Transient flow and heat equations - the Rayleigh-Benard instability

Directory: RayleighBenard
Solvers: HeatSolve, FlowSolve
Tools: ElmerGUI
Dimensions: 2D, Transient

Case definition

This tutorial is about simulating the developing of the Rayleigh-Benard instability in a rectangular domain (Figure 1) of dimensions 0.01 m height and 0.06 m length. The simulation is performed with water and the needed material parameters of water are presented in Table 1. The temperature difference between the upper and lower boundary is set to 0.5 so that lower one has the temperature of 283.5 K and the upper one has the temperature of 283 K.

The density of water is inversely proportional to its temperature. Thus, heated water starts to flow upwards, and colder downwards due to gravity. In this case we assume that the Boussinesq approximation is valid for thermal incompressible fluid flow. In other words, the density of the term $\rho\vec{f}$ in the incompressible Navier-Stokes equation can be redefined by the Boussinesq approximation

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

where β is the heat expansion coefficient and the subscript 0 refers to a reference sate.

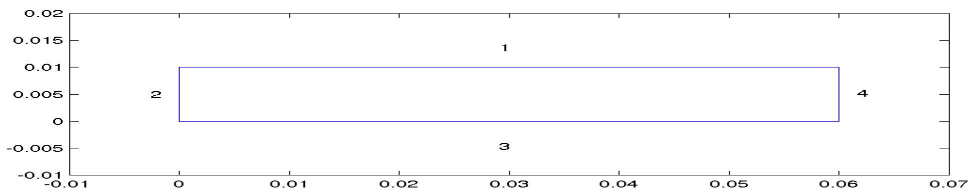

Figure 1: Domain.

Solution procedure

The mesh is given in ElmerGrid format in file `box.grd`, load this file.

Table 1: Material parameters.

parameter	value
density	1000 kg/m^3
viscosity	1040e-6 Ns/m^2
heat capacity	4190 J/(kg·K)
heat conductivity	0.6 W/(m·K)
heat expansion coefficient	1.8e-4 K^{-1}
reference temperature	283 K

```
File
  Open -> box.grd
```

You should obtain your mesh and may check that it consists of 3036 bilinear elements.

There is a possibility to divide and unity edges to simplify the case definition in the future.

```
Choose (left wall + right wall (Ctrl down)) -> unify edge
```

After we have the mesh we start to go through the Model menu from the top to bottom. In the Setup we choose things related to the whole simulation such as file names, time stepping, constants etc. The simulation is carried out in 2-dimensional cartesian coordinates. 2nd order bdf time-stepping method is selected with 200 steps and with step size of two seconds.

```
Model
  Setup
    Simulation Type = Transient
    Steady state max. iter = 20
    Time Stepping Method = bdf
    BDF Order = 2
    Time Step Intervals = 200
    Time Step Sizes = 2.0
    Gravity = ...
```

In the equation section we choose the relavan equations and paremeters related to their solution. In this case we'll have one set of equations (named "Natural Convection") which consists of the heat equation and of the Navier-Stokes equation.

When defining Equations and Materials it is possible to assign the to bodies immediately, or to use mouse selection to assign them later. In this case we have just one body and therefore its easier to assign the Equation and Material to it directly. It is important to select that the convection is computed since that couples the velocity field to the heat equation.

The system may include nonlinear iterations of each equation and steady state iterations to obtain convergence of the coupled system. It is often a good idea to keep to number of nonlinear iterations in a coupled case low. Here we select just one nonlinear iteration for both equations. For the linear system solvers we are happy to use the defaults. One may however, try out different preconditioners (ILU1,...) or direct Umfpack solver, for example.

```
Model
  Equation
    Name = Natural Convection
    Apply to Bodies = 1
    Heat Equation
      Active = on
      Convection = Computed
      Edit Solver Setting
        Nonlinear System
```

```
        Max. iteations = 1
    Navier-Stokes
      Active = on
      Edit Solver Setting
        Nonlinear System
          Max. iteations = 1
```

The Material section includes all the material parameters. They are divided to generic parameters which are direct properties of the material without making any assumptions on the physical model. Such as the mass. Other properties assume a physical law, such as conductivities and viscosity.

```
Model
  Material
    Name = Fluid
    Apply to Bodies = 1
    General
      Density = 1000
      Heat Capacity = 4190
      Reference Temperature = 283
      Heat expansion Coeff. = 1.8e.4
    Heat Equation
      Heat Conductity = 0.6
    Navier-Stokes
      Viscosity = 1.04e-3
```

A Body Force represents the right-hand-side of a equation. It is generally not a required field for a body. In this case, however, we apply the bouyancy resulting from heat expansion as a body force to the Navier-Stokes equation.

```
Model
  Body Force
    Name = Boyancy
    Apply to Bodies = 1
    Navier Stokes
      Boussinesq = on
```

Initial conditions should be given to transient cases. In this case we choose a constant Temperature field and a small initial velocity.

```
Model
  Initial Condition
    Name = Initial Guess
    Heat Equation
      Temperature = 283
    Navier-Stokes
      Velocity 1 = 1.0e-9
      Velocity 2 = 0.0
```

Only one boundary condition may be applied to each boundary and therefore all the different physical BCs for a boundary should be grouped together. In this case the Temperature and Velocity. The side walls are assumed to be adiabatic.

```
Model
  BoundaryCondition
    Name = Bottom
    Heat Equation
      Tempateture = 283.5
    Navier-Stokes
```

```
      Velocity 1 = 0.0
      Velocity 2 = 0.0

    Name = Top
    Heat Equation
      Tempateture = 283
    Navier-Stokes
      Velocity 1 = 0.0
      Velocity 2 = 0.0

    Name = Sides
    Navier-Stokes
      Velocity 1 = 0.0
      Velocity 2 = 0.0
```

The boundary conditions may also be set in the BoundaryCondition menu, or by clicking with the mouse. Here we use the latter approach as that spares us of the need to know the indexes of each boundary.

```
Model
  Set boundary properties
    Choose Bottom -> set boundary condition Bottom
    Choose Top -> set boundary condition Top
    Choose Sides -> set boundary condition Sides
```

After the case have been set-up we may check that all edges and surfaces have been assigned boundary condition, or an equation and material parameters.

```
View
  Select defined edges
  Select defined surfaces
```

If everything is set we may continue, otherwise one should go back to check what data is missing.

```
View
  Reset model view
```

In order to run ElmerSolver needs the mesh file and the command file. We have know basically defined all the information for ElmerGUI to write the command file. After writing it we may also visually inspect the command file.

```
Sif
  Generate
  Edit -> look how your command file came out
```

Before we can execute the solver we should save the files in a directory. Both the command file and the mesh file will be saved simultaneously to the destination directory.

```
File
  Save As
```

After we have succefully saved the files we may start the solver

```
Run
  Run solver
```

A convergence view automatically pops up showing relative changes of each iteration. When there are some results to view we may start the postprocessor also

```
Run
  Run postprocessor
```

Results

Due to the number of the time-steps the simulation may take around ten minutes. After the solver has finished, the results can be postprocessed with ElmerPost. When opening the result file using ElmerGUI ElmerPost only opens the first time-step. Therefore it is important to reopen the file and load the time-steps of interest. Pressing the button **All** selects all the calculated time steps. A video of the results can be viewed by selecting the option **Timestep Control** and pressing the button **Loop** under the **Edit** menu.

In Figures 2 and 3 the obtained temperature distribution and the velocity vectors are presented. The maximum velocity in the system should be about 0.4 mm/s.

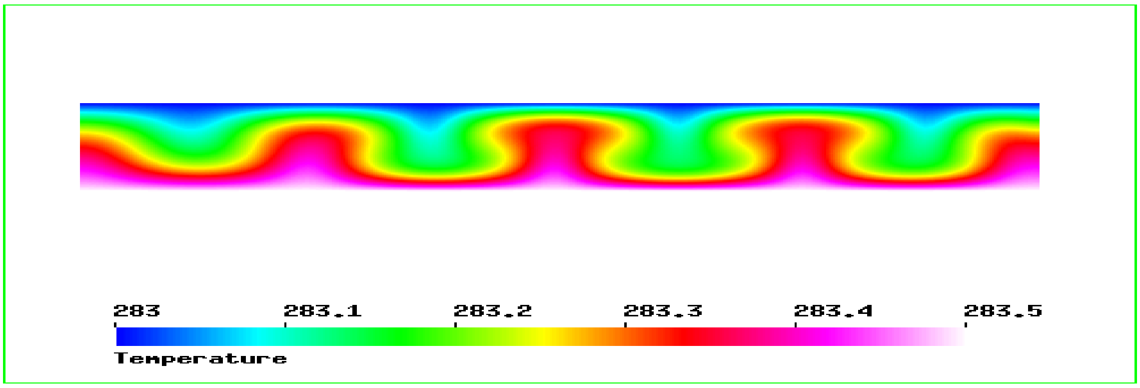

Figure 2: Temperature distribution at 260 s.

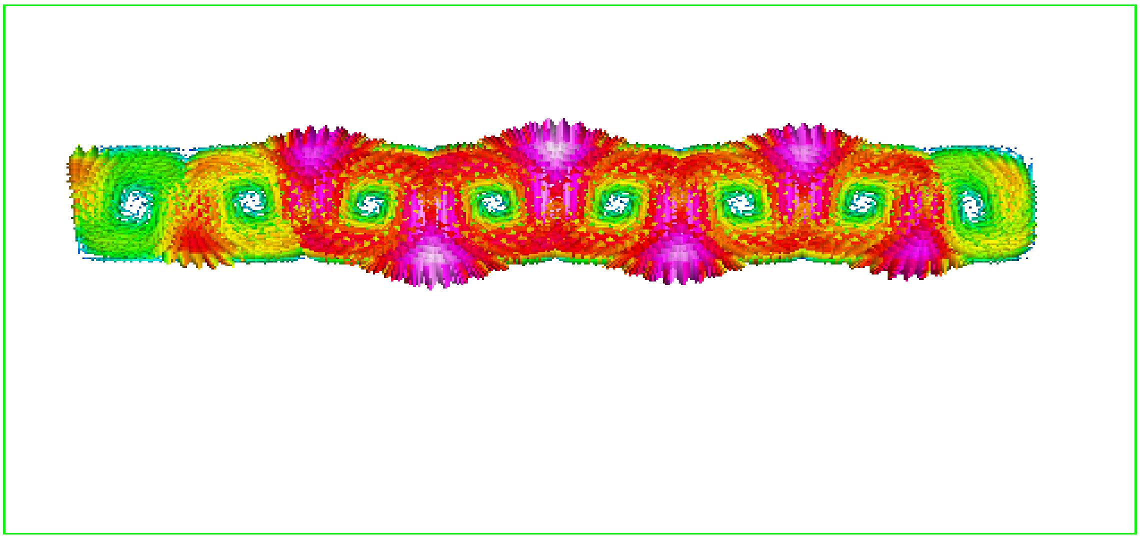

Figure 3: Velocity vectors at 260 s.