A chtMultiRegionSimpleFoam tutorial

Heat transfer

- 3 mechanisms: conduction, radiation and convection
- in solids, fluids and conjugate heat transfer
- Fluid as compressible flow
- Fluid as incompressible flow: the Boussinesq approximation

Boussinesq approximation

- density variation stays limited
- density as variable only in the gravitational term
- $\Delta\rho = \rho_0 \beta \Delta T$
  where
  ρ_0 is the reference density
  β is the coefficient of thermal expansion, and
  ΔT is the temperature difference

Heat transfer in OF21x

- solids: `laplacianFoam`
- fluids: `buoyantPimpleFoam` solver for transient compr cases
- fluids: `buoyantSimpleFoam` solver for steady state compr cases
- fluids: `buoyantBoussinesqPimpleFoam` solver for transient incompr cases
- fluids: `buoyantBoussinesqSimpleFoam` solver for steady state incompr cases
- cht: `chtMultiRegionFoam` solver for transient compr cases
- cht: `chtMultiRegionSimpleFoam` solver for steady state compr cases

chtMultiRegionSimpleFoam

```
cd $FOAM_SOLVERS/heatTransfer/chtMultiRegionFoam/chtMultiRegionSimpleFoam
ls
vi chtMultiRegionSimpleFoam.C
vi fluid/solveFluid.H
```

Comparing chtMultiRegionSimpleFoam

```
cd $FOAM_SOLVERS/heatTransfer/buoyantSimpleFoam
```

Comparing chtMultiRegionSimpleFoam

```
#include "createFluidMeshes.H"
#include "createSolidMeshes.H"
#include "createFluidFields.H"
#include "createSolidFields.H"
```

Plane Wall tutorial

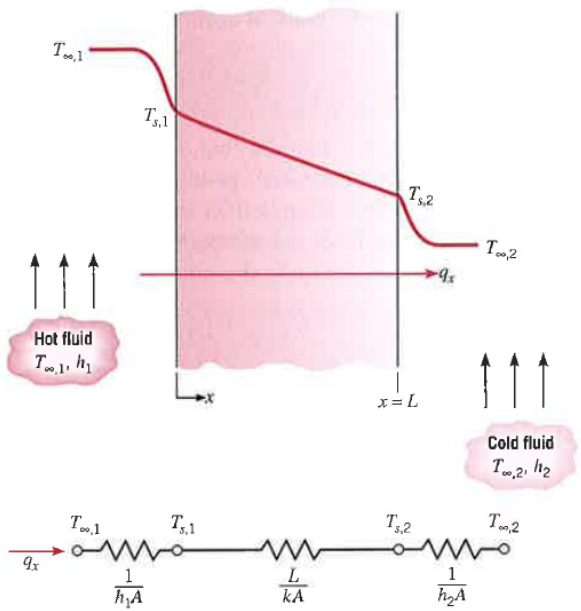


FIGURE 3.1 Heat transfer through a plane wall. (*a*) Temperature distribution. (*b*) Equivalent thermal circuit.

$$q = h_1(T_w - T_{ws}) = \frac{k}{L}(T_{ws} - T_{cs}) = h_2(T_{cs} - T_c)$$

Plane Wall tutorial

```
tar xzf planeWall2D.tar.gz
cd planeWall2D
```

Plane Wall tutorial - Allrun

```
runApplication blockMesh
runApplication topoSet
runApplication splitMeshRegions -cellZones -overwrite

remove fluid fields from solid regions
remove solid fields from fluid regions

for i in bottomAir topAir wall
do
   changeDictionary -region $i > log.changeDictionary.$i
done
runApplication chtMultiRegionSimpleFoam
paraFoam -touchAll
```

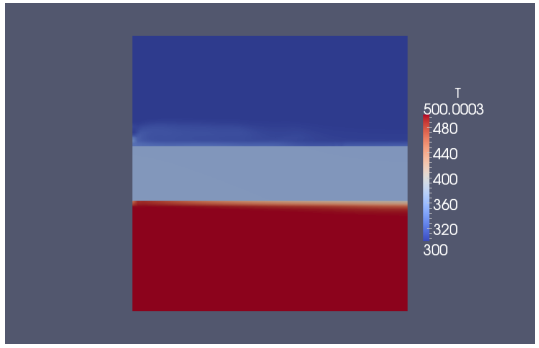
Plane Wall tutorial

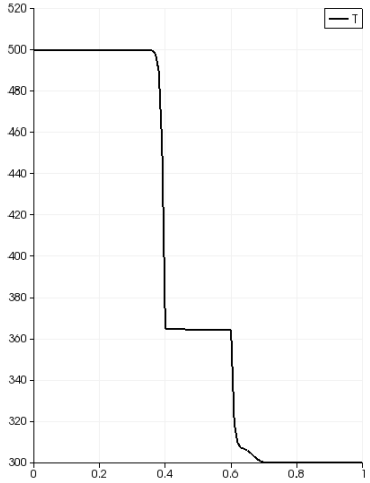
```
log.chtMultiRegionSimpleFoam

planeWall2D{bottomAir}.OpenFOAM
planeWall2D{topAir}.OpenFOAM
planeWall2D{wall}.OpenFOAM

paraview
GroupDatasets
PlotOverLine
```

Plane Wall tutorial





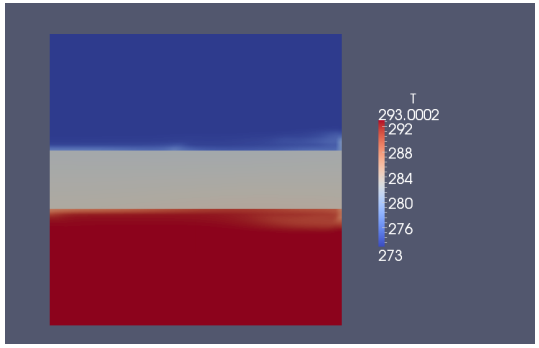
Plane Wall tutorial

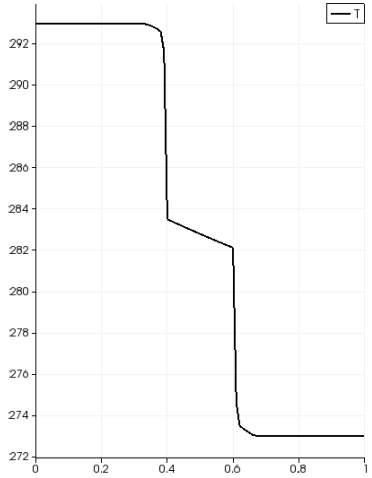
topAir 0.6 to 1m - where the air is colder at 273K and flows from the left to the right at 1m/s and at 4m/s. The top patch is a symmetry plane.

wall 0.4 to 0.6m - where a solid wall is placed, initiated at 273K and has a thermal conductivity of 5 W/mK, density of 2400 kg/m3 and thermal capacity of 880 J/kgK (average for concrete).

bottomAir 0 to 0.4m - where the air is hot at 293K and flows from the left to the right at 1m/s and at 4m/s. The bottom patch is a symmetry plane.

Plane Wall tutorial





chtMultiRegionSimpleBoussinesqFoam