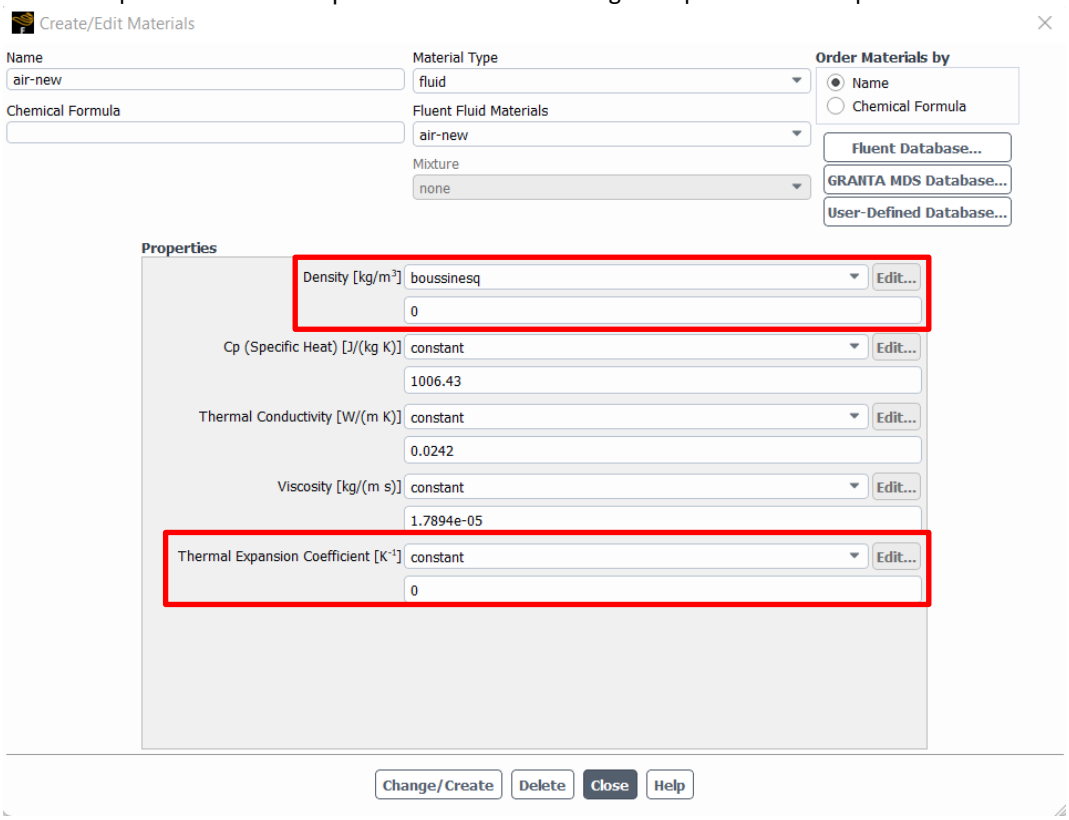
3.2 Boussinesq approximation

Next, for the simplification of the case, the Boussinesq approximation is used, which assumes constant density for all the governing equations apart from the buoyancy term in the momentum equation, where the density is defined using equation:

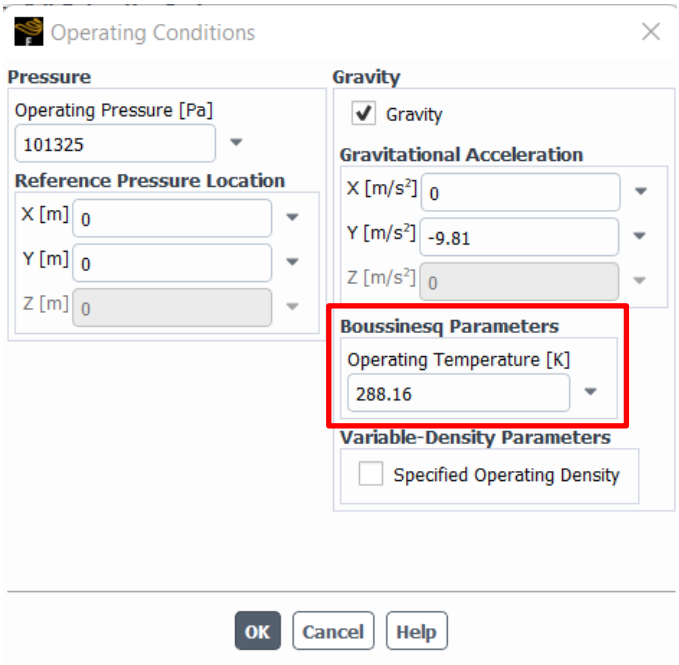
$$\rho = \rho_0(1 - \beta(T - T_0))$$

where ρ_0 is operating (constant) density, β is thermal expansion coefficient and T_0 is operating (constant) temperature. To turn on the Boussinesq approximation and introduce the properties to Fluent, go to Materials and change the boussinesq in the density list and introduce there the ρ_0 value calculated for the reference temperature. Also the

thermal expansion coefficient β calculated for the average temperature of the problem.



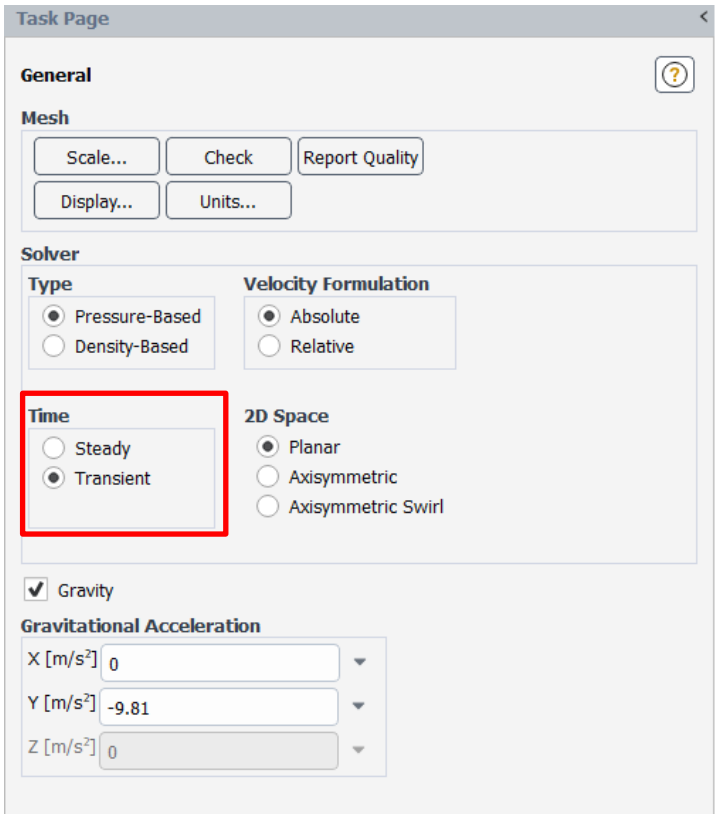
To specify the operating temperature T_0, go to Cell Zone Conditions -> Operating conditions -> Boussinesq parameters, and introduce there the temperature for which the operating density and all the other fluid constant properties were defined:



The operating density option in Variable-Density Parameters doesn't need to be turned on, as it's the operating density which would appear in other fluids body-force term of the momentum equation, which do not use the Boussinesq approximation. In our domain we use only one zone with single fluid.

3.3 Transient analysis

To turn on the transient settings for the solver, go to General tab and change the settings of the solver:



For the transient analysis, one of the most important parts is the initialization of the domain for the T_0, from which the simulation will begin. In order to initialize the domain with temperature T_0, the standard initialization with prescribed initial temperature is suggested.

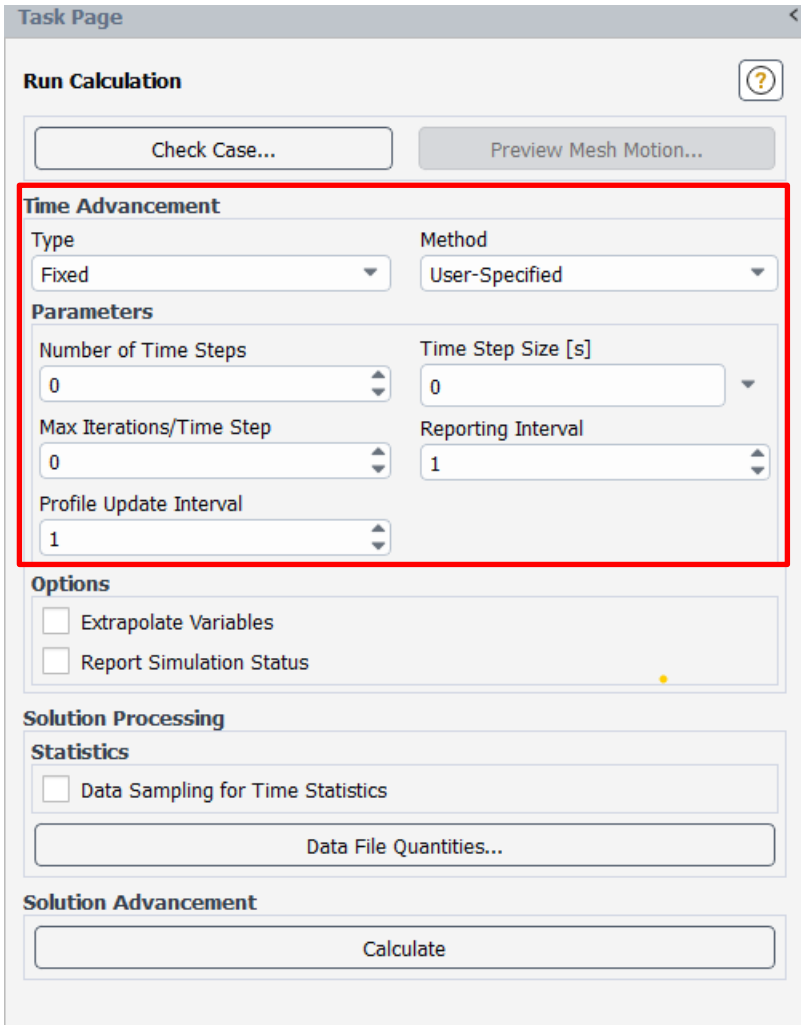
Next, the time step value for the given problem should be set. It can be estimated using the following equations:

$$\tau = \frac{L}{\sqrt{g\beta\Delta T L}}$$

$$\Delta t \approx \frac{\tau}{n}$$

where n is the integer value (4 for this problem), depending on the expected accuracy of the time evolution.

In the Run Calculation settings use the Fixed time step, introduce the time step value and desired number of timesteps and the maximal number of iterations has to be adjusted in order to achieve the convergence for each time step. It is suggested to start with bigger value (e.g. 50 iterations) and decrease them after few timesteps to speed up the simulation.



3.4 Create the animation

In order to create a simple animation of the solution in Fluent, go to Solution -> Acitvities -> Create -> Solution animations and define new animation for contours of temperature. Also, the Autosave activity needs to be set up for every timestep.

Then the animation can be played or saved in form of movie in Results -> Animation -> Playback window.