5th United Kingdom & Éire OpenFOAM® User Meeting
16th-17th January 2017
University College Dublin

# A hybrid slurry CFD model: Euler-Euler to Euler-Lagrange (in development)

Alasdair Mackenzie

Weir Advanced Research Centre, University of Strathclyde, Glasgow, Scotland

# Outline

- Background, context and motivation to the problem
- Development of hybrid model will be explained
- Test case will be shown

- Tutorial can be found on Chalmers website (end of January): http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2016/

# Background



- Weir group produce equipment for the mining and oil and gas industries
- Erosion is a large problem
- CFD modelling is used to predict erosion = better designs
- Longer pump life = happy customer :)

WEIR Advanced Research Centre

# Impellers

**Before**

**After**

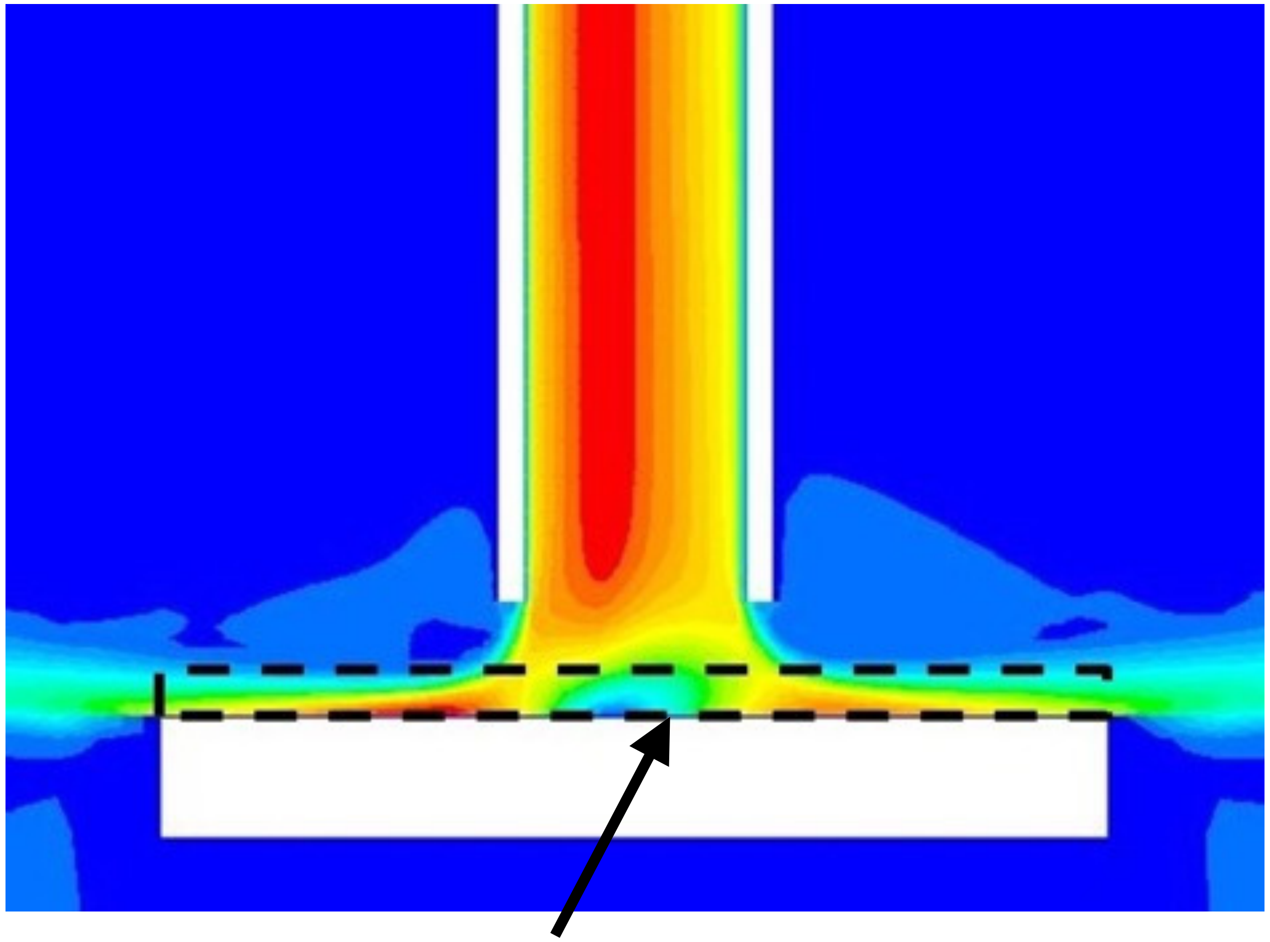
Could be as little as 2 weeks of continuous running for this to happen

# Problem/Motivation

University of Strathclyde Glasgow

- Need particle impact data at the wall for erosion modelling
- Fluid/particulate flow simulation is computationally expensive: especially for dense slurries
- Solution to make faster: Combine with two-fluid model

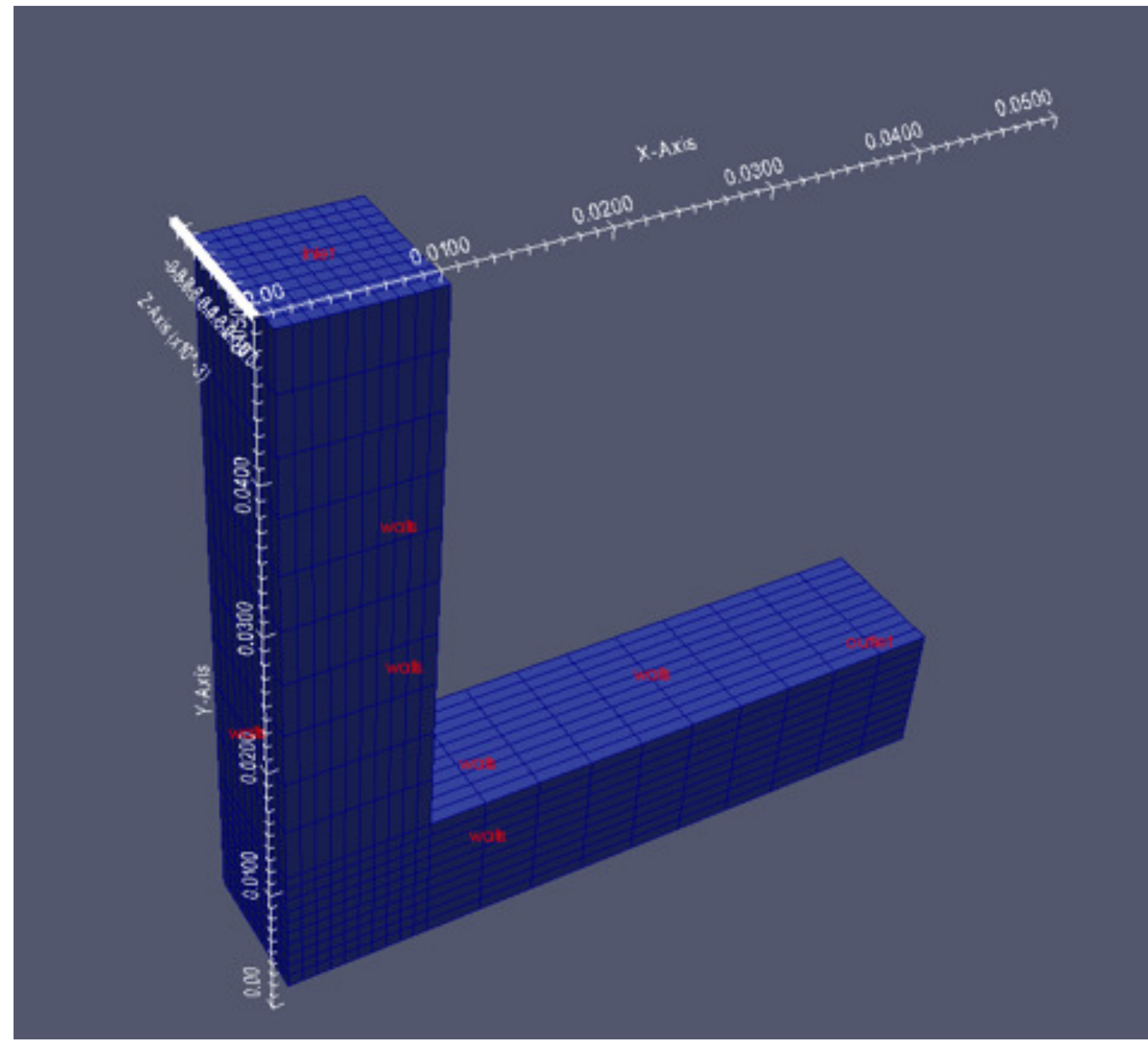
*Velocity contours of submerged jet impingement test*



Dotted region where particles are necessary for impact data

# Geometry and Solvers

- A simple geometry was chosen for solver development
- reactingTwoPhaseEulerFoam for Euler-Euler
- DPMFoam for Euler-Lagrange
- OpenFOAM 3.0.x was used




*Geometry shown with sizes in metres*

# Description of Solvers

University of Strathclyde Glasgow

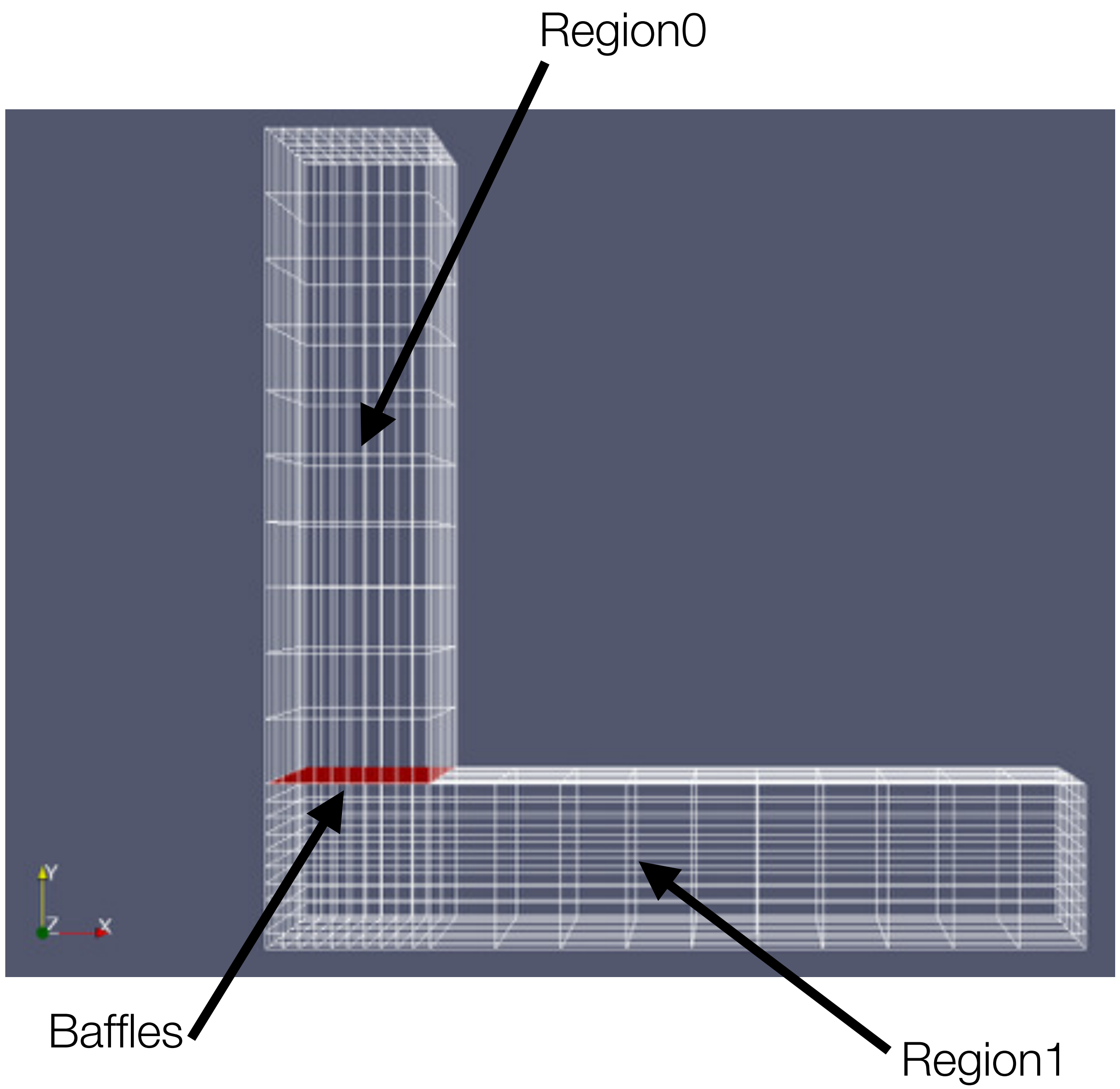
| reactingTwoPhaseEulerFoam | DPMFoam |
|---|---|
| Euler-Euler | Euler-Lagrange |
| Two fluid model | Fluid/particle model |
| Both phases treated as continuum | Transient solver for coupled transport of kinematic particle clouds |
| Incompressible model: setting in dictionary | Includes the effect of volume fraction of the particles on the continuous phase |
| Fast to solve | |

# Combining the solvers

- A new solver was made based on the EE model
- To have 2 solvers running, 2 regions were created
- To go from fluid to particles, we need a transition
- An outlet/inlet is needed for particle phase, but shouldn't affect the rest of the flow
- Solution…

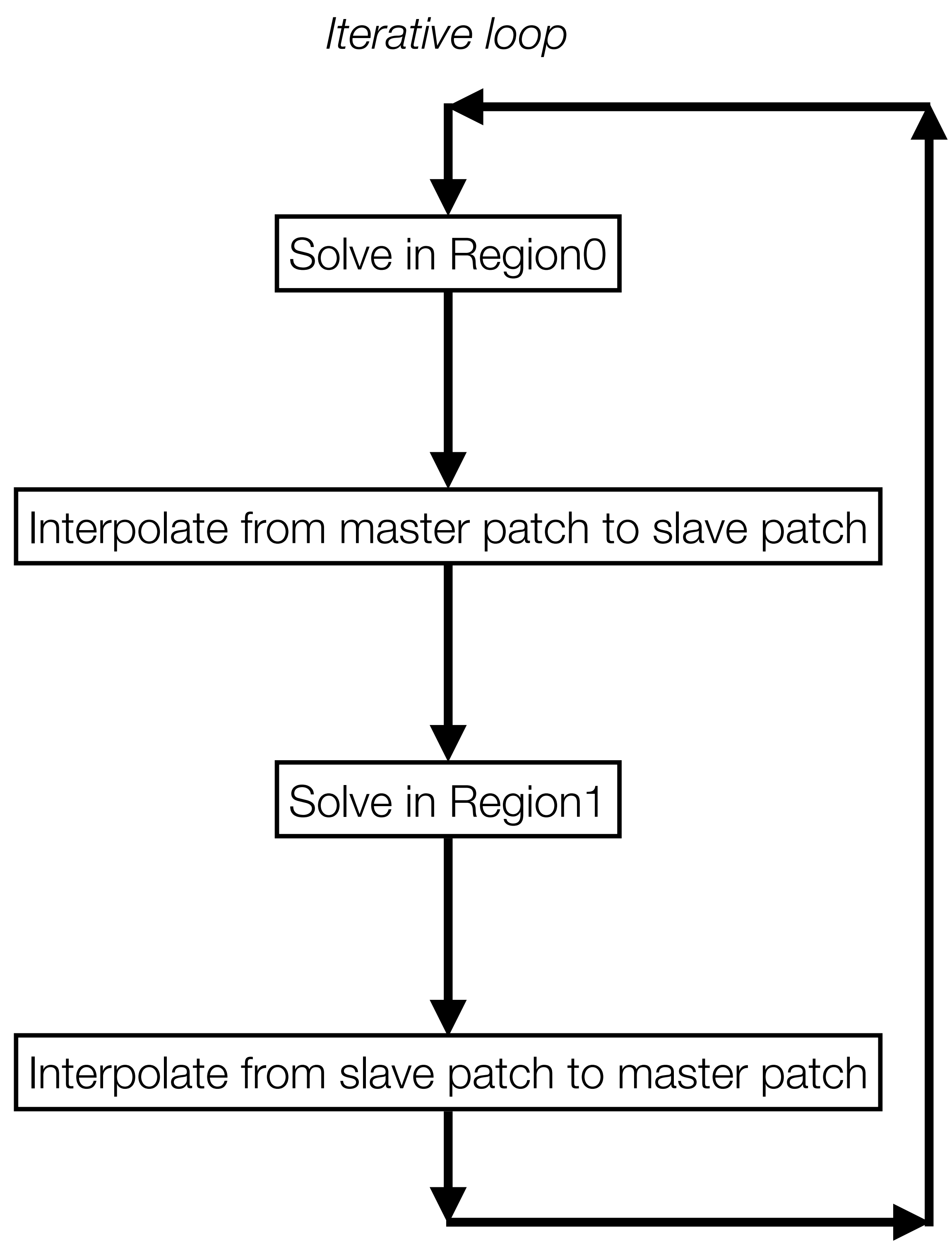
# Baffles + Regions

- createBaffles: makes internal surface into boundary face
- *master* and *slave* patch created
- splitMeshRegions: Splits mesh into 2 separate regions
- BC's can now be applied to baffle patches
- chtMultiRegionFoam: Inspiration for solving regions sequentially



# Interpolation

- patchToPatchInterpolation: transfers data between two patches
- All variables are interpolated: U1, U2, p, p_rgh, alpha1, alpha2, k, epsilon, nut, and theta
- After this is implemented, the domain runs as if it was one region, not two: the surface doesn't affect the flow
- 'back pressures' are taken into account by interpolating upstream



# DPMFoam added

University of Strathclyde Glasgow

- Code from DPMFoam was added to new solver
- Particles injected from slave patch after back interpolation (slave to master)
- Particles are only in region1 (near wall)
- Injection values based on phase 2 from region0 by using a lookup table: kinematicLookupTableInjection

# DPMFoam injection

```
/* (x y z) (u v w) d rho mDot numParcels
     where:
   x, y, z = global cartesian co-ordinates [m]
   u, v, w = global cartesian velocity components [m/s]
   d       = diameter [m]
   rho     = density [kg/m3]
   mDot    = mass flow rate [kg/m3]
   numParcels = number of Parcels
   Dictionary for the KinematicLookupTableInjection */
(
(0.0005 0.01 -0.0005) (0.01417 0.01831 -0.001718) 5.5e-05 2750 0.005 -2
(0.0015 0.01 -0.0005) (0.06206 -0.1608 -0.001616) 5.5e-05 2750 0.005 10
(0.0025 0.01 -0.0005) (0.1088 -0.3422 -0.0005019) 5.5e-05 2750 0.005 19
(0.0035 0.01 -0.0005) (0.1497 -0.4695 -0.001312) 5.5e-05 2750 0.005 24
```

- Modified kinematicLookupTableInjection used to inject particles
- Lookup table is updated every time step (but not read every time step: advice welcome!)
- 1 line = 1 cell (100 cells in this case)
- Values for particle injection are based on new updated values so solver can deal with geometry changes etc. See Lopez' presentation for more details:

https://sourceforge.net/projects/openfoam-extend/files/OpenFOAM_Workshops/OFW10_2015_AnnArbor/Presentations/Lopez-present-OFW10-16.pdf/download

# DPMFoam injection

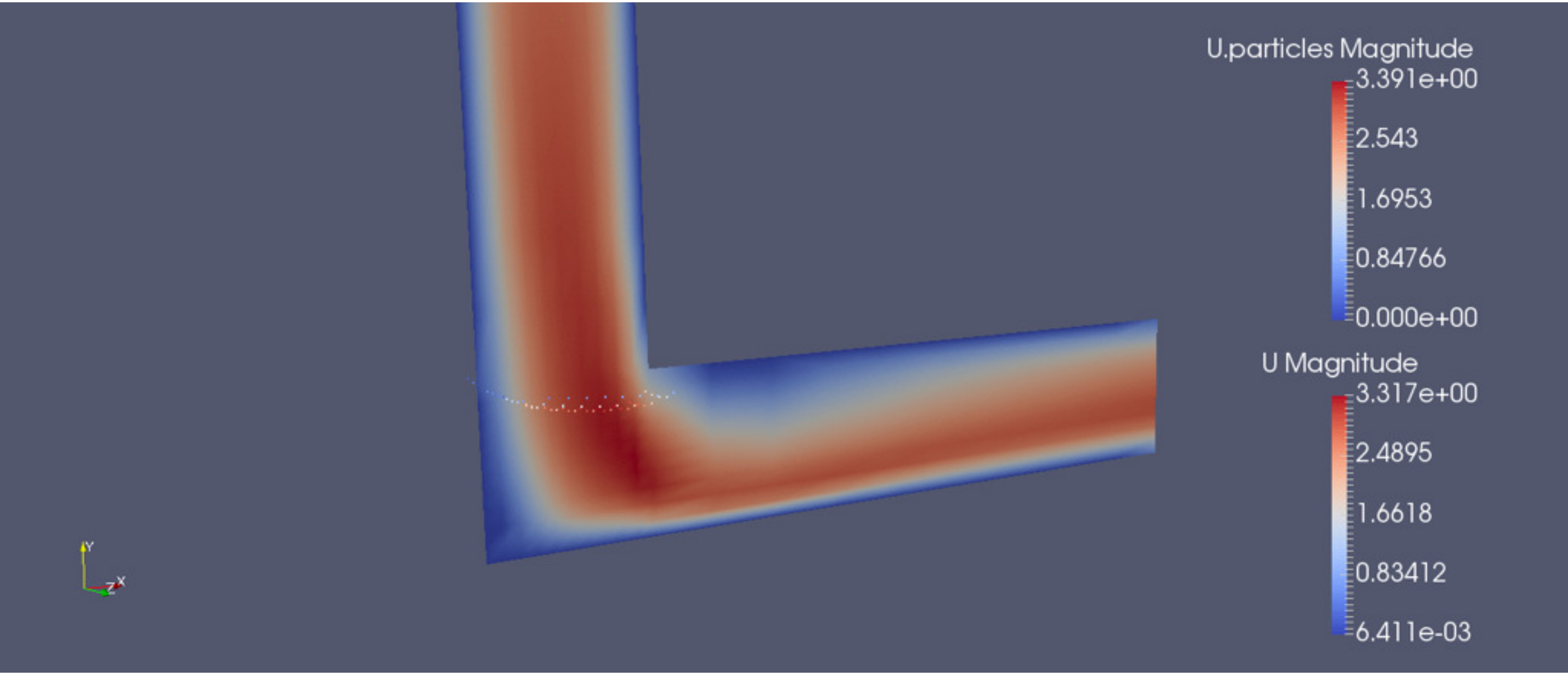
University of Strathclyde Glasgow

```
    os <<"/* (x y z) (u v w) d rho mDot numParcels  \n";
    os <<"     where: \n";
    os <<"   x, y, z = global cartesian co-ordinates [m] \n";
    os <<"   u, v, w = global cartesian velocity components [m/s] \n";
    os <<"   d       = diameter [m] \n";
    os <<"   rho     = density [kg/m3] \n";
    os <<"   mDot    = mass flow rate [kg/m3] \n";
    os <<"   numParcels = number of Parcels  \n";

    os <<"   Dictionary for the KinematicLookupTableInjection */ \n";
    os << "( " << endl;
       forAll(interpolatedInletU1, i)
           {
               os << centres[i] << " " << interpolatedInletU1[i] << " " << 55e-6 << " " << 2750 << " " << 0.005 << " "
 << floor ((alpha1[i]*(mag(normalSlaveVector[i]))*uNormal[i])/((8.71e-14)*3*(-1)*5000)) << endl;
           }
    os << "//The end"<< endl;
    os << ");"<< endl;
```

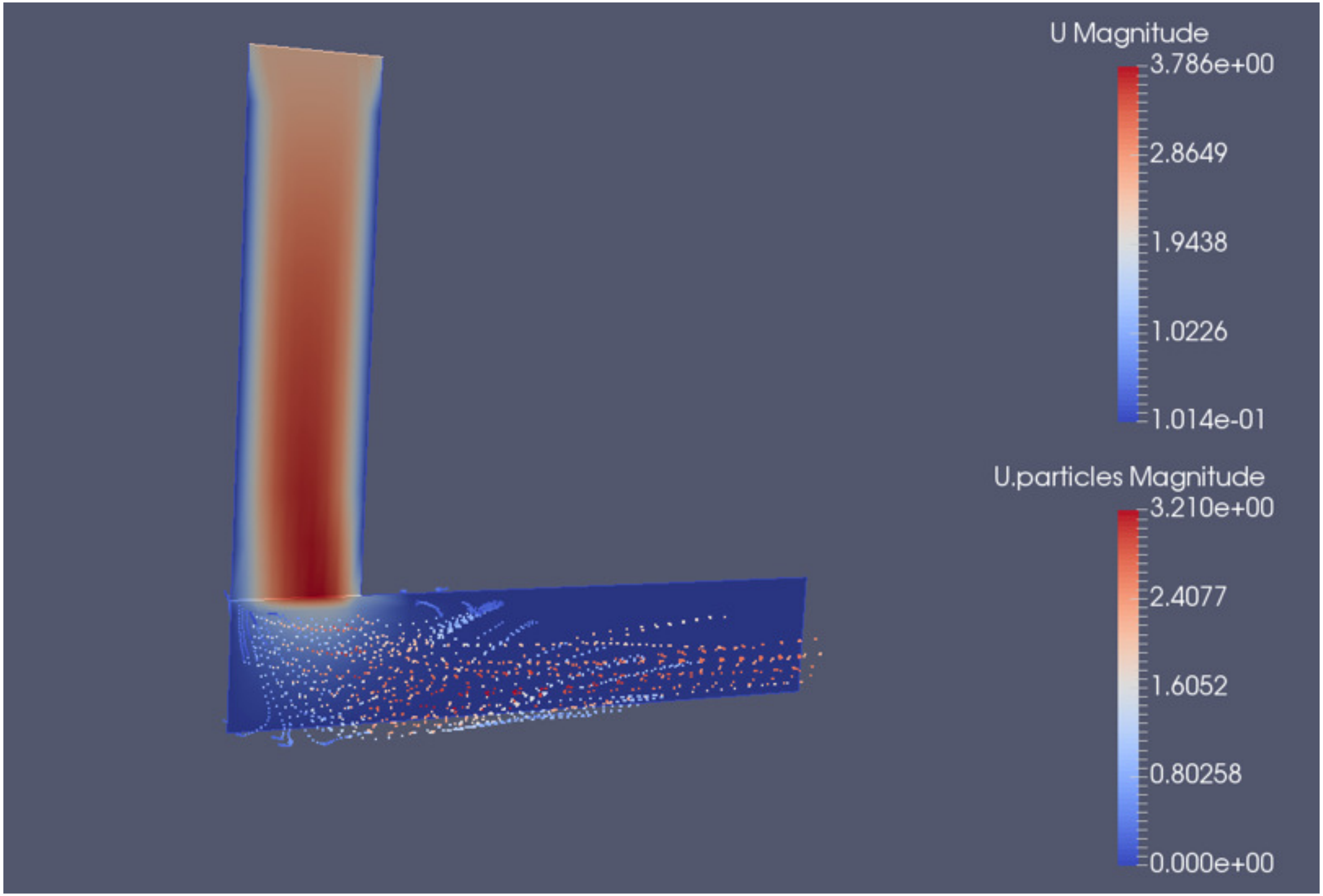
- Number of parcels to be injected is calculated from volume flow rate of 2nd phase of fluid.
- Number of parcels/cell = (alpha particles * area of cell * normal velocity component to cell boundary face) / (volume of one particle * number of particles/parcel * number of time-steps/ second)

# Velocity contours




- 2D slice through Z normal. Particles injected from slave patch

# Velocity contours



# Comparison

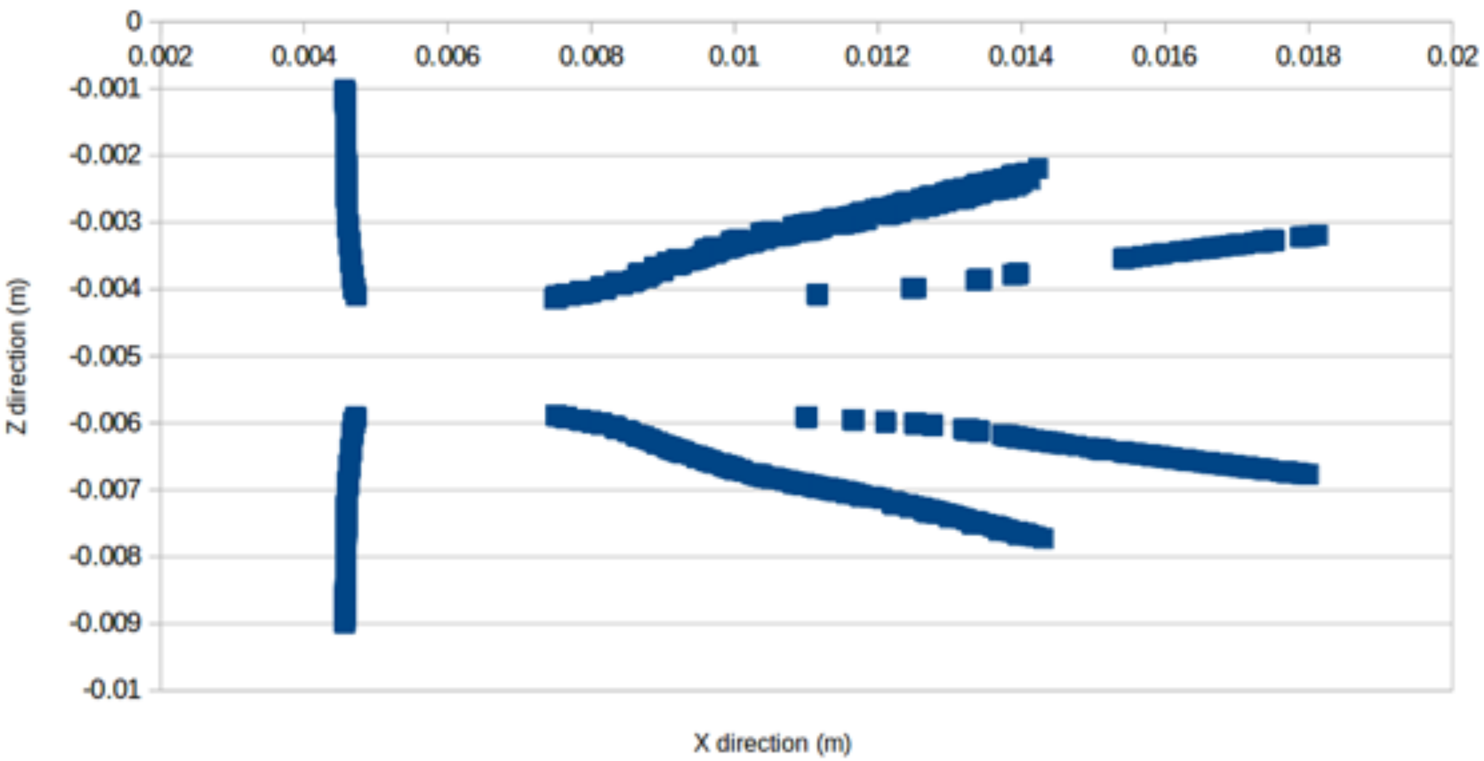
University of Strathclyde Glasgow

- New solver was compared against standard EL and EE solvers
- Hybrid model is almost double the speed of the EL

Execution time from 0-0.39s: % mass concentration (MC)

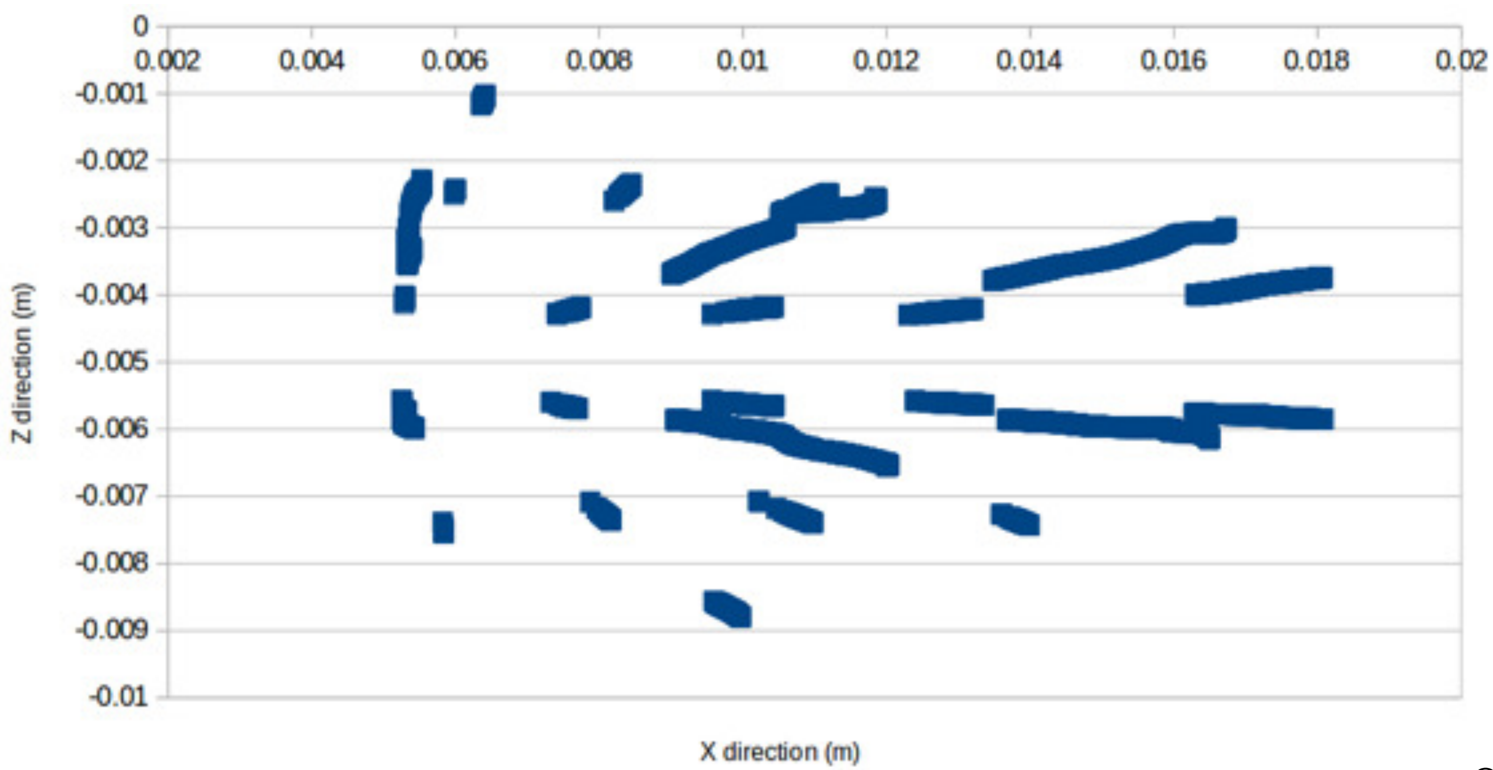
| Model | Execution Time (s) | |
|---|---|---|
| | 1% MC | 2% MC |
| Hybrid model | 225 | 298 |
| Euler-Lagrange | 420 | 585 |
| Euler-Euler | 102 | 105 |

# Comparison




Hybrid Model particle impacts

Data taken from bottom wall on pipe bend




DPMFoam particle impacts

# Future work

- Validation of hybrid model: CFD and experimental (PIV)
- Particles to fluid, for after region of interest…
- Move lookupTable to memory?
- Make solver re-read the lookupTable (suggestions welcome)

# Conclusion

University of Strathclyde Glasgow

- Solver should dramatically reduce computational time
- Particle data should still be present near walls, where required
- Enable better design of mining equipment

*Worn impeller of slurry pump*

5th United Kingdom & Éire OpenFOAM® User Meeting
16th-17th January 2017
University College Dublin

# Thank you. Questions?

*alasdair.mackenzie.100@strath.ac.uk*

Weir Advanced Research Centre, University of Strathclyde, Glasgow, Scotland