

# Comparison of FEM with FVM applied to multiphysics cases - II

OpenFOAM Finite Volume Solver for Fluid-Solid Interaction (Heat Transfer)

Mesh and OpenFOAM model setup by Roberto Davià

Presented by Raffaele Ponzini, CINECA

07/21

Univerza v Ljubljani



Co-funded by the  
Erasmus+ Programme  
of the European Union

This project has been funded with support from the European Commission.

This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.

- Coffee cup transient analysis: the physical problem
  - The thermal problem
  - The fluid side
  - The solid side
  - Expected physics
- Coffee cup transient analysis: the CFD modelling
  - The geometry
  - The mesh
  - The BC
  - The thermal properties
  - The solver setup
- Coffee cup transient analysis: the data visualization, comparison and animation
  - Visualization of main physical quantities
  - Animation of the time history of the flow field
  - Comparing different meshes

The physical problem

- We want to show a possible coupled thermal application in which we have hot fluid (coffee at 100 Celsius) into a solid cup made of glass at ambient temperature (30 Celsius)
- The fluid is set to be in a starting rest condition with no velocity given from external sources
- We want to see the time evolution of the thermal flux from the hot source (coffee) to the cold container (cup)
- Steady heat conduction is described by a Laplace equation while unsteady conduction is governed by the heat equation
- Heat conduction in a solid needs to be considered along with heat convection in the adjacent fluid: so called conjugate heat transfer problem
- This problem must be solved by iterating between the equations describing the two types of heat transfer

- From the solid side we expect that the thermal flux will give an increasing of the temperature starting from the interface in contact with the fluid side and moving within the internal region of the solid
- How fast this phenomenon will occur is going to be governed by the thermal diffusivity coefficient of the solid that in turn will depend on the material thermal properties (thermal diffusivity)

$$\alpha = \frac{k}{\rho C_p}$$

$\alpha$  = thermal diffusivity

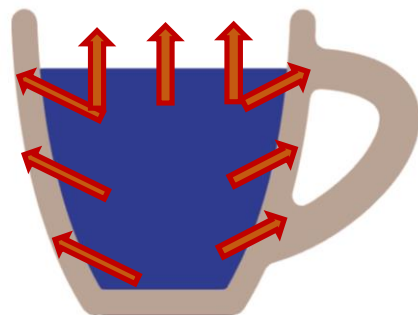
$k$  = thermal conductivity

$C_p$  = specific heat

$\rho$  = density

- In fluids, heat transfer occurs through three mechanisms: conduction, radiation and convection. We will not consider radiation here.
- In flows accompanied by heat transfer, the fluid properties are normally temperature-dependent, such as the density. Variations in density can be the cause of the fluid motion even starting from a 0 velocity condition.
- In such a case, the properties are calculated using the temperature on its current iteration, then the temperature is updated.
- More important is that for these compressible flows, energy and momentum equations are coupled and must be solved simultaneously.

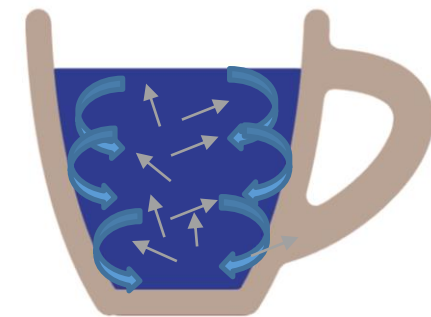
- In our problem we therefore expect that over time:
  - the thermal exchange between fluid and solid will increase the temperature within the solid region
  - changes in temperature in the fluid will affect density changes and thus will bring slow internal motion of the fluid (convective motions)
  - convection currents form because a heated fluid expands when density. The heated fluid are less-dense and rises away from the heat source. As it rises, it pulls cooler fluid down to replace it. This fluid in turn is heated, rises and pulls down more cool fluid (we will check this density changes aspects using an animation)



1. thermal exchange

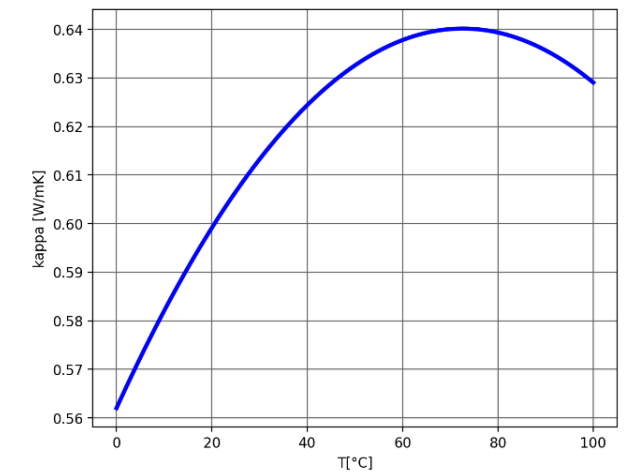
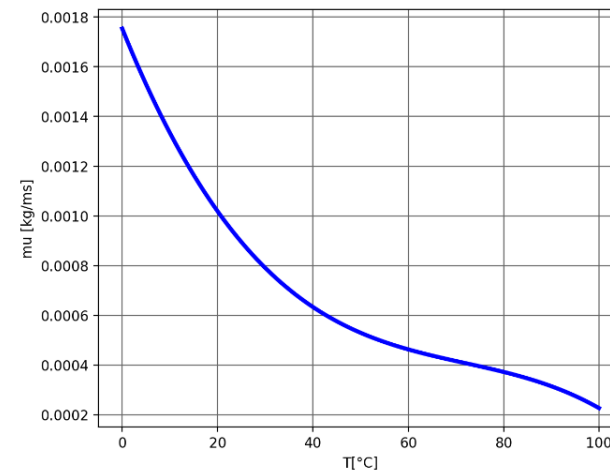
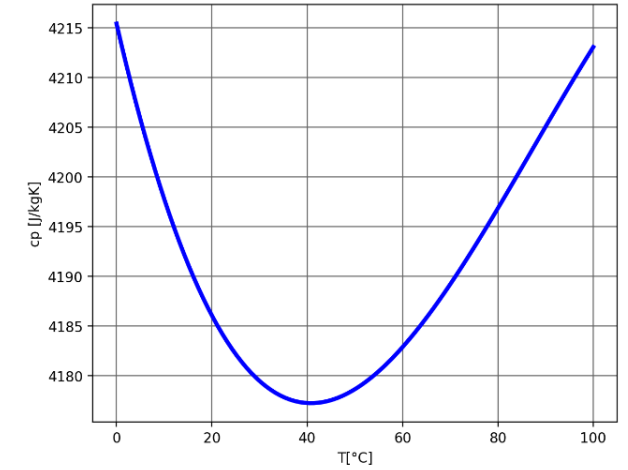
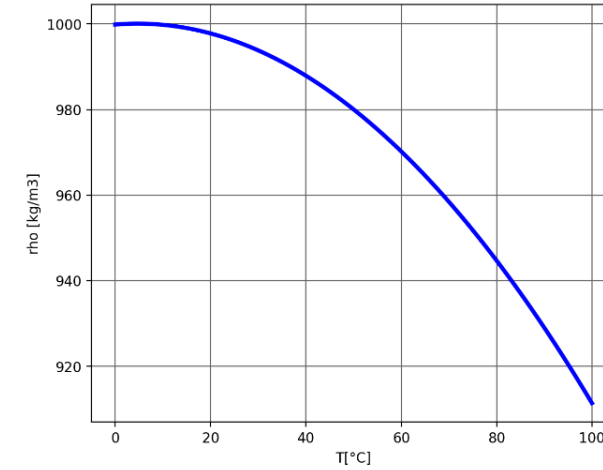
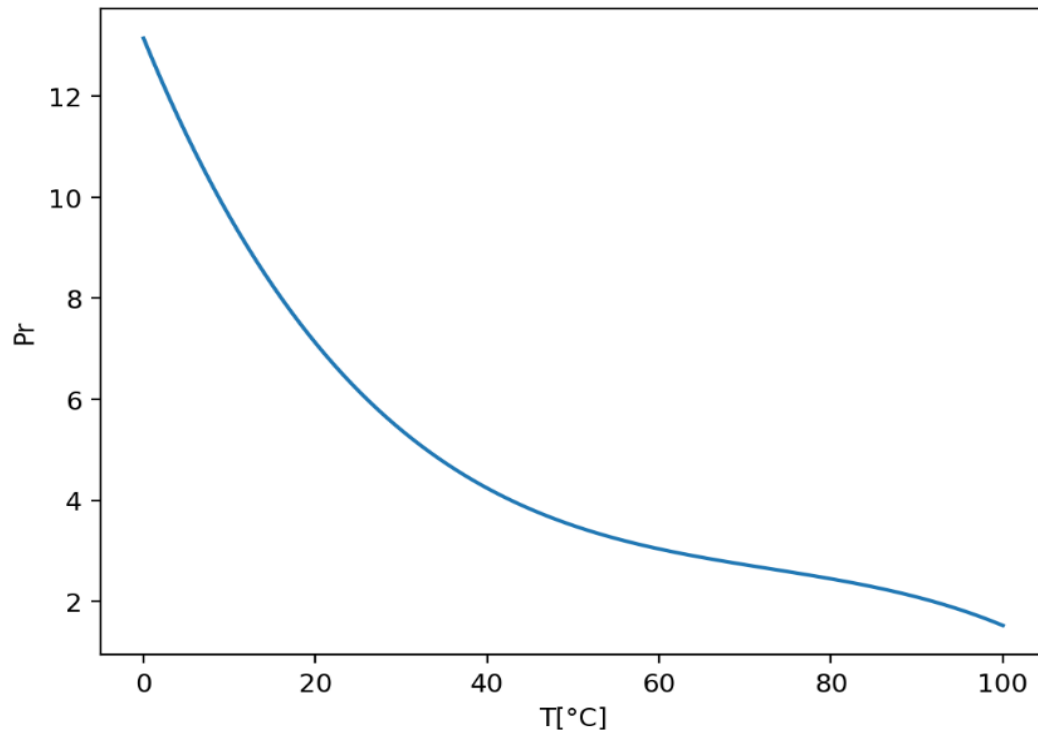


2. Density changes



3. Convective currents

In our problem range of temperature (T) we have a great variability of the physical quantities involved





## The CFD modelling

- In OF there are several thermal solvers:

```
>> ls $FOAM_SOLVERS/heatTransfer
```

- ***buoyantSimpleFoam***: Steady-state, natural convection, compressible (sub-sonic), including radiation
- ***buoyantPimpleFoam***: transient, natural convection, compressible(sub-sonic), including radiation
- ***thermoFoam***: Transient, evolves the thermophysical properties for a frozen velocity field
- ***chtMultiRegionFoam***: Transient, compressible, conjugate heat transfer between solid and fluid

- The geometry is made of a set of 3D watertight surfaces
- The surfaces are divided named and labeled to allow for patch definition into the mesh first and into the CFD model as BC later
- We have two regions (volumes) in this problem: fluid region (coffee) and solid region (cup)



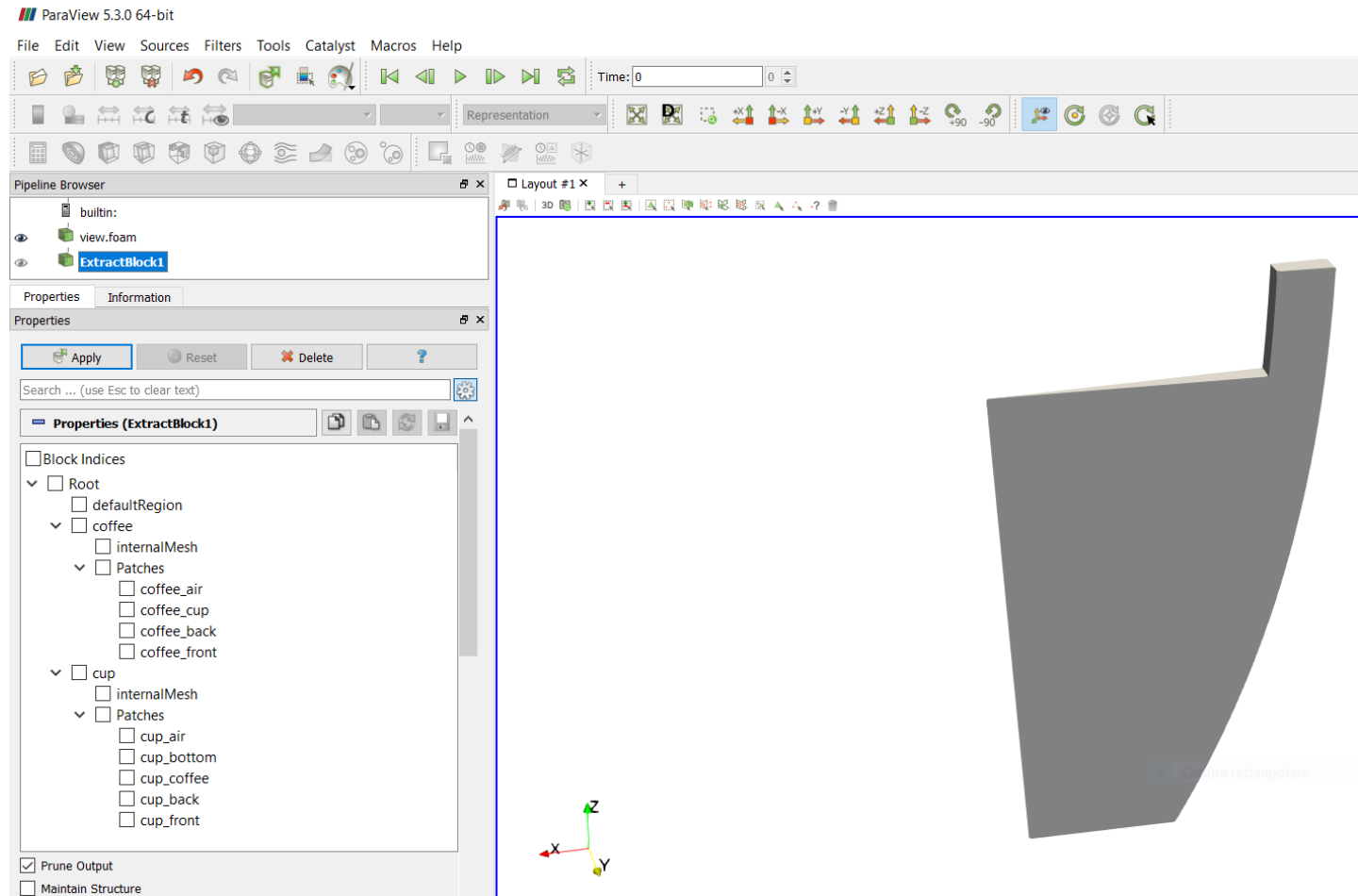
- The coupled problem requires to mesh both the solid and the fluid side of the mesh
- Meshing for coupled thermal problem is crucial in order to capture correctly both the physics of the thermal BL and the velocity BL
- The quality of the mesh in coupled problem is central to avoid solver instability. It is really likely that instability can be caused by bad quality or illegal cells in the mesh.
- For this reason in this guided case we provided the mesh directly.
- You will find the mesh at:

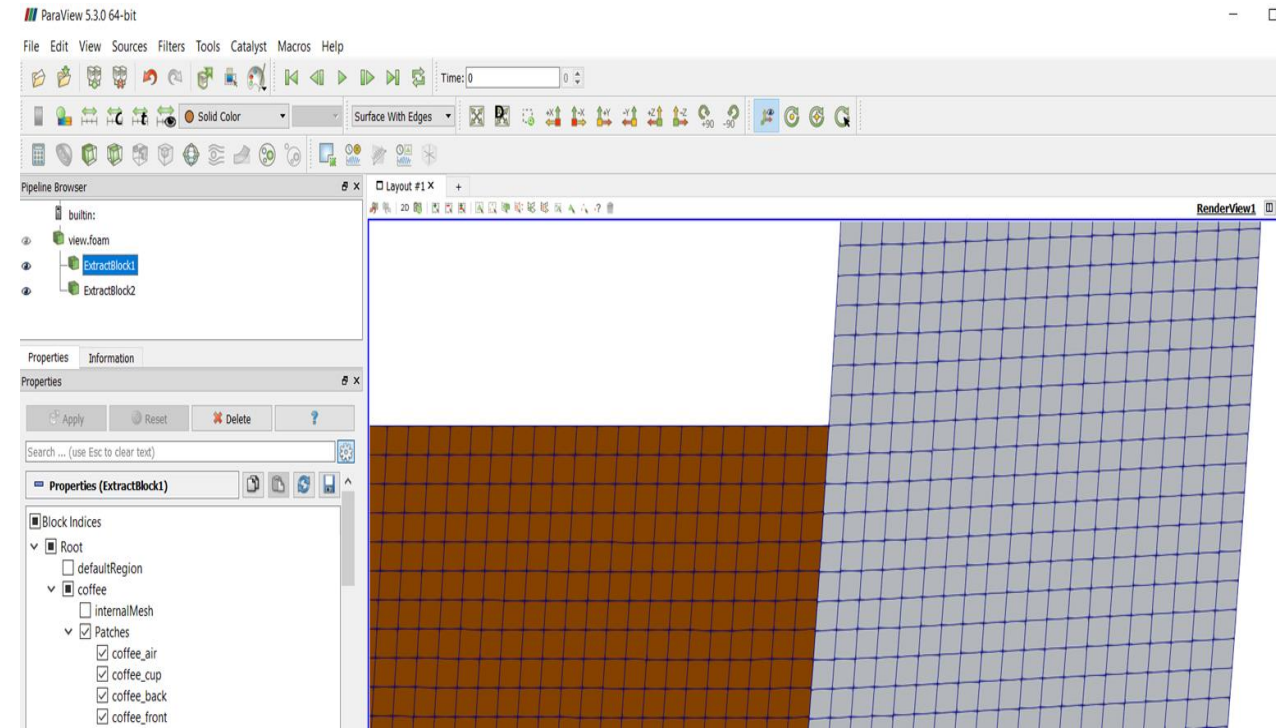
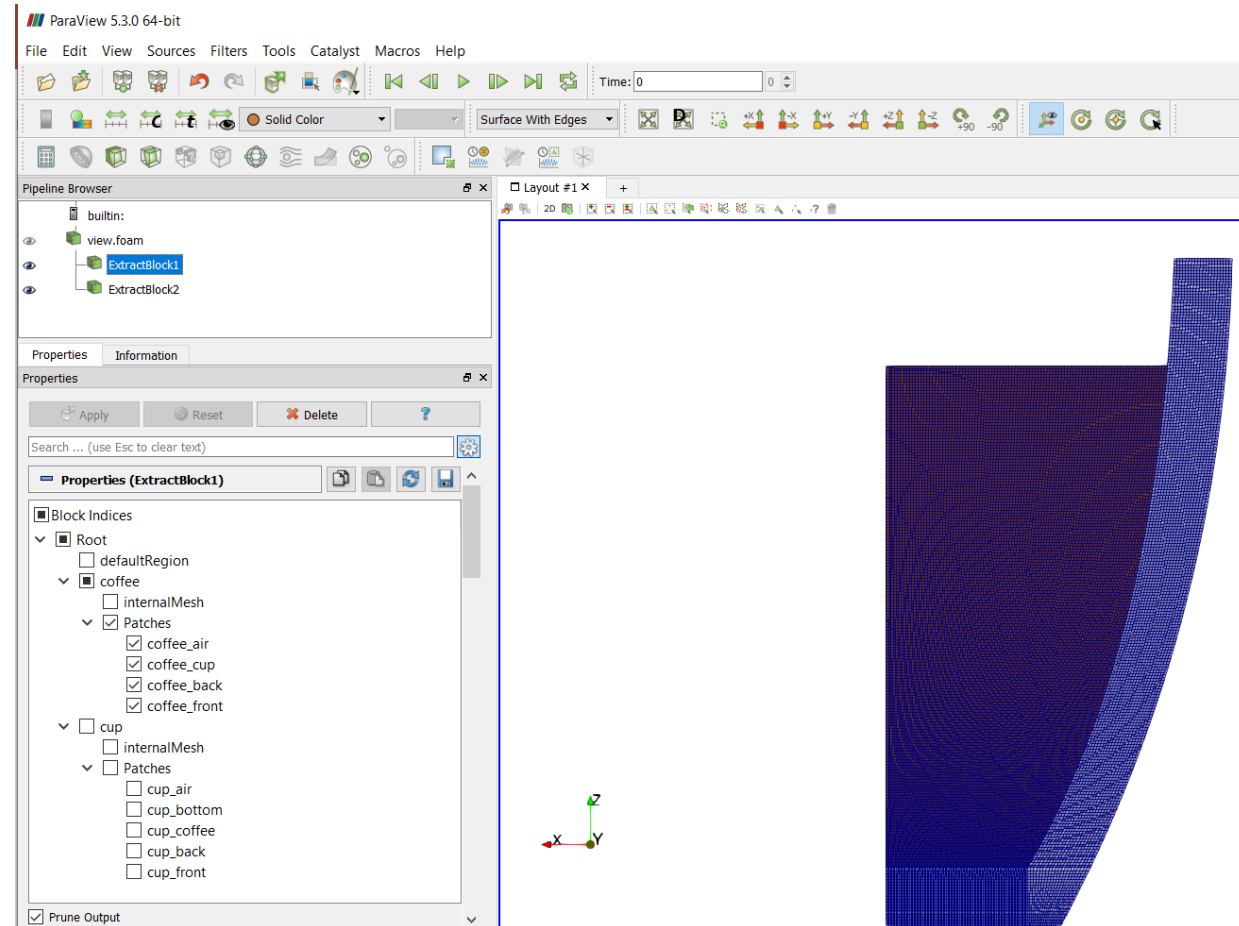
<https://bscw.lecad.fs.uni-lj.si/bscw/bscw.cgi/362134>

- Analysis of the mesh will reveal that:
  - we have 2 regions: coffee and cup
  - it has been built using conformal meshing approach
- Conformal mesh is a mesh where two cells at a given interface are sharing a common element (face or edges)
- This is very convenient when interpolated data are shared like in the case of the coupled thermal solver where  $T$  and  $grad(T)$  are computed and interpolated between the fluid and the solid
- The quality of the mesh regions can be checked using the usual *checkMesh* function with the options *-region*:

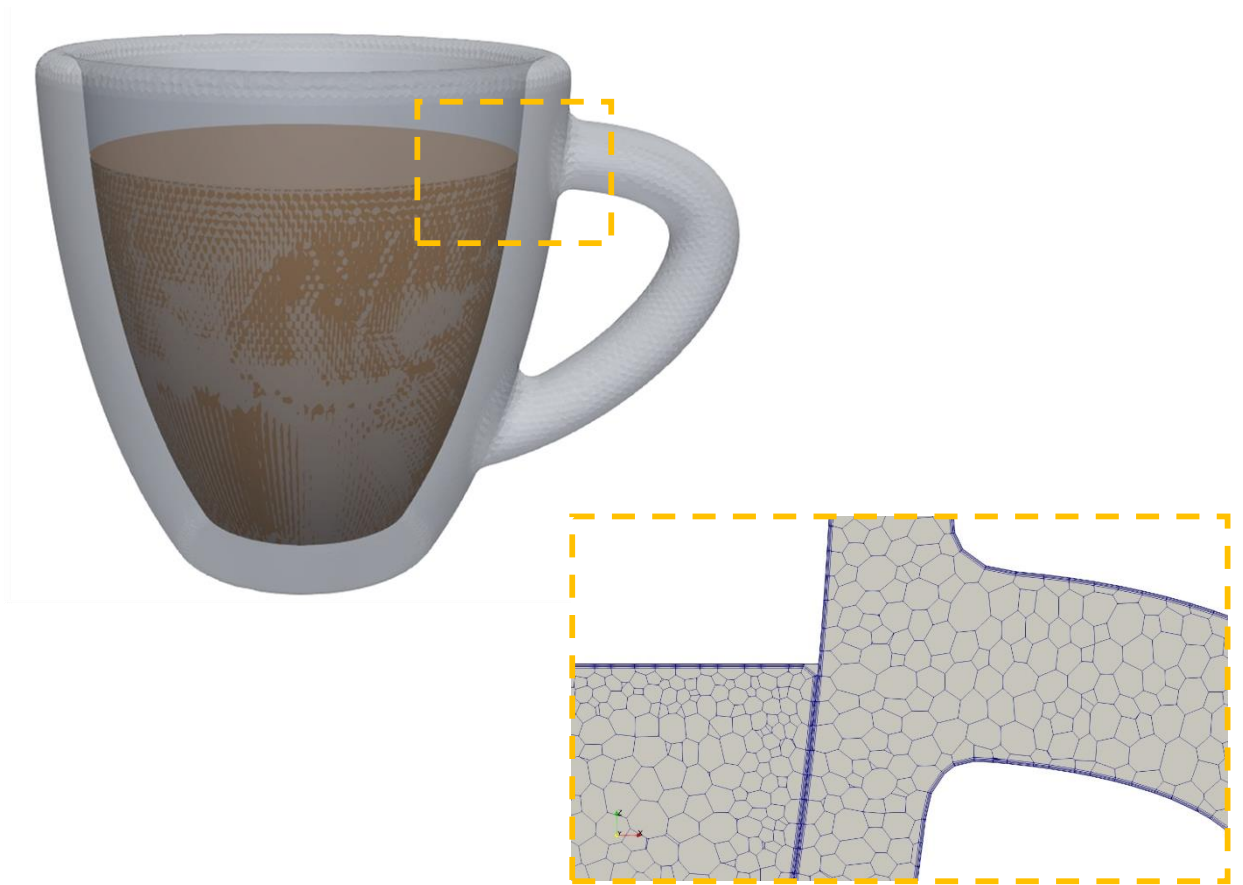
```
>> checkMesh -region coffee
>> checkMesh -region cup
```

- Opening the 2d mesh we can see the two regions
- Single blocks for the two regions can be extracted for further analyses and visualization using the *extractBlock* filter in Paraview or Parafoam as said
- Selecting a visualization with edges the mesh cells can be inspected





- The 3D mesh has been built to have similar behavior at the BL interfaces
- The remaining mesh zones are obtained starting from a tetrahedral mesh converted into polyhedral one
- As for the case of the 2D the mesh has been generated using the NetGen library from Salome GUI
- Netgen mesher allows for tetrahedral/quad/prism for 2D and 3D geometries: <https://ngsolve.org/>





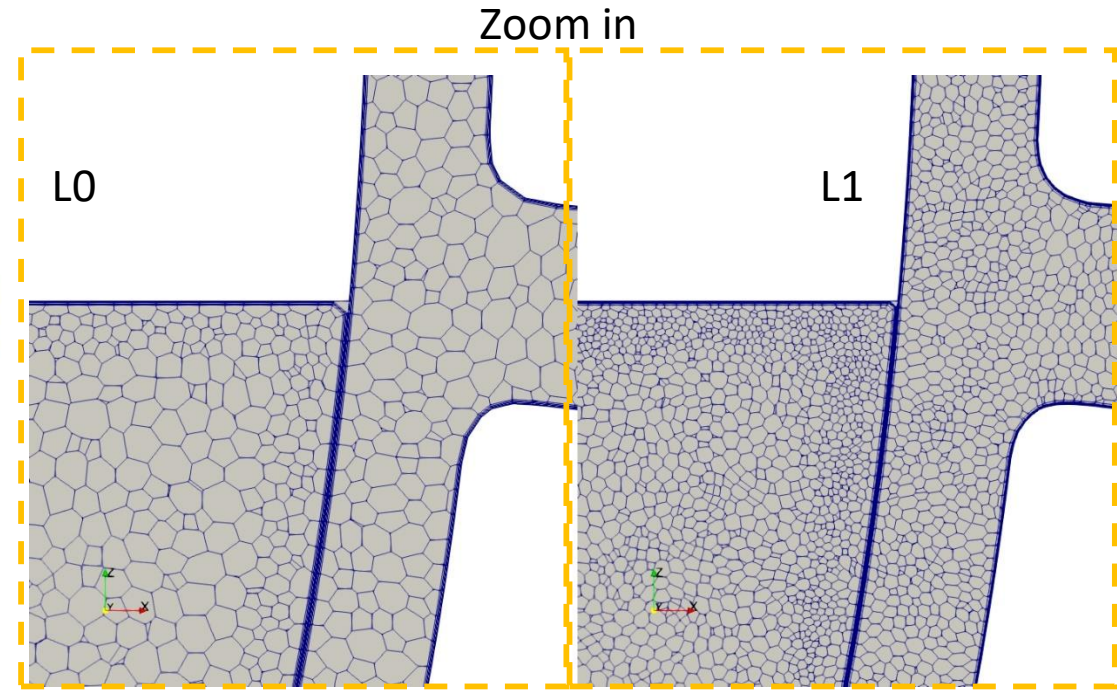
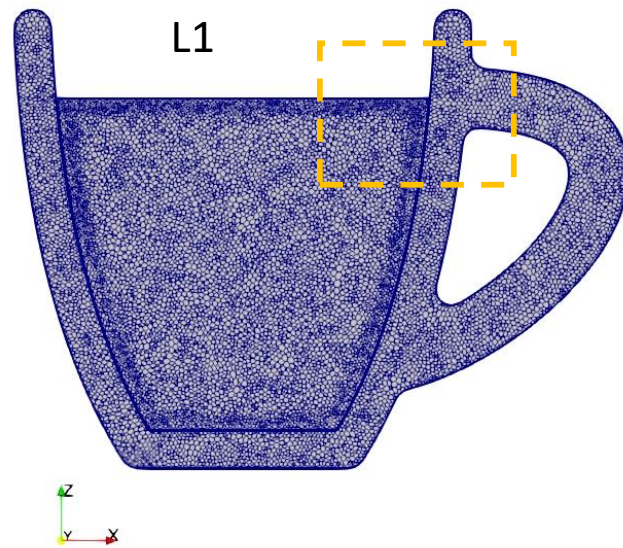
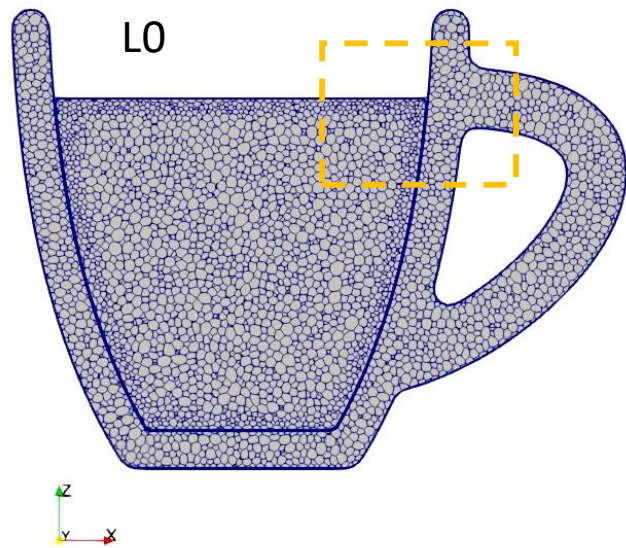
- In synthesis we have planned to analyze different meshes with the same solver setup to see to which extent meshing changes will affect solution differences
- This point is one of the main practical issue for engineers using CFD tools (mesh sensitivity)
- The BL has been added to all meshes by means of a practical approach (trial/modify) in order to perform in a meaningful way at the solid/fluid interface from a physical point of view

Mesh name	Avg cell edge [m]	Total num cells [-]	Num cell solid [-]	Num cell fluid [-]	Num BL layers [-]
3D_L0	0,0016	247,822	135,377	112,445	4
3D_L1	0,0008	1,542,022	778,475	763,547	4

Note: the meshes are obtained converting from tetrahedral mesh into polyhedral meshes using the *polyDualMesh* function in OpenFOAM:

```
>> polyDualMesh 130
```

# Comparing L0 and L1 meshes



- Using the function patchSummary we have:

```
>> patchSummary -region cup
```

```
wall      : cup_air
```

```
  scalar          p          calculated  
  scalar          T          externalWallHeatFluxTemperature
```

```
mappedWall      : cup_coffee
```

```
  scalar          p          calculated  
  scalar          T          compressible::turbulentTemperatureCoupledBaffleMixed
```

```
wall      : cup_ground
```

```
  scalar          p          calculated  
  scalar          T          zeroGradient
```

- For the solid we have set the BC so that:
  - Cup and air can exchange a heat flux due to temperature difference between the cup and the air
  - Cup and coffee can exchange a conduction of temperature due to their different values (100/30 Celsius)
  - Cup bottom cannot exchange form a thermal point of view with the ground (adiabatic)

Patch name	p	T
Interface cup/air	calculated	externalWallHeatFluxTemperature
Interface cup/coffee	calculated	turbulentTemperatureCoupledBaffleMixed
Interface cup/bottom	calculated	zeroGradient

```
>> patchSummary -region coffee
```

```
wall      : coffee_air
```

```
  scalar      p_rgh      totalPressure  
  scalar      p          calculated  
  scalar      T          externalWallHeatFluxTemperature  
  vector      U          fixedValue
```

```
mappedWall : coffee_cup
```

```
  scalar      p_rgh      fixedFluxPressure  
  scalar      p          calculated  
  scalar      T          compressible::turbulentTemperatureCoupledBaffleMixed  
  vector      U          noSlip
```

- For the fluid we have set the BC so that:
  - Coffee and air can exchange a heat flux due to temperature difference between the coffee and the air
  - Coffee and cup can exchange a conduction of temperature due to their different values (100/30 Celsius)
  - Coffee is at 0 velocity at the beginning and with atmospheric pressure value

Patch name	P_rgh	T	U
Interface coffe/air	totalPressure	externalWallHeatFluxT emperature	0
Interface coffe/cup	fixedFluxPressure	turbulentTemperatureC oupledBaffleMixed	No slip

- For the thermal properties of the fluid side we set as for **hot water**:

```
constant/coffee/thermophysicalProperties
```

```
thermoType
{
    // Water
    specie
    {
        nMoles      1;
        molWeight   28.9;
    }
    type           heRhoThermo;
    mixture        pureMixture;
    transport      polynomial;
    thermo         hPolynomial;
    equationOfState icoPolynomial;
    specie         specie;
    energy         sensibleEnthalpy;
    thermodynamics
    {
        CpCoeffs<8> (9850.69 -48.6714 0.13736 0.000127063 0 0 0 0);
        Hf          0;
        Sf          0;
    }
    // pRef          1e5;
    dpdt off;
    transport
    {
        muCoeffs<8> (0.116947 -0.00100532 2.90283e-6 -2.80572e-9 0 0 0 0);
        kappaCoeffs<8> (-1.12925 0.0102391 -1.48128e-5 0 0 0 0 0);
    }
}
}
```



- For the thermal properties of the solid side we set as for **glass**:

```
constant/cup/thermophysicalProperties
{
  thermoType
  {
    type          heSolidThermo;
    mixture       pureMixture;
    transport     constIso;
    thermo        eConst;
    equationOfState rhoConst;
    specie        specie;
    energy        sensibleInternalEnergy;
  }
}

mixture
{
  specie
  {
    nMoles      1;
    molWeight   20;
  }
  transport
  {
    kappa      1.38;
  }
}

thermodynamics
{
  Hf      0;
  Cv      703;
}

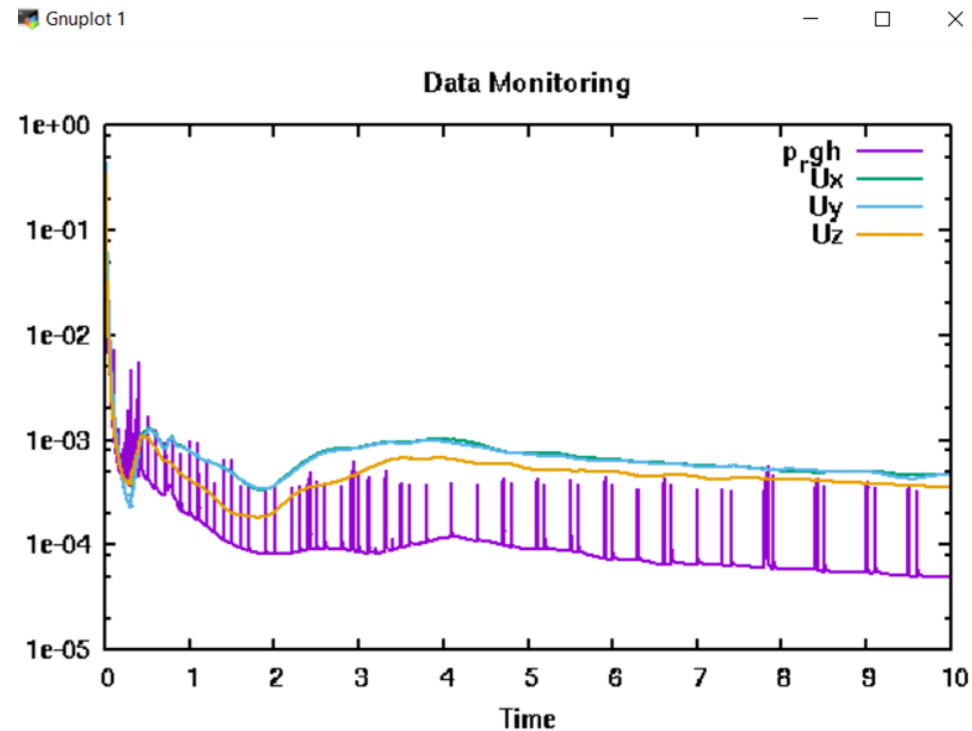
equationOfState
{
  rho      2203;
}
```

- The solver we selected is the *chtMultiRegionFoam*
- The time step size is set at 1.0E-04 and the end time is set at 10s.
- We added functions to monitor residuals, thermal probes and fluid flow 2d slices
- The coupling between the solid and the fluid regions is made by the BC *compressible::turbulentTemperatureCoupledBaffleMixed*;

The data analysis and visualization

- As said the first analysis we can do is the residual monitoring:

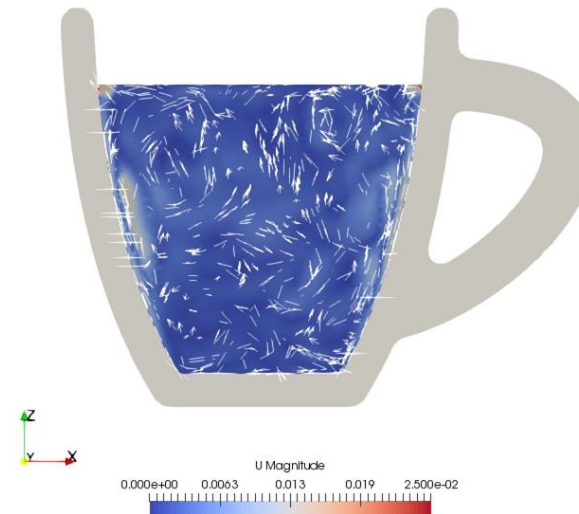
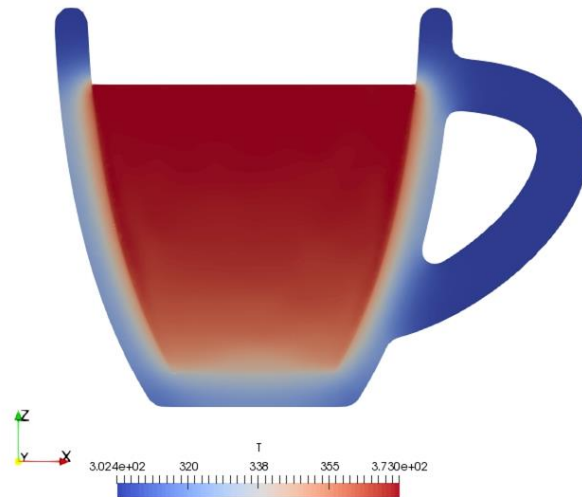
```
>> foamMonitor -l postProcessing/cup/coffeeResiduals/0/residuals.dat
```



- We don't see any particular problematic pattern
- We can assume that the numeric is doing ok
- We can start looking at the physics

- To have an understanding of the solver outcomes we can plot U and T distribution on a y-normal slice for the L0 mesh after 10s
- From our a-priori knowledge of the physical problem we can try to check and verify some founding

Diffusion of T from the fluid to the solid: ok that's actually what we **are** looking for



Very slow phenomenon in U: ok makes sense with the given physics

- The model setup allows for solving the desired physical problem as a first approximation
- What we should look at in our simulation and why
- One possible question could be what is temperature evolution value after 10s in some given point in the solid and/or in the fluid
- We can add probes monitors using the *foamGet* function:  

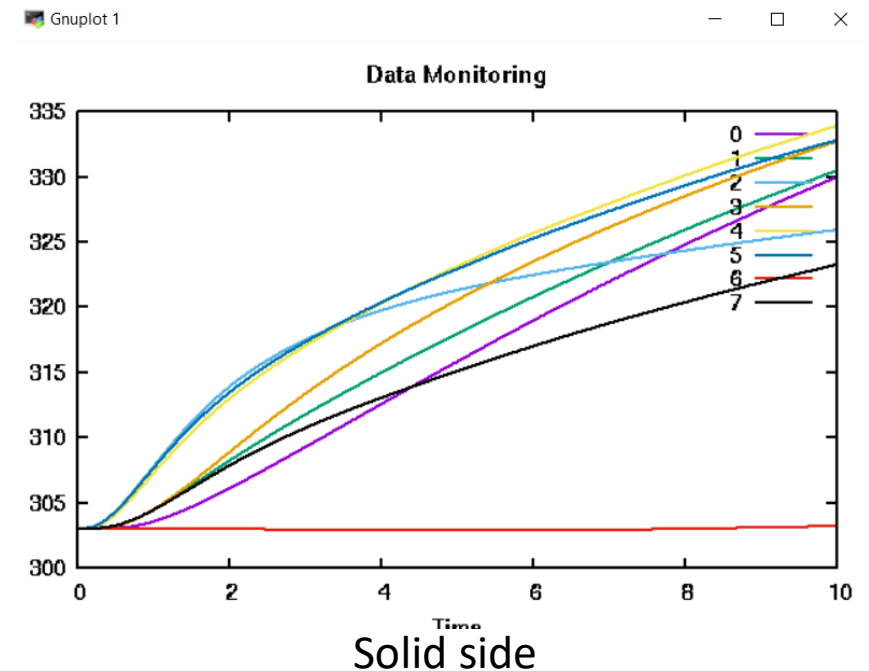
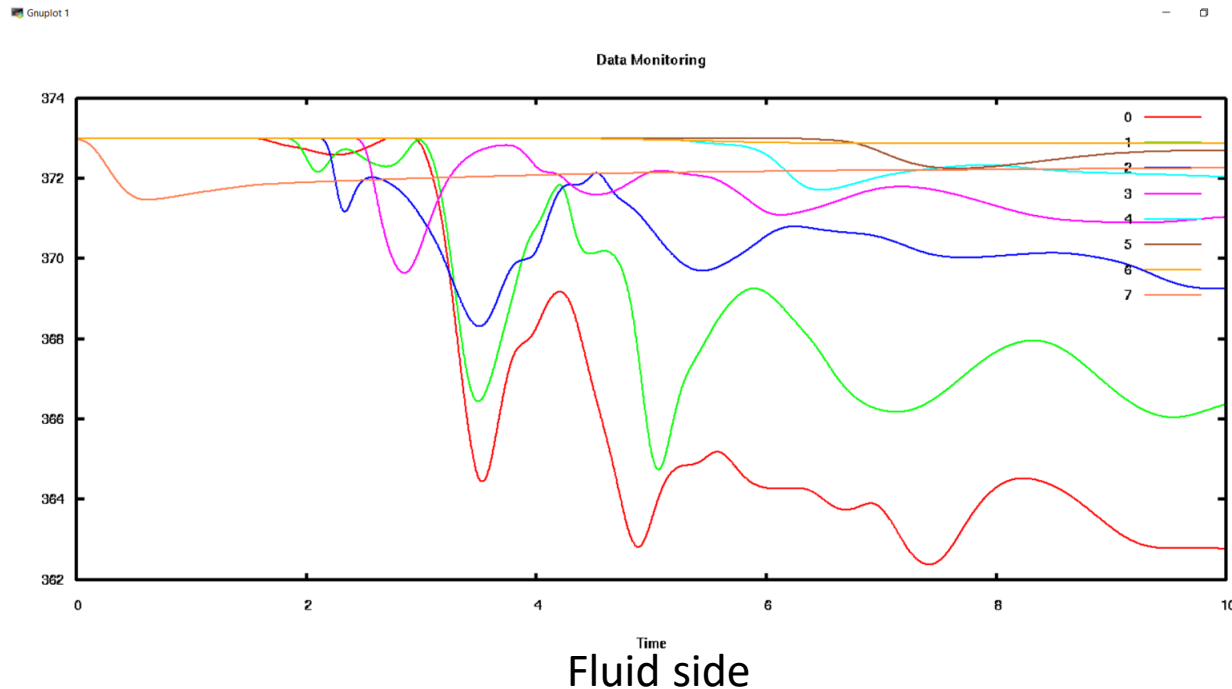
```
>> foamGet probes
```

And then we can specify selected points in the desire region of the domain and selected quantities (T)

```
fields (T);  
region coffee;  
probeLocations  
(  
    (-0.012  0  0.00)  
    (-0.012  0  0.005)  
    (-0.012  0  0.01)  
    (-0.012  0  0.015)  
    (-0.012  0  0.02)  
    (-0.012  0  0.025)  
    (-0.012  0  0.03)  
    (-0.012  0  0.035)  
);
```

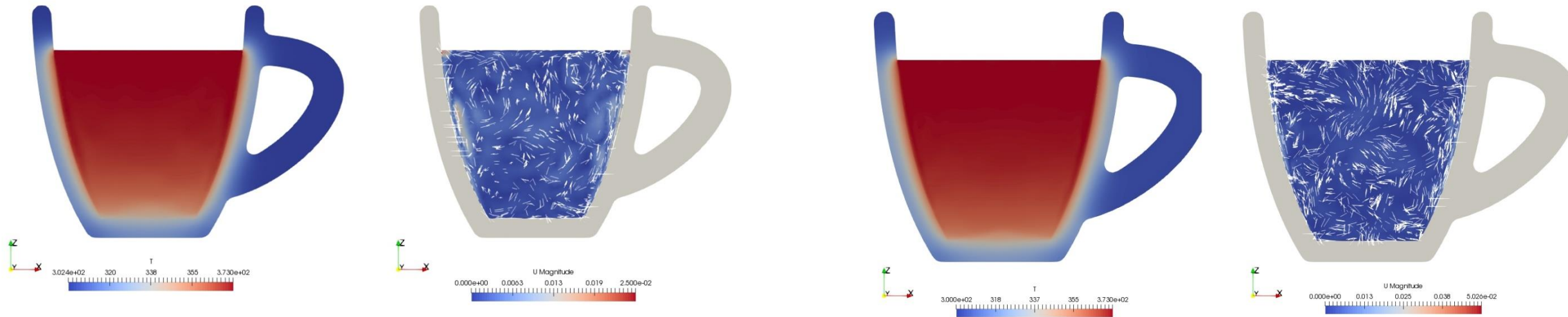
```
fields (T);  
region cup;  
probeLocations  
(  
    (-0.025814  0  0.0228594 )  
    (-0.0235098  0  0.0118817 )  
    (-0.00698106  0 -0.0113663 )  
    (-0.0265283  0  0.0318013 )  
    (-0.0211965  0  0.00579988)  
    (-0.0187883  0 -0.00065686)  
    (-0.0273956  0  0.0431362 )  
    (-0.0157162  0 -0.00730431)  
);
```

- To get more quantitative understanding of the temperature patterns we can plot the point probes files as set during calculation using the *foamMonitor* function





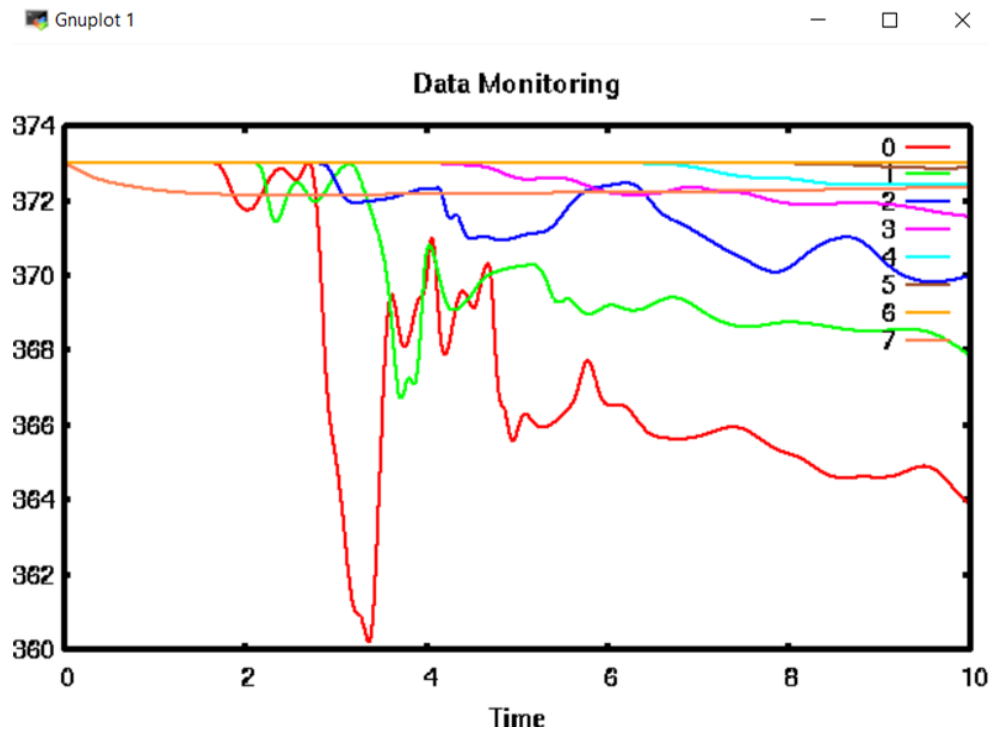
- To have an understanding of different meshing outcomes we can plot U and T distribution for the L1 mesh after 10s and compared it to L0 mesh



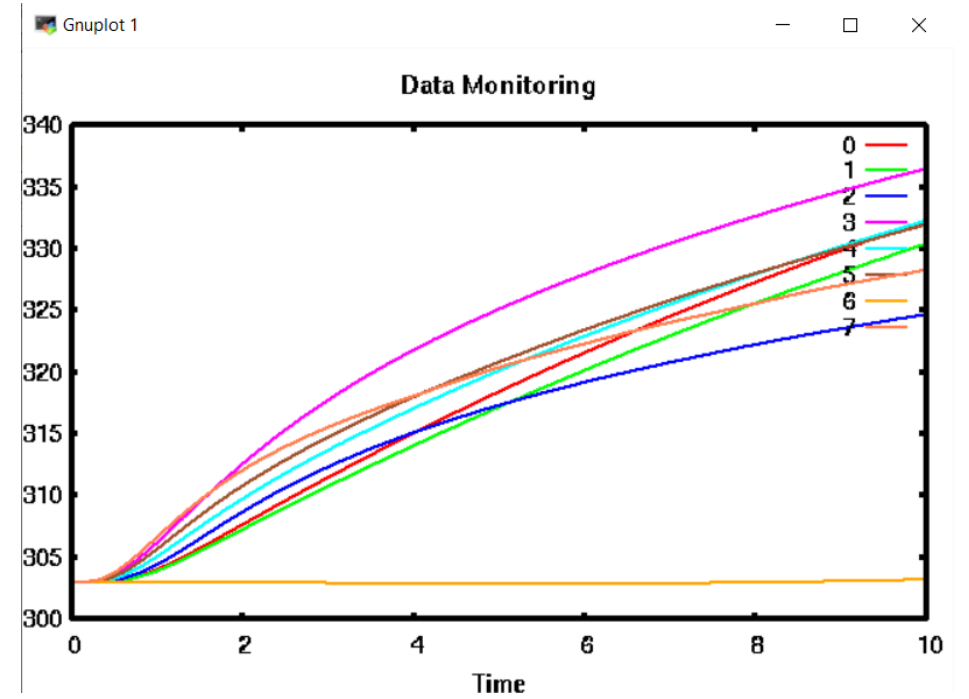
L0 mesh after 10s

L1 mesh after 10s

- Again we can plot the point T probes for the fluid and the solid part



Fluid side



Solid side



Animation over time colored by  $T$  in the solid and by  $\rho$  in the fluid for the different meshes with the same solver setup

- We are looking at very different meshes in size and we are getting qualitatively very similar results
- The reason behind that is the powerful support of the *compressible::turbulentTemperatureCoupledBaffleMixed* BC
- Treating the BL in a proper way, i.e. using a meaningful number of layers and thickness, we are taking full advantage of a modelling strategy that enables to limit the total cell count without loosing in quality of the solution for the average quantities
- The coffee cup case is a toy case where we don't have any reference measures so we can only look at the qualitative average results
- Other quantitative comparative analysis makes no sense here

The students can play with the proposed model by:

1. Verify more in details deltas in T values for the different probes in mesh L0 and L1 (data analysis for mesh sensitivity)
2. Add some probe in the cup handle to check if after a given time we are getting over-heated (design oriented analysis)
3. Quantify what is the maximum temperature in the solid and where over time (design oriented analysis)
4. Changing the reference temperatures of the fluid and or the solid (design oriented analysis)
5. Changing the material properties of the solid (design oriented analysis)

- The case setup has been developed by Roberto Davià starting from a public case:  
<https://holzmann-cfd.com/community/training-cases/gin-tonic>
- An alternative approach would be start from the templates/ as defined in the OpenFOAM-v8.0 release in:  
*\$FOAM\_ETC/templates/singleFluidCHT*  
and follow the README file included

Thank you for your attention!

<http://sctrain.eu/>

Univerza v Ljubljani



TECHNISCHE  
UNIVERSITÄT  
WIEN



VSB TECHNICAL  
UNIVERSITY  
OF OSTRAVA

IT4INNOVATIONS  
NATIONAL SUPERCOMPUTING  
CENTER



Co-funded by the  
Erasmus+ Programme  
of the European Union

This project has been funded with support from the European Commission.

This publication [communication] reflects the views only of the author, and the Commission cannot be held responsible for any use which may be made of the information contained therein.