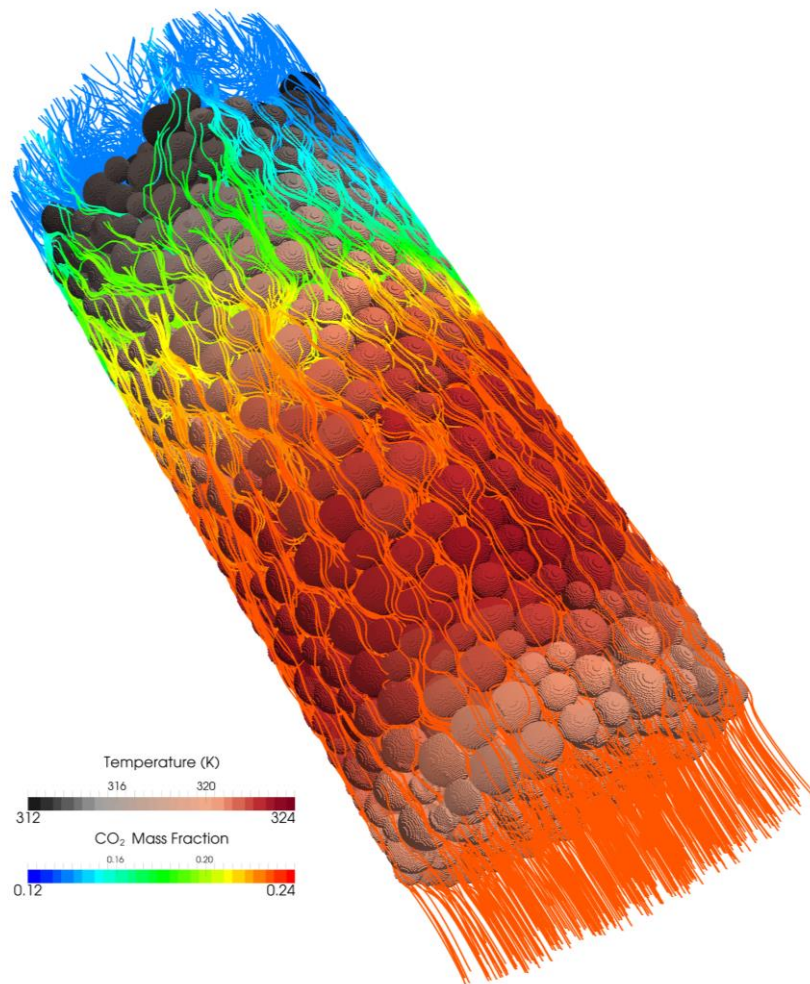


## OpenFOAM® Basic Training

### Tutorial Six



3<sup>rd</sup> edition, Feb. 2015



This offering is not approved or endorsed by ESI® Group, ESI-OpenCFD® or the OpenFOAM® Foundation, the producer of the OpenFOAM® software and owner of the OpenFOAM® trademark.

Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>


Editors and Contributors:

- Bahram Haddadi (TU Wien)
- Christian Jordan (TU Wien)
- Jozsef Nagy (JKU Linz)
- Clemens Gößnitzer (TU Wien)
- Vikram Natarajan (TU Wien)
- Sylvia Zibuschka (TU Wien)
- Michael Harasek (TU Wien)



Cover picture from:

- Bahram Haddadi, The image presented on the cover page has been prepared using the Vienna Scientific Cluster (VSC).

 Except where otherwise noted, this work is licensed under <http://creativecommons.org/licenses/by-nc-sa/3.0/>

Attribution–NonCommercial–ShareAlike 3.0 Unported (CC BY–NC–SA 3.0)

This is a human-readable summary of the Legal Code (the full license).

Disclaimer

You are free:

- to Share — to copy, distribute and transmit the work
- to Remix — to adapt the work

Under the following conditions:

Attribution — You must attribute the work in the manner specified by the author or licensor (but not in any way that suggests that they endorse you or your use of the work).

Noncommercial — You may not use this work for commercial purposes.

Share Alike — If you alter, transform, or build upon this work, you may distribute the resulting work only under the same or similar license to this one.

With the understanding that:

Waiver — Any of the above conditions can be waived if you get permission from the copyright holder.

Public Domain — Where the work or any of its elements is in the public domain under applicable law, that status is in no way affected by the license.

Other Rights — In no way are any of the following rights affected by the license:

Your fair dealing or fair use rights, or other applicable copyright exceptions and limitations;

The author's moral rights;

Rights other persons may have either in the work itself or in how the work is used, such as publicity or privacy rights.

Notice — For any reuse or distribution, you must make clear to others the license terms of this work. The best way to do this is with a link to this web page.

## simpleFoam – pitzDaily (turbulence, steady state)

### Simulation

Use simpleFoam solver, run a steady state simulation with following turbulence models:

- kEpsilon (RAS)
- kOmega (RAS)
- LRR (RAS)

### Objectives

- Understanding turbulence modeling
- Understanding steady state simulation

### Post processing

Show the results of U and the turbulent viscosity in two separate contour plots.

## Step by step simulation

### Copy tutorial

```
~/OpenFOAM/OpenFOAM-2.3.0/tutorials/incompressible/simpleFoam
```

```
/pitzDaily
```

### 0 directory

When a turbulent model is chosen, the value of its constants and its boundary values should be set in the appropriate files, for example in kEpsilon model the k and epsilon files should be edited. e.g. epsilon:

```
// * * * * * //
dimensions      [0 2 -3 0 0 0 0];
internalField   uniform 14.855;
boundaryField
{
  inlet
  {
    type          fixedValue;
    value         uniform 14.855;
  }
  outlet
  {
    type          zeroGradient;
  }
  upperWall
  {
    type          epsilonWallFunction;
    value         uniform 14.855;
  }
  lowerWall
  {
    type          epsilonWallFunction;
    value         uniform 14.855;
  }
  frontAndBack
  {
    type          empty;
  }
}
// ***** //
```

*Note:* Here is a list of files which should be available at 0 directory and need to be modified for each turbulence model:

- *laminar: no file*
- *kEpsilon (RAS): k and epsilon*
- *kOmega (RAS): k and omega*
- *LRR (RAS): k, epsilon and R*
- *Smagorinsky (LES): nuSgs*
- *oneEqEddy (LES): k and nuSgs*

- *SpalartAllmaras (LES): nuSgs and nuTilda*

Some files are available, e.g. epsilon, k and nuTilda, some files should be created by the user, e.g. R, omega. Templates for these files can be also found in the examples of older versions of OpenFOAM®, e.g. 1.7.1.

*Note: Missing R file can be created by OpenFOAM®. In the constant directory in RASProperties file set the RASModel to kEpsilon. The turbulenceProperties file is also needed. Copy it from another tutorial and set simulationType to RASModel in the file. Run the command “R” from terminal, it will create the R file in the 0 directory.*

### **constant directory**

The type of simulation turbulence model is set in turbulenceProperties file, e.g. it is a RASModel or LESModel (this file is not available in this tutorial, but can be copied from other tutorials). For choosing a specific turbulence model the RASProperties file should be checked (e.g. here kEpsilon).

```
// * * * * *
RASModel          kEpsilon;

turbulence        on;

printCoeffs       on;

// *****
```

*Note: For the laminar model both turbulenceProperties and RASProperties should be set to laminar. In the RASProperties set turbulence and also printCoeffs to off.*

### **system directory**

*Note: Since it is a steady state simulation controlDict endTime show the number of iterations instead of time and deltaT should be 1, because it is the amount of increase in the iteration number.*

### **Running simulation**

```
>blockMesh

>simpleFoam
```

*Note: When the solution converges, “SIMPLE solution converged in ... iterations” message will be displayed in the Shell window. If nothing happens and you see no message after a while (it is not the case in here, it converges after a short time), then you should check the residuals which are displayed in the Shell window manually (you should check initial residual values, it shows the difference between this iteration and the last one), if all of the Initial residual (see below) values are close to amounts you have set in the fvSolution then you can stop simulation (ctrl+c).*

Time = 817

```
smoothSolver: Solving for Ux, Initial residual = 0.00013826, Final residual =
9.87886e-06, No Iterations 2
smoothSolver: Solving for Uy, Initial residual = 0.000994709, Final residual =
7.317e-05, No Iterations 2
GAMG: Solving for p, Initial residual = 0.00192871, Final residual =
0.000174838, No Iterations 7
time step continuity errors : sum local = 0.000840075, global = 6.13868e-05,
cumulative = -0.193739
smoothSolver: Solving for epsilon, Initial residual = 0.000175322, Final
residual = 1.138e-05, No Iterations 2
smoothSolver: Solving for k, Initial residual = 0.000404928, Final residual =
2.99083e-05, No Iterations 2
ExecutionTime = 20.11 s ClockTime = 20 s
```

SIMPLE solution converged in 817 iterations

### Exporting simulation

The simulation results are as follows:

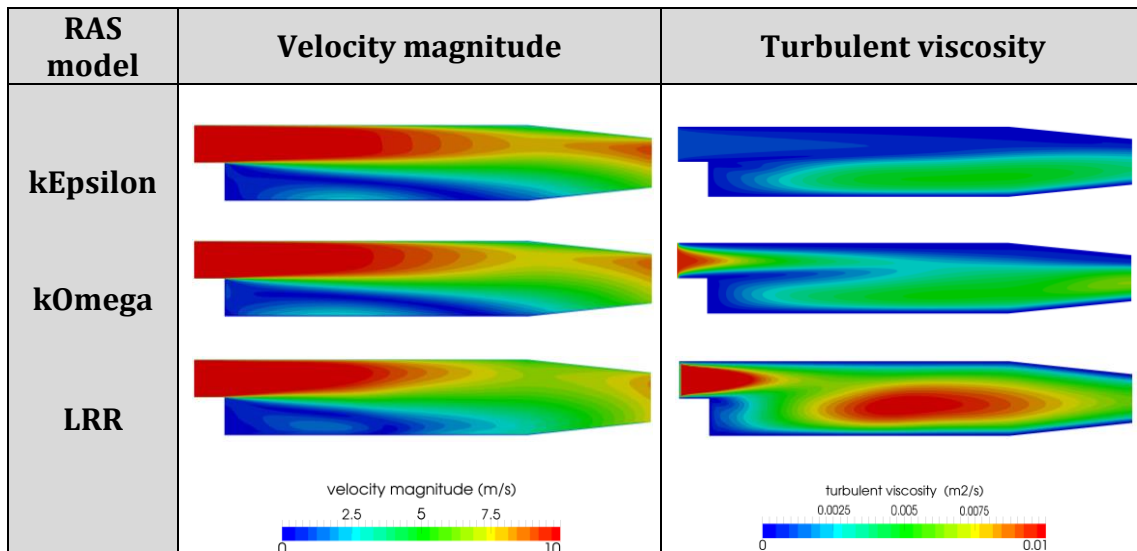


Figure 6.1 Comparison of different turbulent models at steady state