A brief introduction to dieselFoam

by

Per Carlsson

Agenda

- What is dieselFoam
- dieselFoam Tutorial; geometry, boundary conditions and chemistry
- Running the dieselFoam solver
- Post processing i ParaView

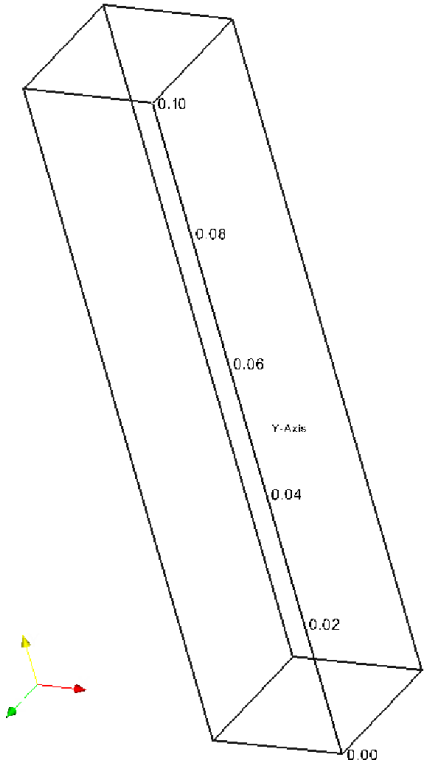
What is dieselFoam

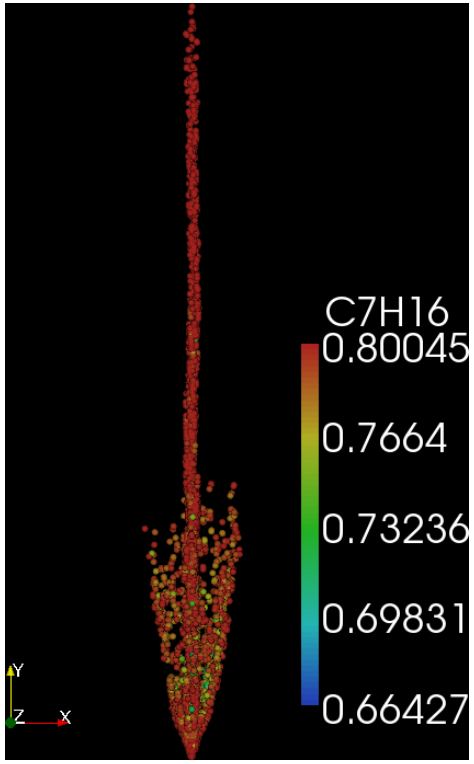
dieselFoam is a solver for:

- Two phase
- Compressible
- Turbulent
- Reacting
- Euler-Lagrangian particle tracking
- Transient-

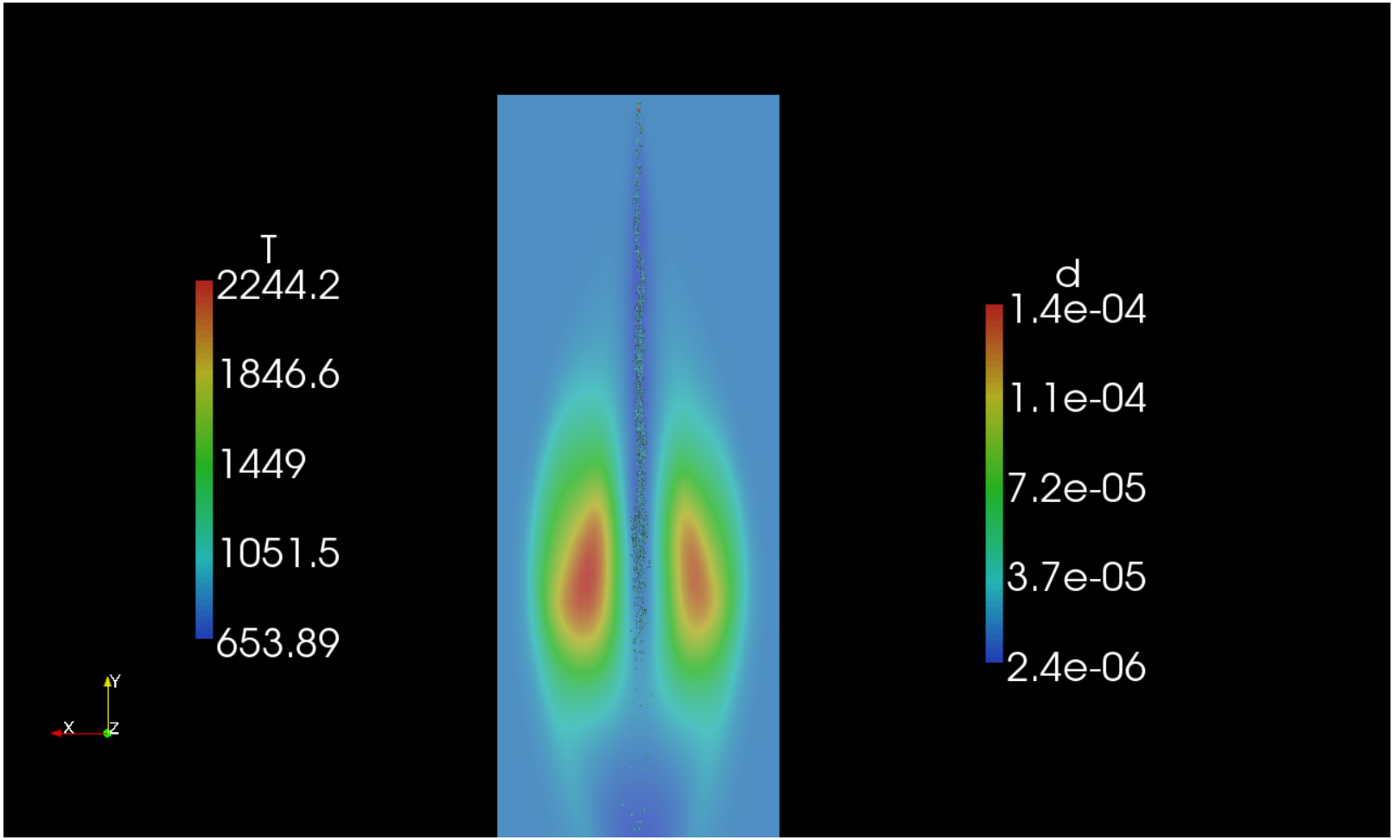
flow

dieselFoam tutorial, Introduction





dieselFoam tutorial, Introduction



Directories

Copy the tutorial.

```
cp -r  $FOAM_TUTORIALS/dieselFoam/aachenBomb $FOAM_RUN
cd $FOAM_RUN/aachenBomb
```

The case directoris

```
/0
/chemkin
/constant
/constant/polyMesh
/system
```

Constant directory

Properties file	General content
chemistryProperties	Chemistry on/off and schemes used to solve chemistry
environmentalProperties	Gravity
combustionProperties	Ignition point on or off, timing and duration of ignition point
thermophysicalProperties	Specify the mixture type and which gas phase reaction scheme to use as well as thermodynamic database
injectorProperties	Location and boundary conditions for the injector
sprayProperties	Spray submodels

Injector properties

```
    injectorType          unitInjector;
    unitInjectorProps
    {   position          (0 0.0995 0);
        direction         (0 -1 0);
        diameter          0.00019;
        Cd                0.9;
        mass              6e-06;
        temperature       320;
        nParcels          5000;
        X
        (
            1.0
        );
massFlowRateProfile
        (
            (0 0.1272)
            (4.16667e-05 6.1634)
            (8.33333e-05 9.4778)....
```

Spray properties

Model	General meaning
atomizationModel	How atomization is treated
breakupModel	If secondary break up is used
injectorModel	Which injector model to use
collisionModel	Particle - particle interaction
evaporationModel	Which evaporation model to use
heatTransferModel	Particle heat transfer model
dispersionModel	If turbulent dispersion is used or not
dragModel	Particle drag model
wallModel	What happens to particles hitting the walls

Chemistry; thermophysicalProperties

```
thermoType hMixtureThermo<reactingMixture>;

CHEMKINFile         "$FOAM_CASE/chemkin/chem.inp";
CHEMKINThermoFile   "~OpenFOAM/thermoData/therm.dat";

inertSpecie     N2;
liquidComponents
(
    C7H16
);

liquidProperties
{
    C7H16  C7H16 defaultCoeffs;
}
```

Chemistry; chemkin directory

```
ELEMENTS
H   O    C   N   AR
END
SPECIE
C7H16 O2 N2 CO2 H2O
END
REACTIONS
 C7H16 + 11O2  => 7CO2 + 8H2O   5.00E+8  0.0   15780.0! 1
FORD / C7H16 0.25 /
FORD / O2 1.5 /
END
```

- **Take a look in** `chem.inp.full`

0 directory

Variable	Initial conditions
ϵ	internalField uniform 90.0, walls zeroGradient
k	internalField uniform 1.0, walls zeroGradient
N_2	internalField uniform 0.766, walls zeroGradient
O_2	internalField uniform 0.233, walls zeroGradient
p	internalField uniform 5e+06, walls zeroGradient
$spray$	empty
T	internalField uniform 800, walls zeroGradient
U	internalField uniform (0 0 0), walls uniform (0 0 0)

Running dieselFoam

- Remove ft and fu and in the aachenBomb/0 directory since these are not needed for this setup.

```
cd $FOAM_RUN/aachenBomb/0
rm ft fu
```

Turn chemistry on in the /constant/chemistryProperties file

```
chemistry                on;
```

Mesh the geometry using blockMesh, and start the dieselFoam solver.

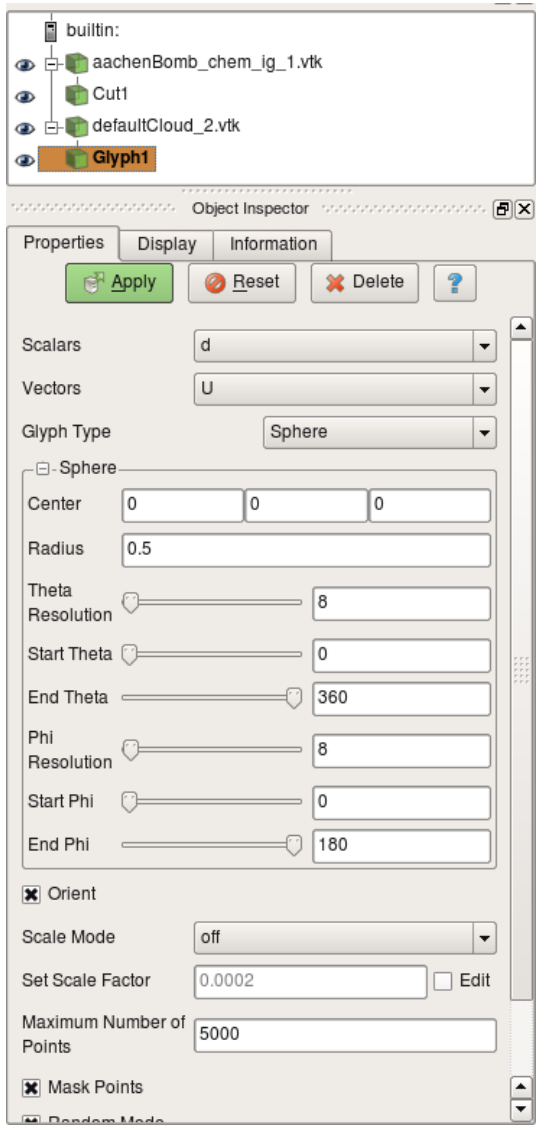
```
cd $FOAM_RUN/aachenBomb
blockMesh
dieselFoam
```

Post process

Since paraFoam can not handle Lagrangian particles use foamToVTK and then ParaView.

```
cd $FOAM_RUN/aachenBomb
foamToVTK
paraview
```

In the /VTK directory open the case file (aachenBomb_1.vtk), also open the particles in the /Lagrangian/defaultCloud_2.vtk file. Create glyphs for the particles.



Written tutorial

Look in the written tutorial on how to:

- Add a second liquid to the droplets
- Copy and compile your own evaporation model
- Copy and compile the solver

Questions?