12th OpenFOAM® Workshop, University of Exeter
24th-27th July 2017

A new hybrid slurry CFD model compared with experimental results

Alasdair Mackenzie[1], Vanja Škurić[2], MT Stickland[1], WM Dempster[1]

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

2. University of Zagreb, Zagreb, Croatia

WEIR Advanced Research Centre

University of Strathclyde Glasgow

Outline

- Background, context and motivation to the problem
- Development of hybrid model
- PIV experiments/validation work

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, better for customer

Advanced Research Centre

Ball mill video

Impeller

Before | **After**

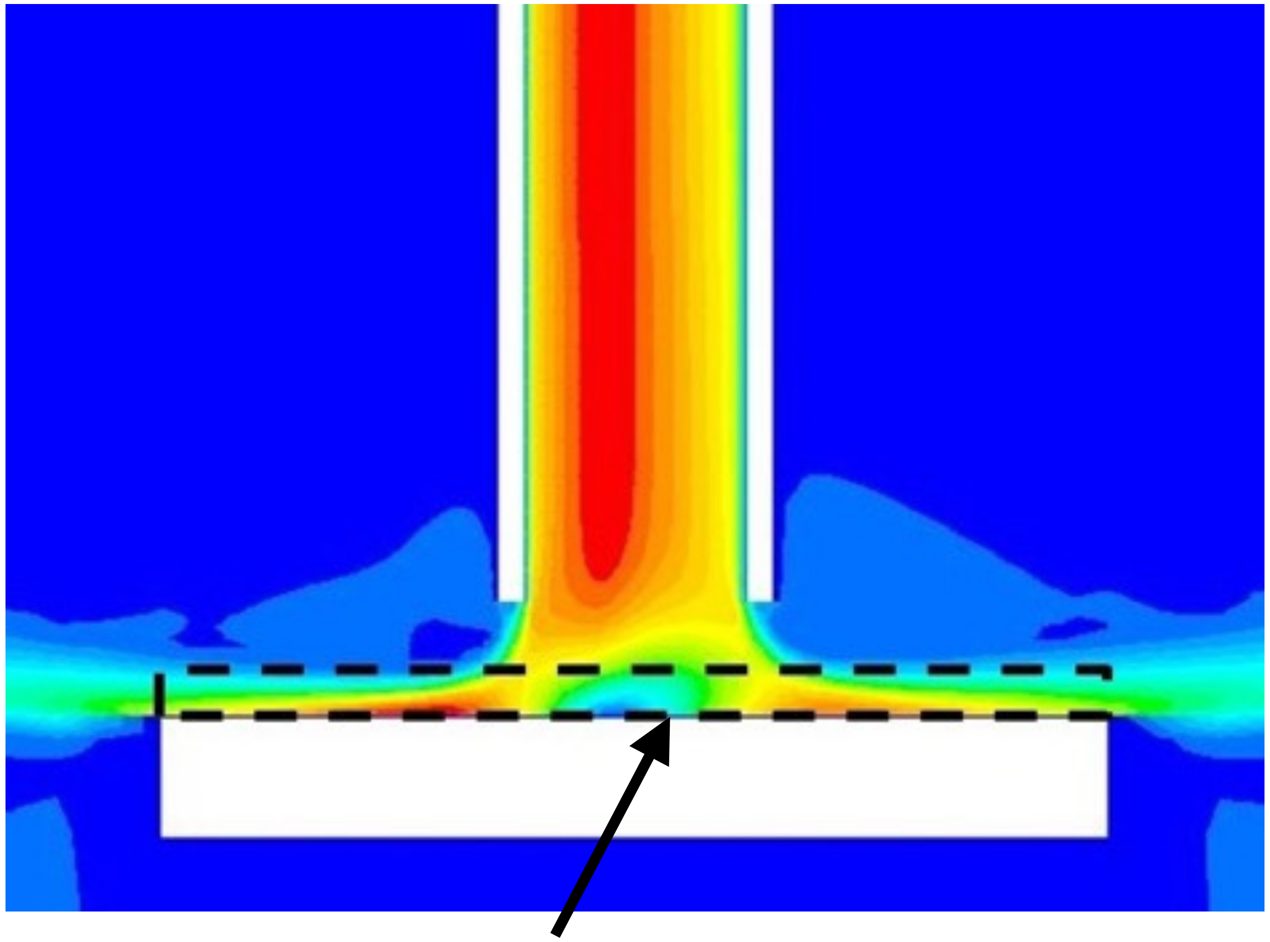
It could be as little as 2 weeks of continuous running for this to happen

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
note: old asymmetric geometry pictured

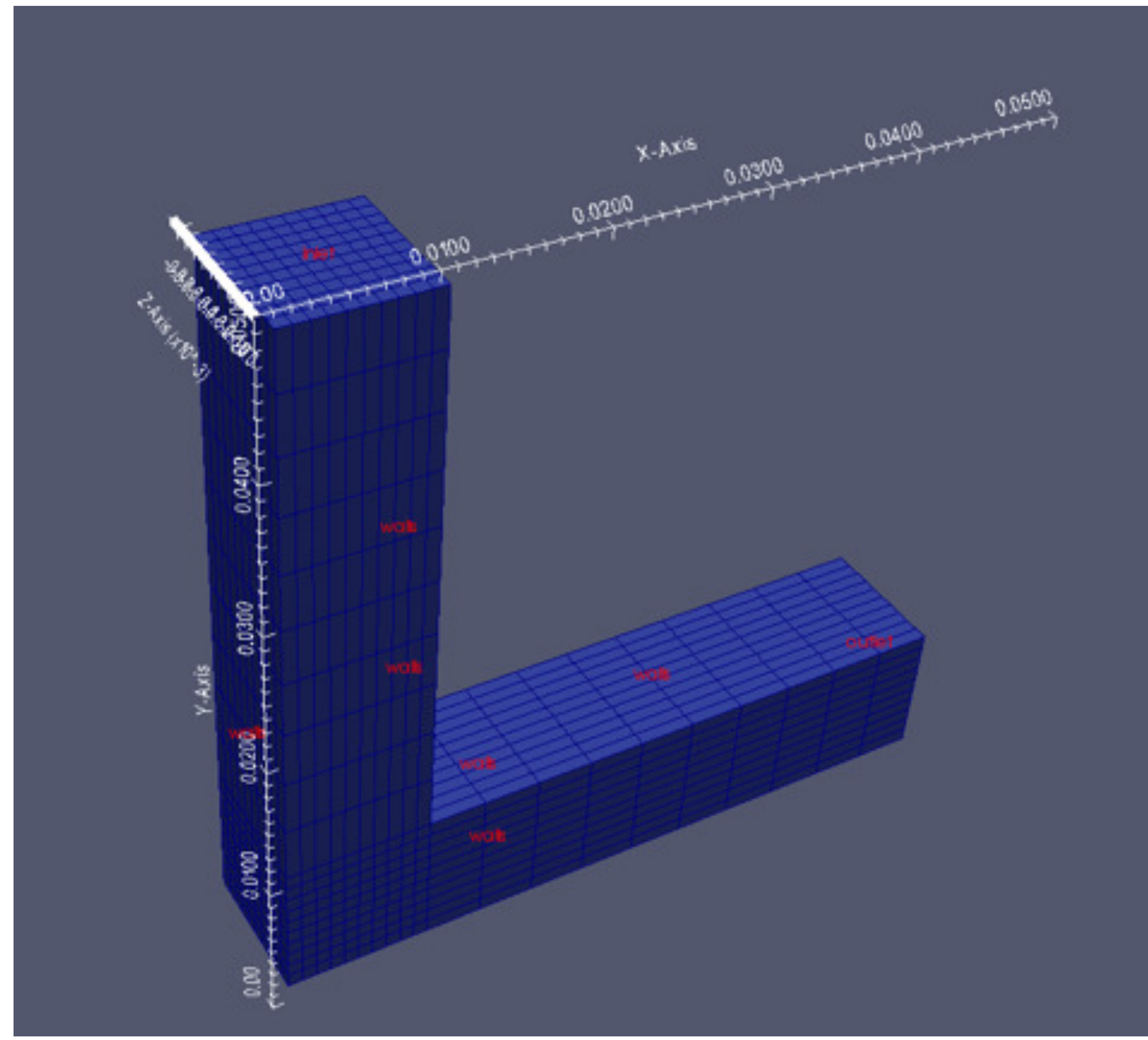


Dotted region where particles are necessary for impact data

Geometry and Solvers

University of Strathclyde Glasgow

- A simple geometry was chosen for solver development
- reactingTwoPhaseEulerFoam for Euler-Euler
- DPMFoam for Euler-Lagrange
- OpenFOAM 3.0.x was used
- Tutorial available at: http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2016/AlasdairMackenzie/tutorial1.pdf



Geometry shown with sizes in metres

Description of Solvers

University of Strathclyde Glasgow

reactingTwoPhaseEulerFoam

Euler-Euler

Two fluid model

Both phases treated as continuum

Incompressible model: setting in dictionary

Fast to solve

DPMFoam

Euler-Lagrange

Fluid/particle model

Transient solver for coupled transport of kinematic particle clouds

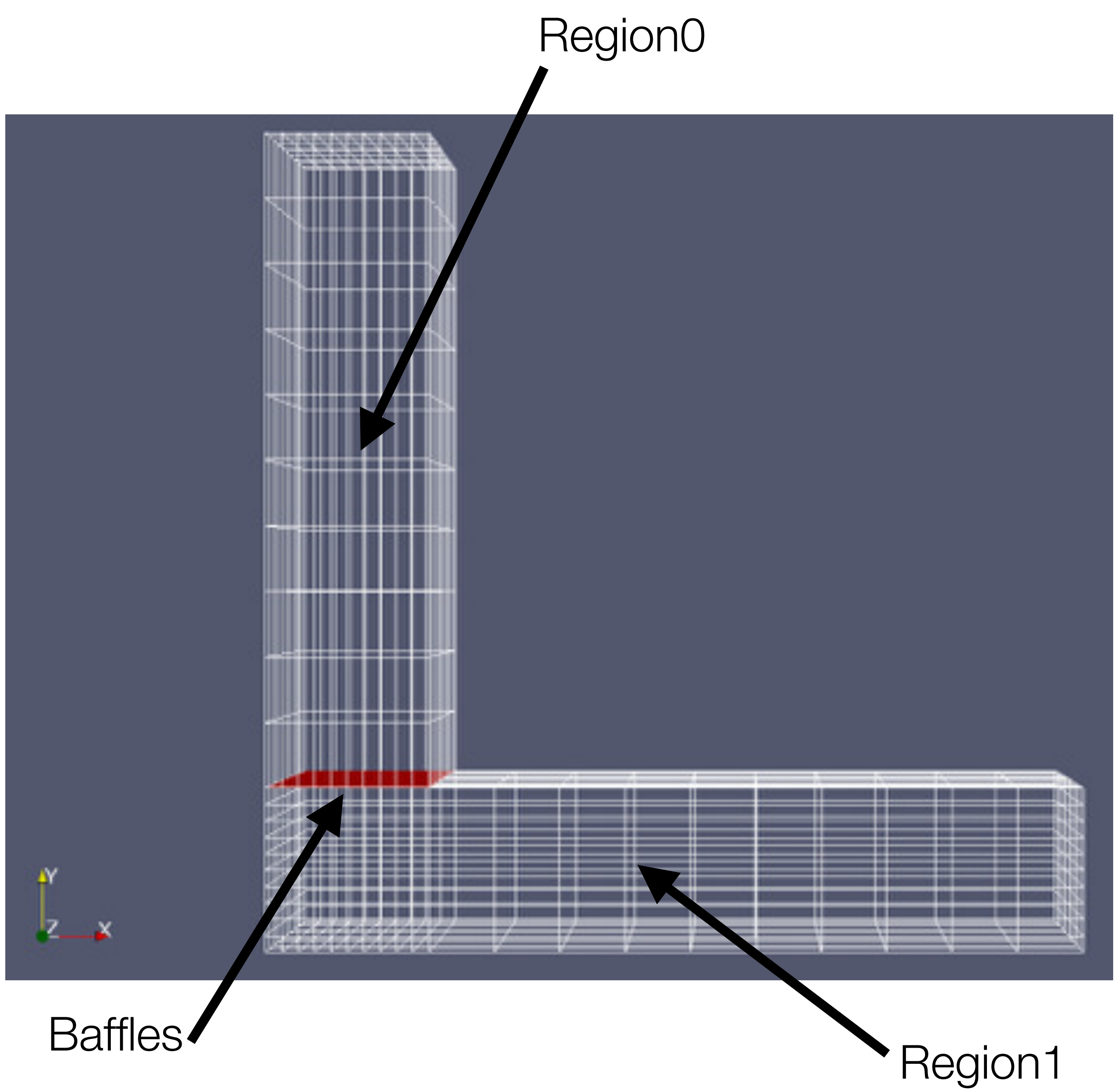
Includes the effect of volume fraction of the particles on the continuous phase

Combining the solvers

- A new solver was made based on the EE model
- To have 2 solvers running, we need 2 regions
- 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

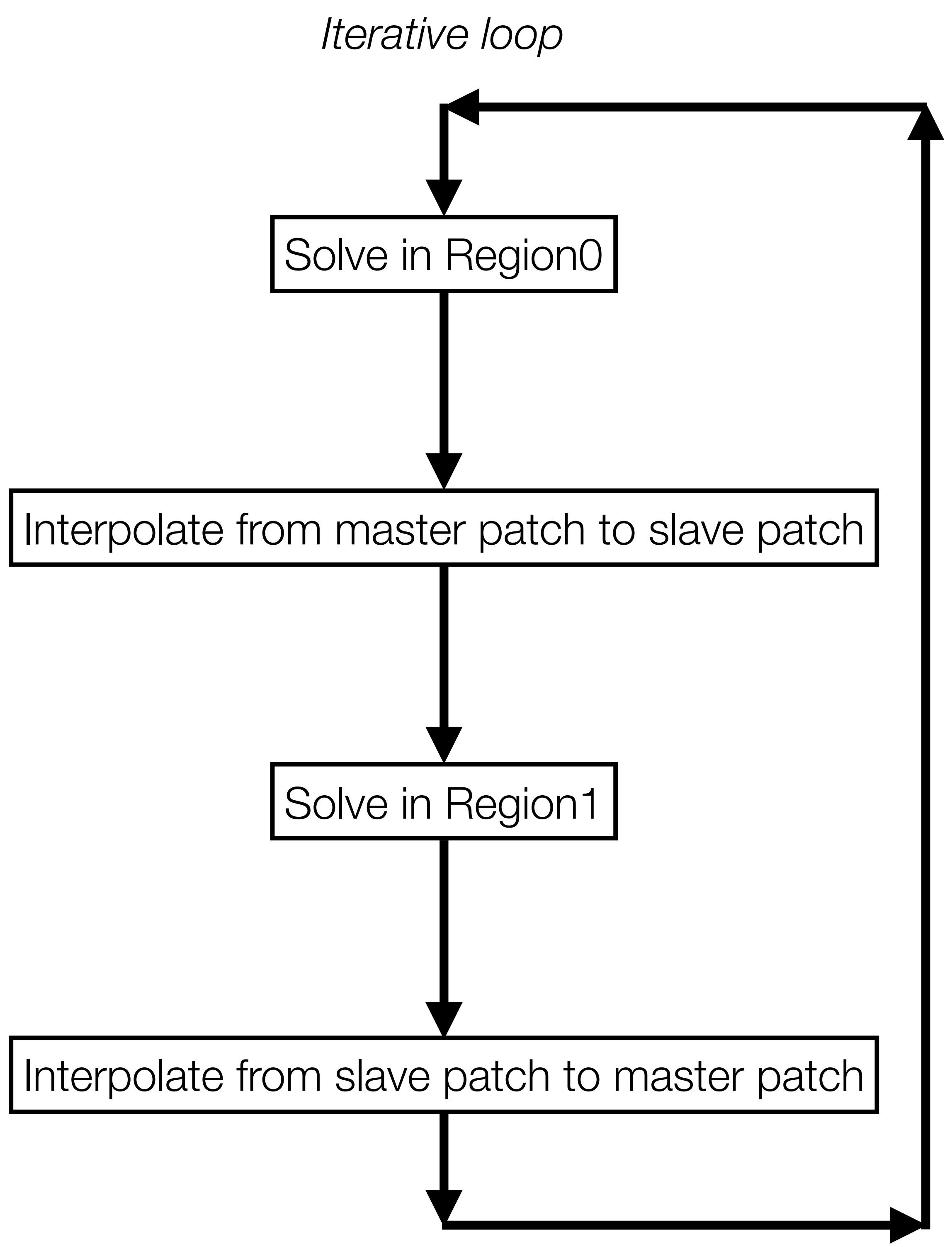
- createBaffles: makes internal surface into boundary face
- *master* and *slave* patch created
- splitMeshRegions: Splits mesh into 2 separate regions
- BC's can now be applied to baffle patches
- chtMultiRegionFoam: Inspiration for solving regions sequentially



Interpolation

University of Strathclyde Glasgow

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



DPMFoam added

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

DPMFoam injection

University of Strathclyde Glasgow

```
/* (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
- 1 line = 1 cell
- 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

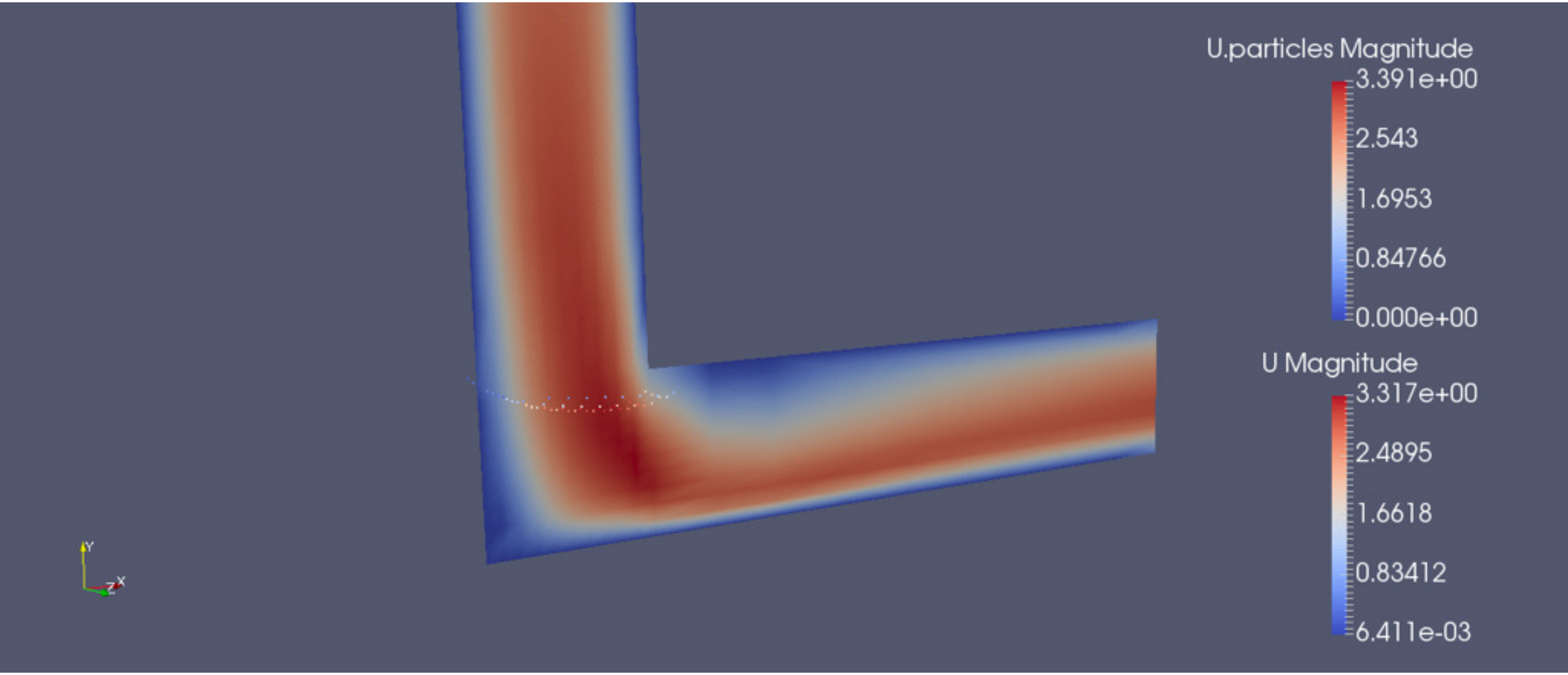
DPMFoam injection

```
    kinematicParcelInjectionDataIOList& injectors =
        const_cast<kinematicParcelInjectionDataIOList&>
        (
            mesh.lookupObject<kinematicParcelInjectionDataIOList>("kinematicLookupTableInjection")
        );

    forAll(injectors, i)
    {
        injectors[i].x() = centres[i]; //forgot to add this when in Croatia
        injectors[i].U() = U1.boundaryField()[slave][i];
        injectors[i].numParticles() = abs((alpha1.boundaryField()[master][i]*(mag(normalSlaveVector[i]))
        *uNormal[i])/(((((pi)*pow3(injectors[i].d()))/6))*nParticle*(-1)*timestepsPerSecond));
    }

            injectors.write();

}
```

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

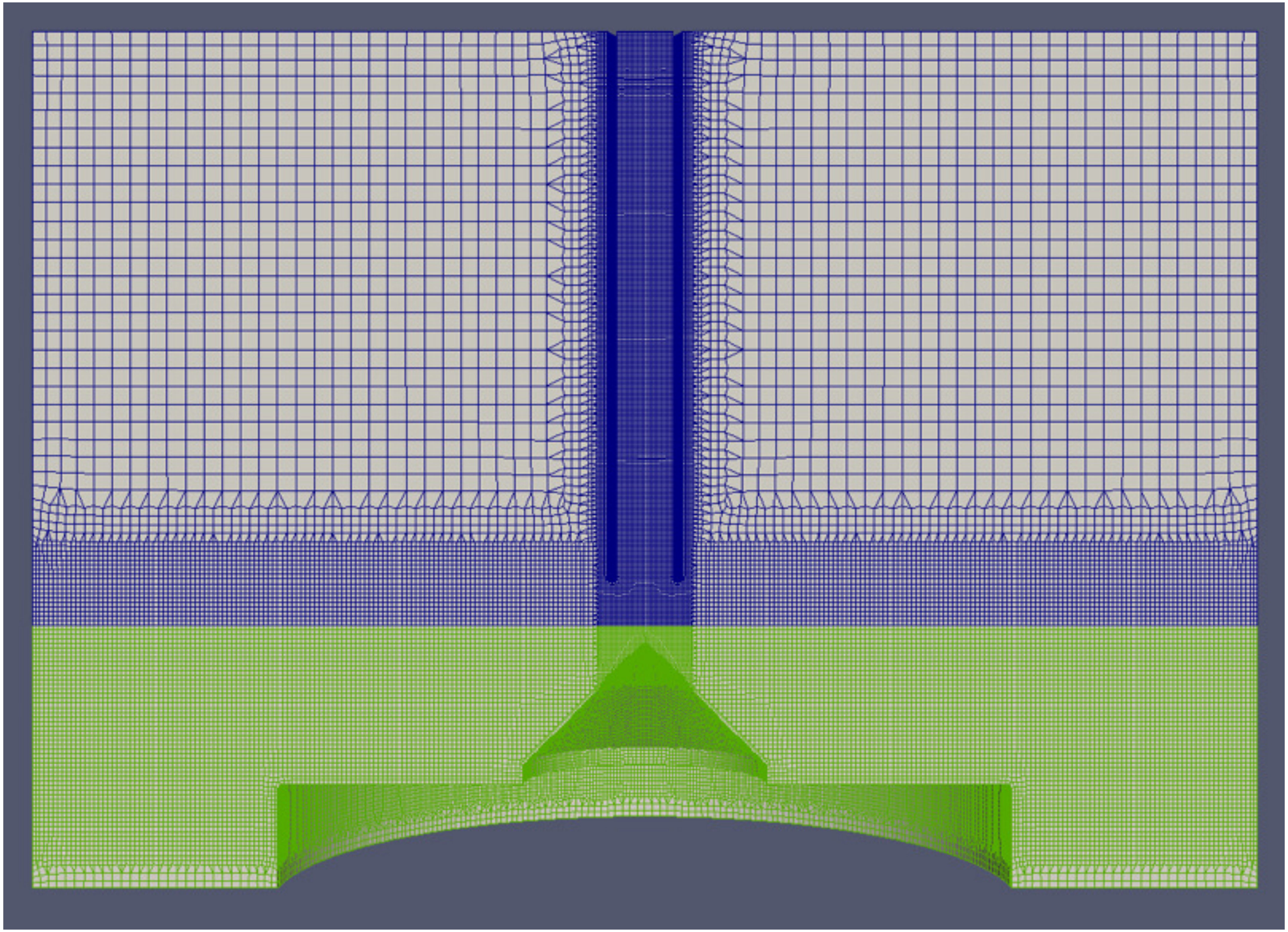
Velocity contours

University of Strathclyde Glasgow

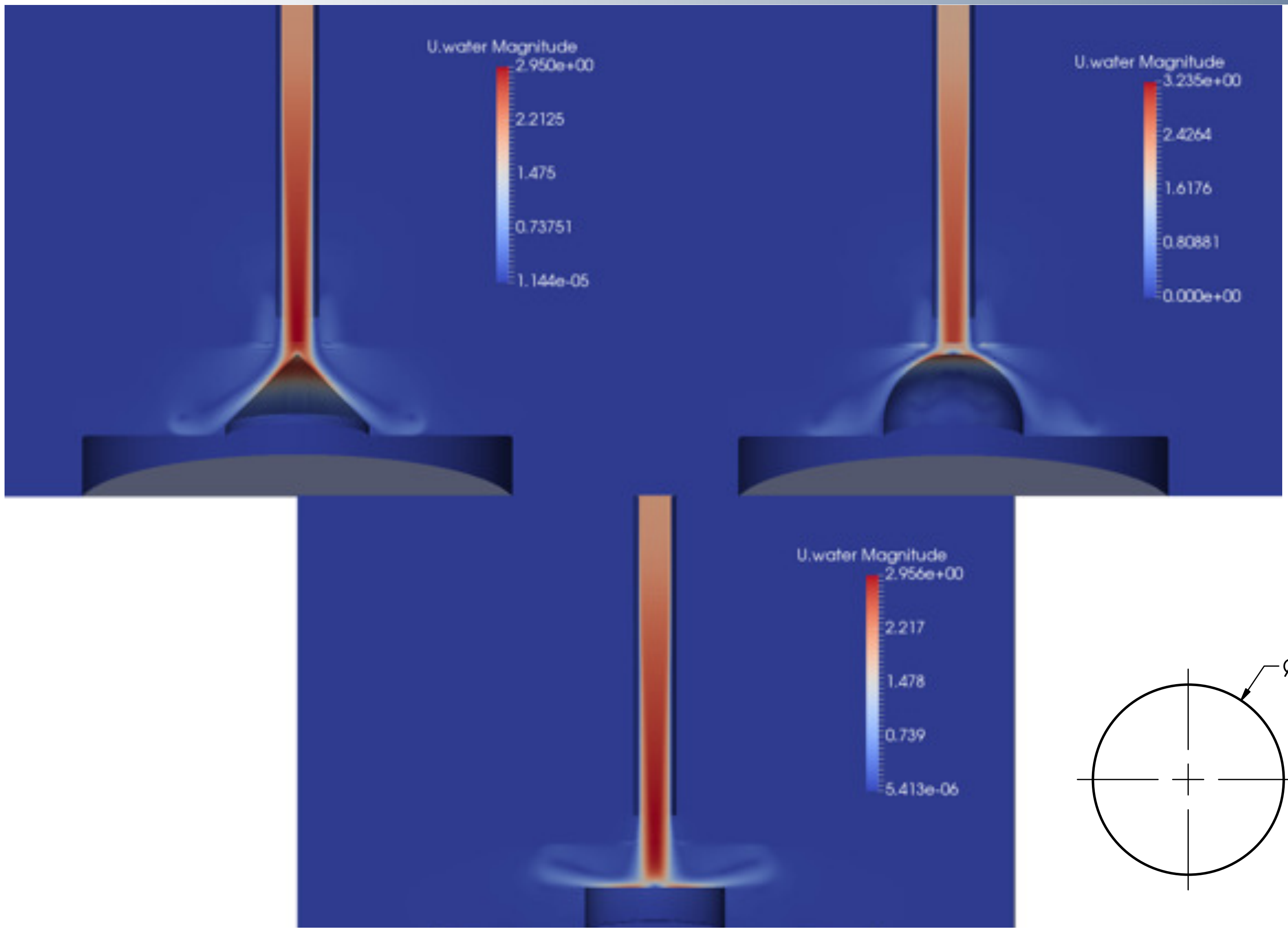


- 2D slice through Z normal. Particles injected from slave patch

Real geometry setup

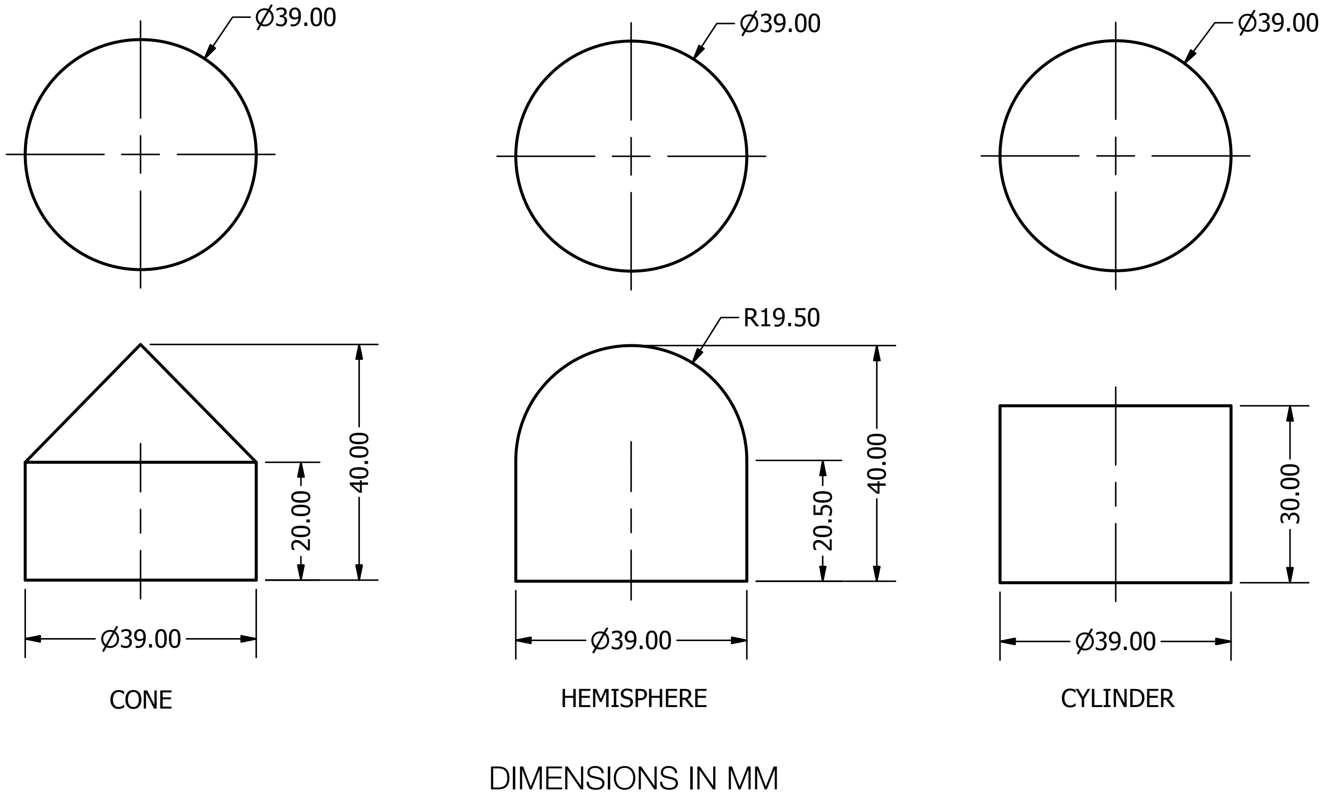


3 sample geometries



New solver was tested on the shown geometries

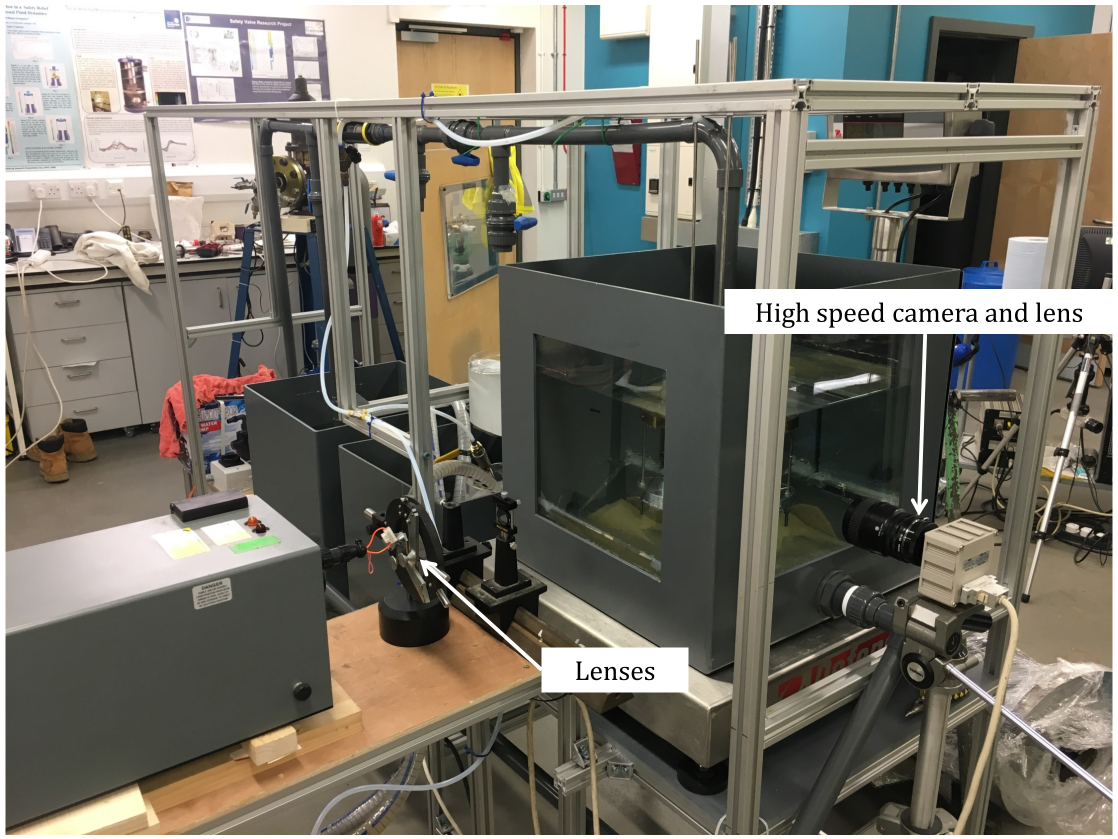
Particle Image Velocimetry was carried out for validation

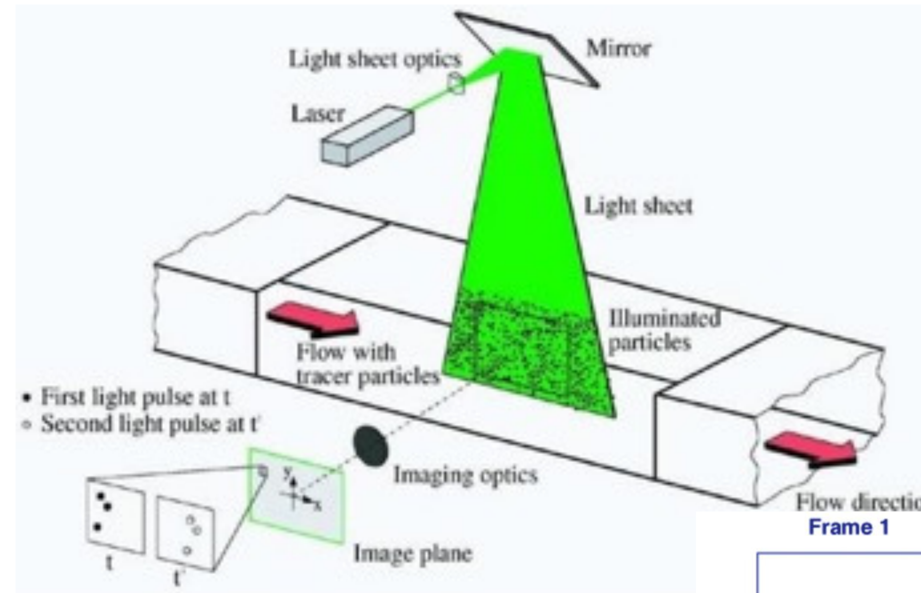


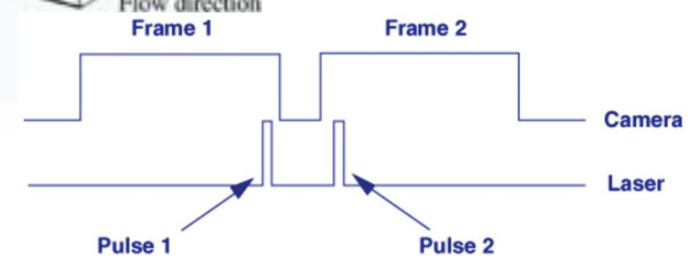
Mass flow inlet (~2m/s)
K-Omega SST turbulence model used
Only first phase compared **(so far)**

Experimental setup

University of Strathclyde Glasgow







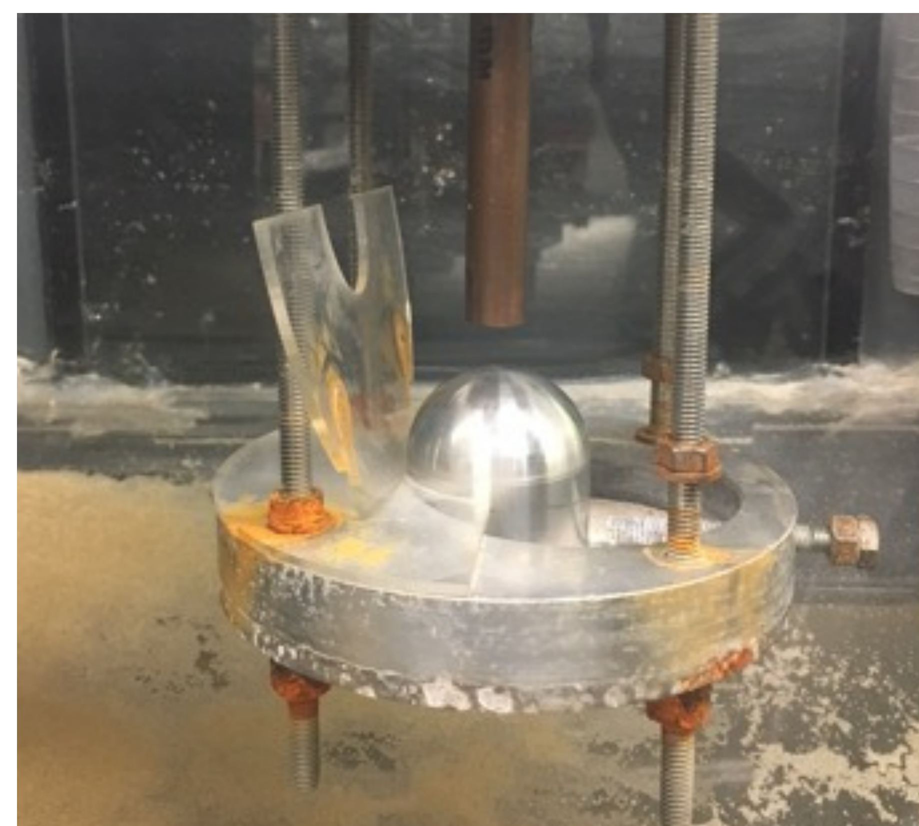
<u>Particle Image Velocimetry</u>

Frame straddling used by laser ΔT=67μs

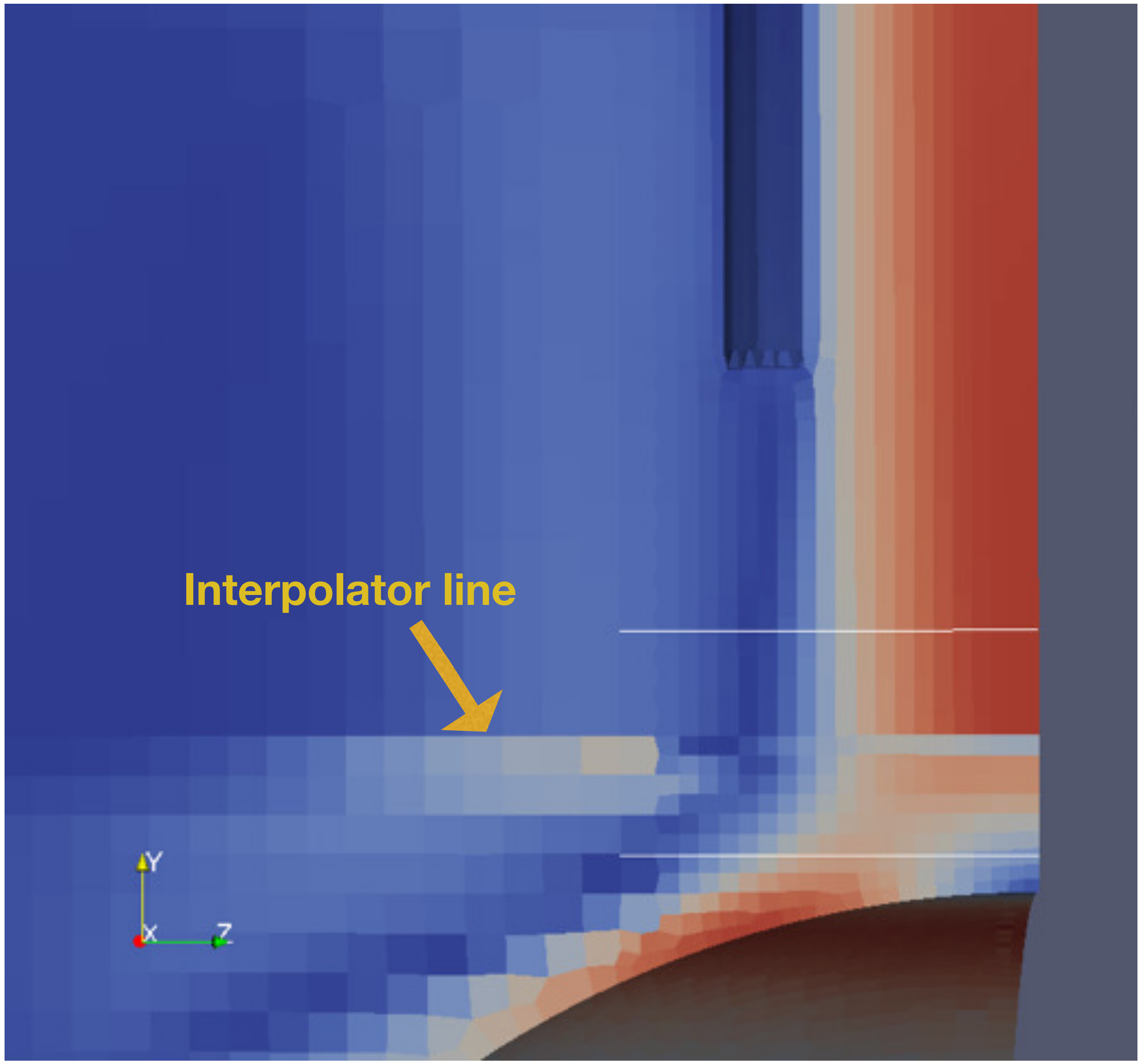
DantecDynamics laser system

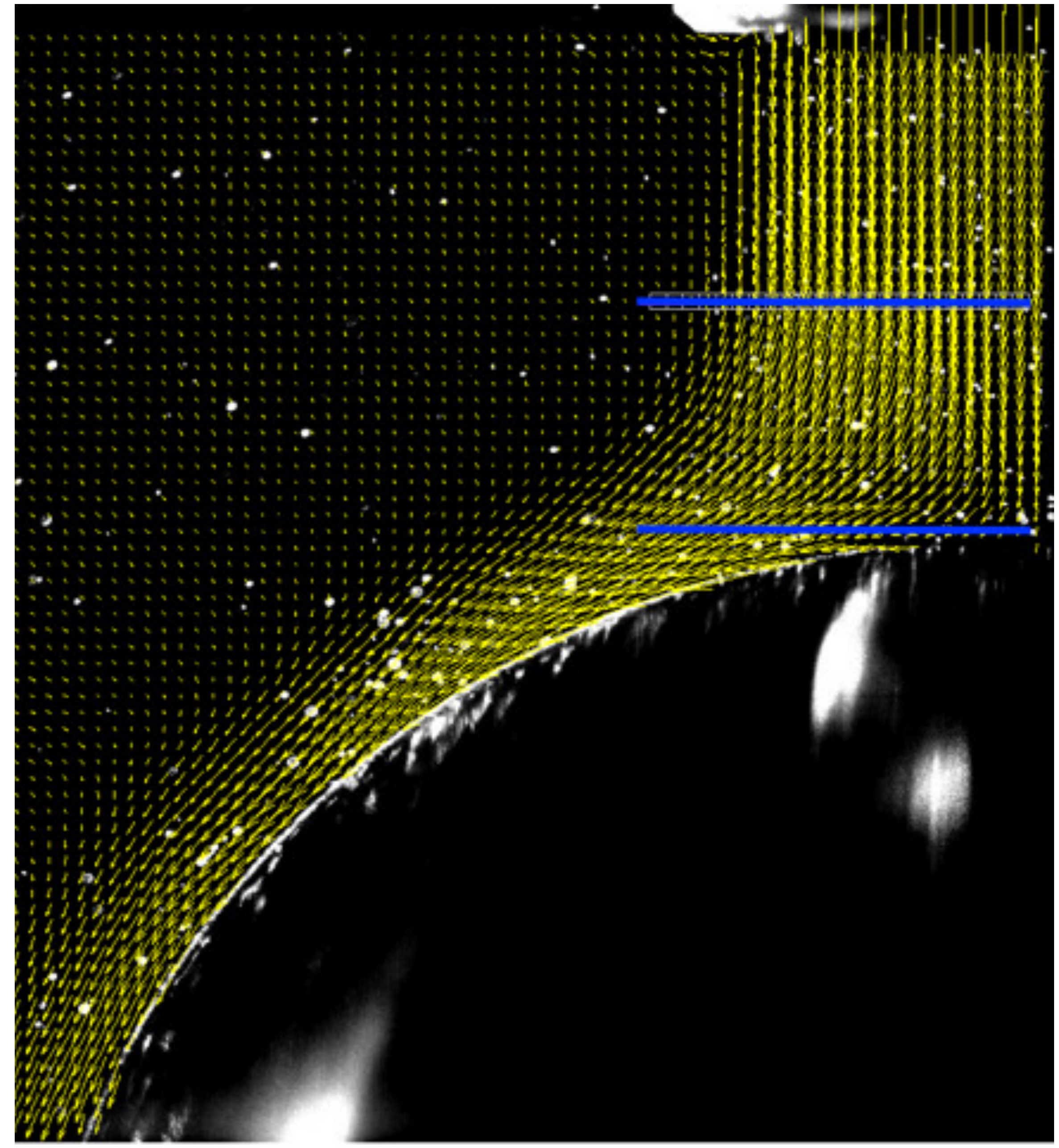
250 images used, 125 image pairs

Reynolds numbers of experiments and CFD are both around 10^5 (so are comparable)



Comparison of data

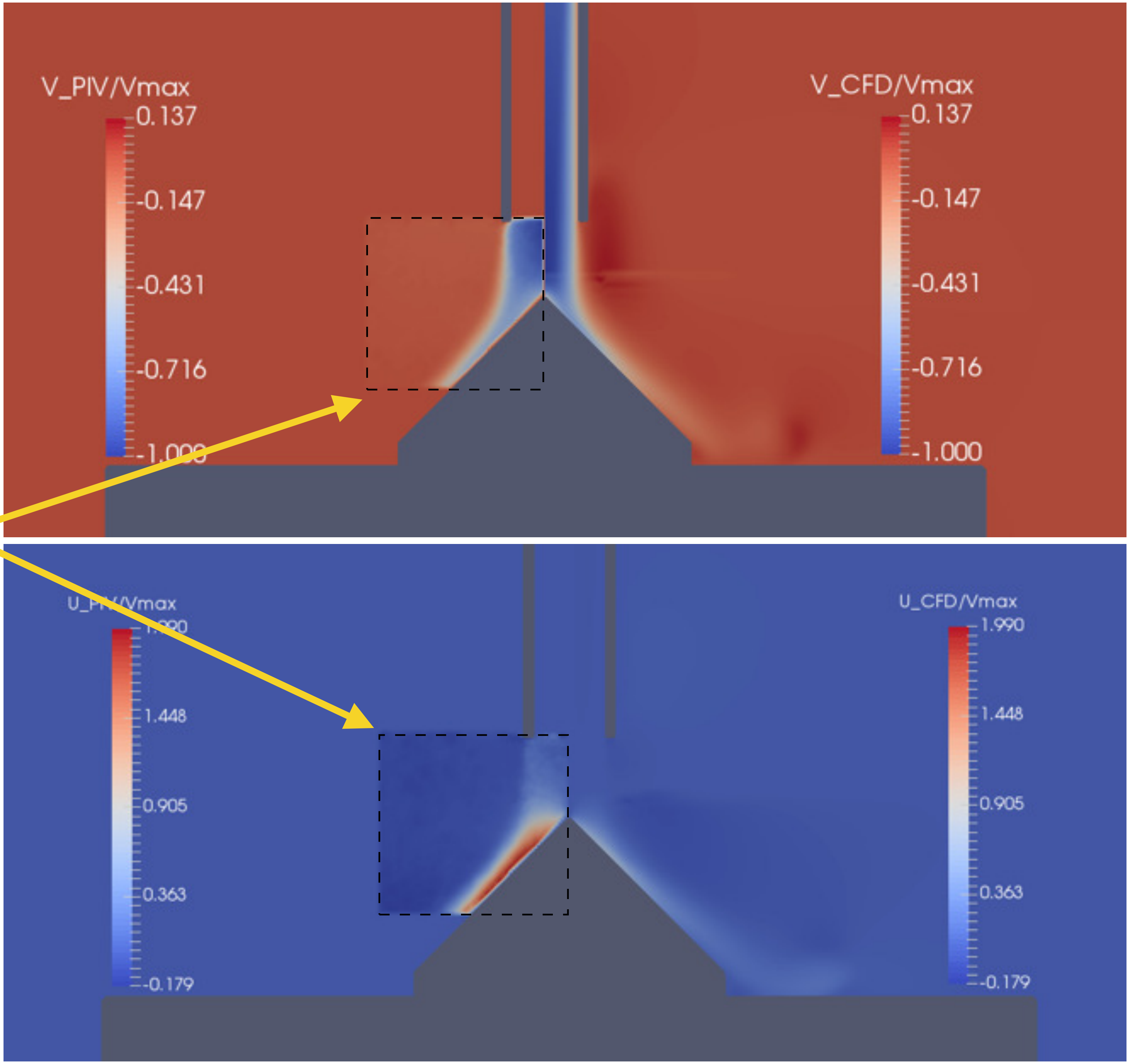




Lines show where data is taken from:
top is 5mm from nozzle, bottom is 9.5/10mm from nozzle
Interpolator is in between sample lines

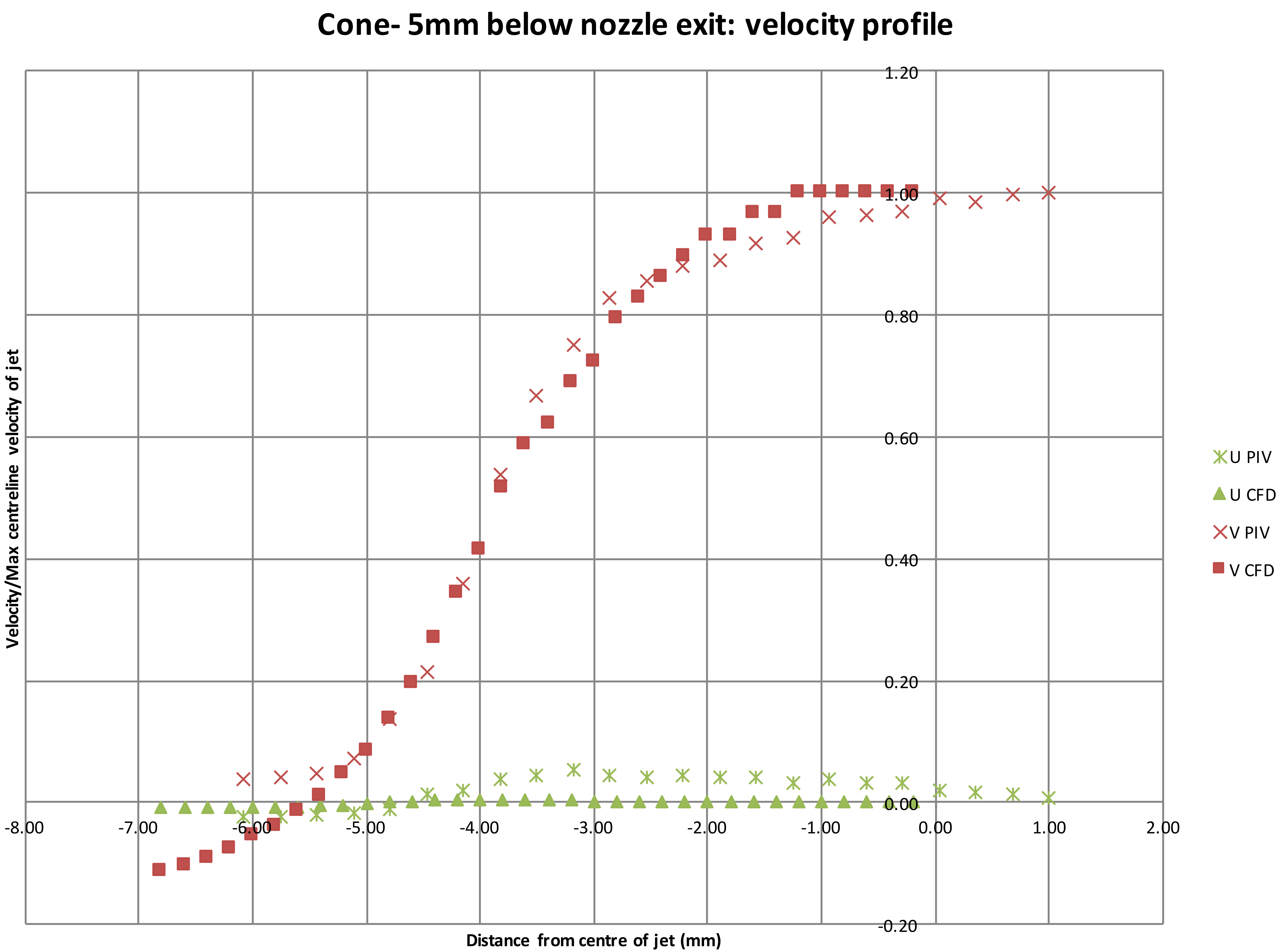
University of Strathclyde Glasgow

Cone velocity contours



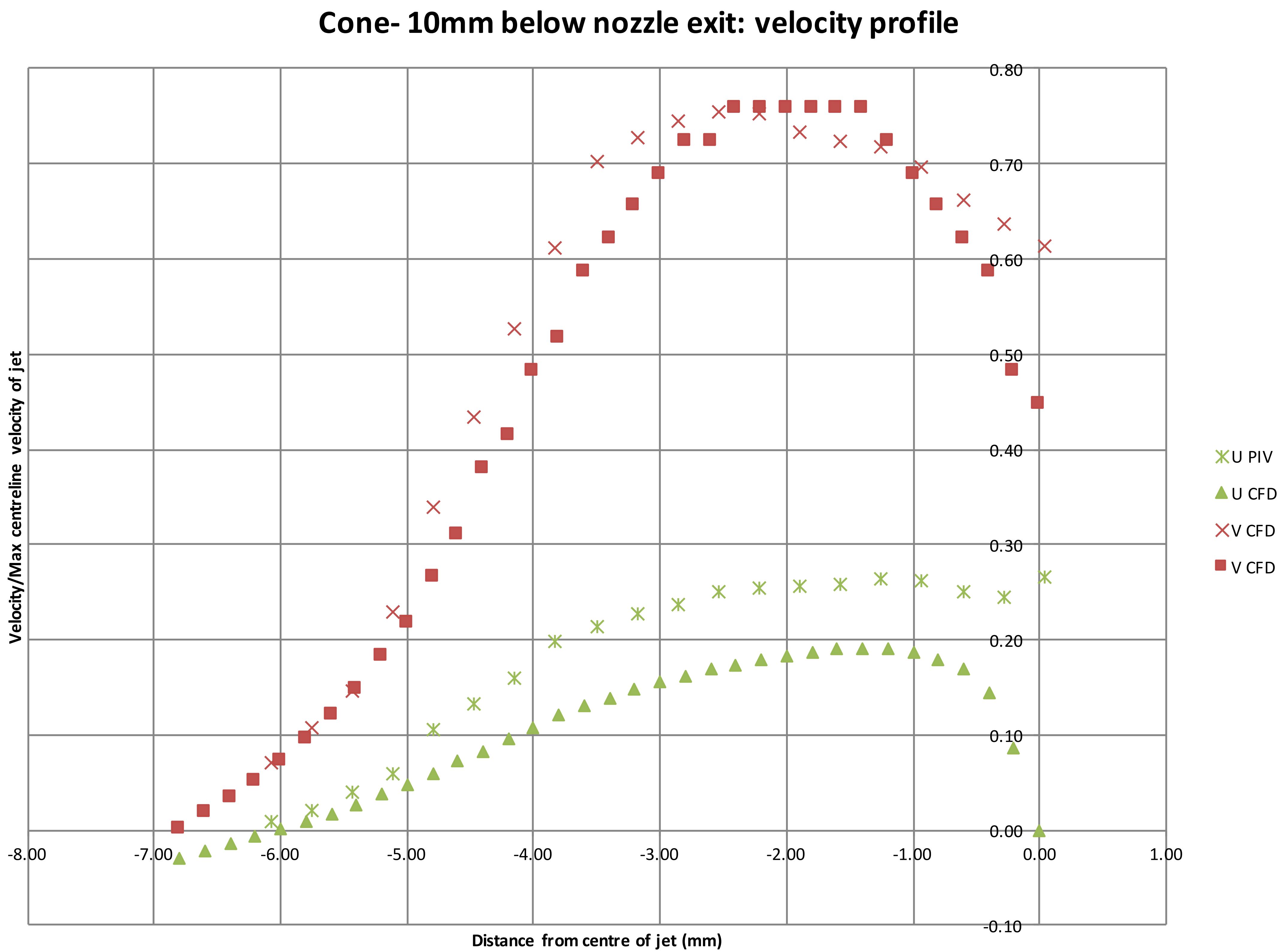
PIV data
(same for
other slides)

Cone

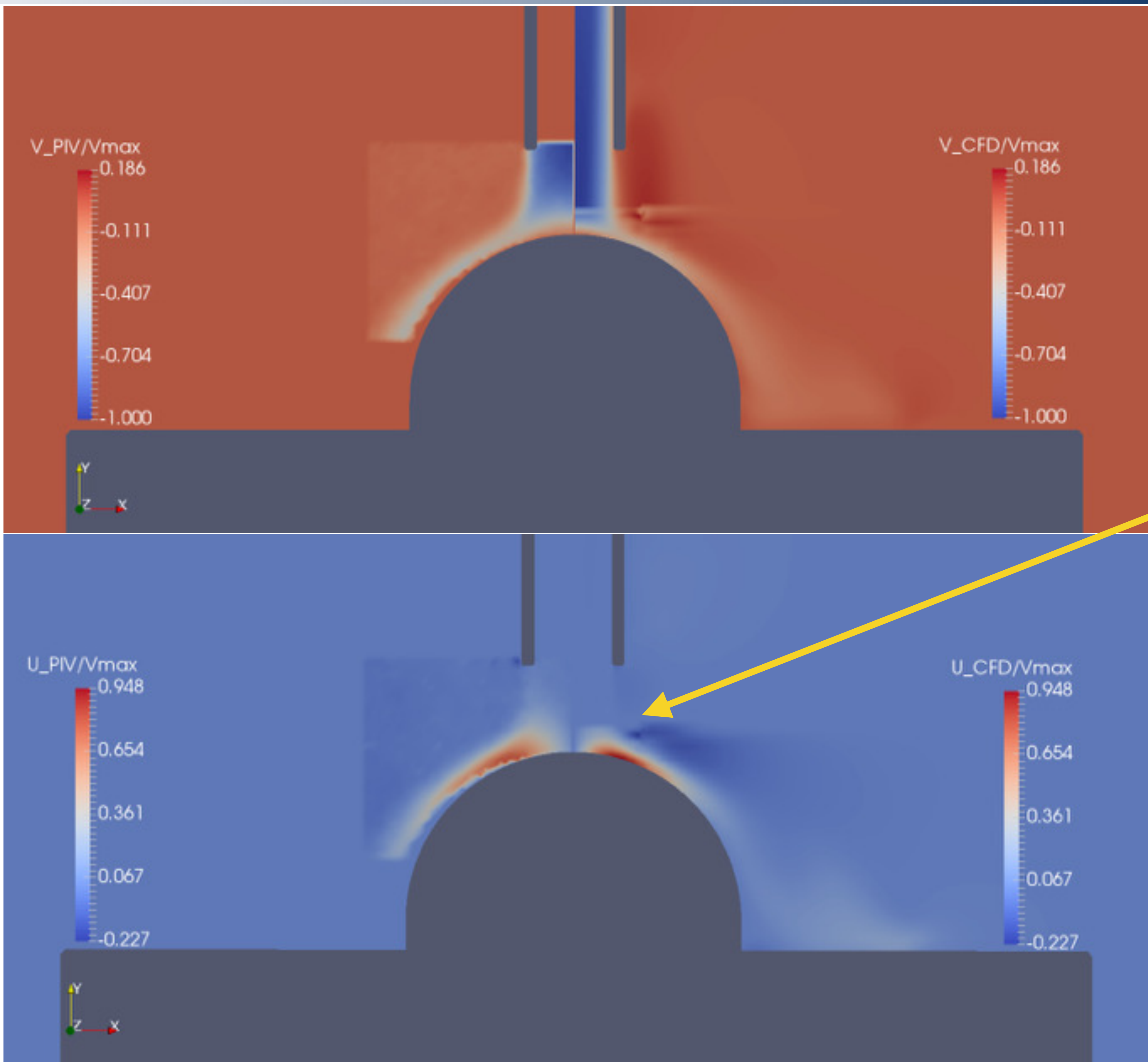


Cone- 5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Cone

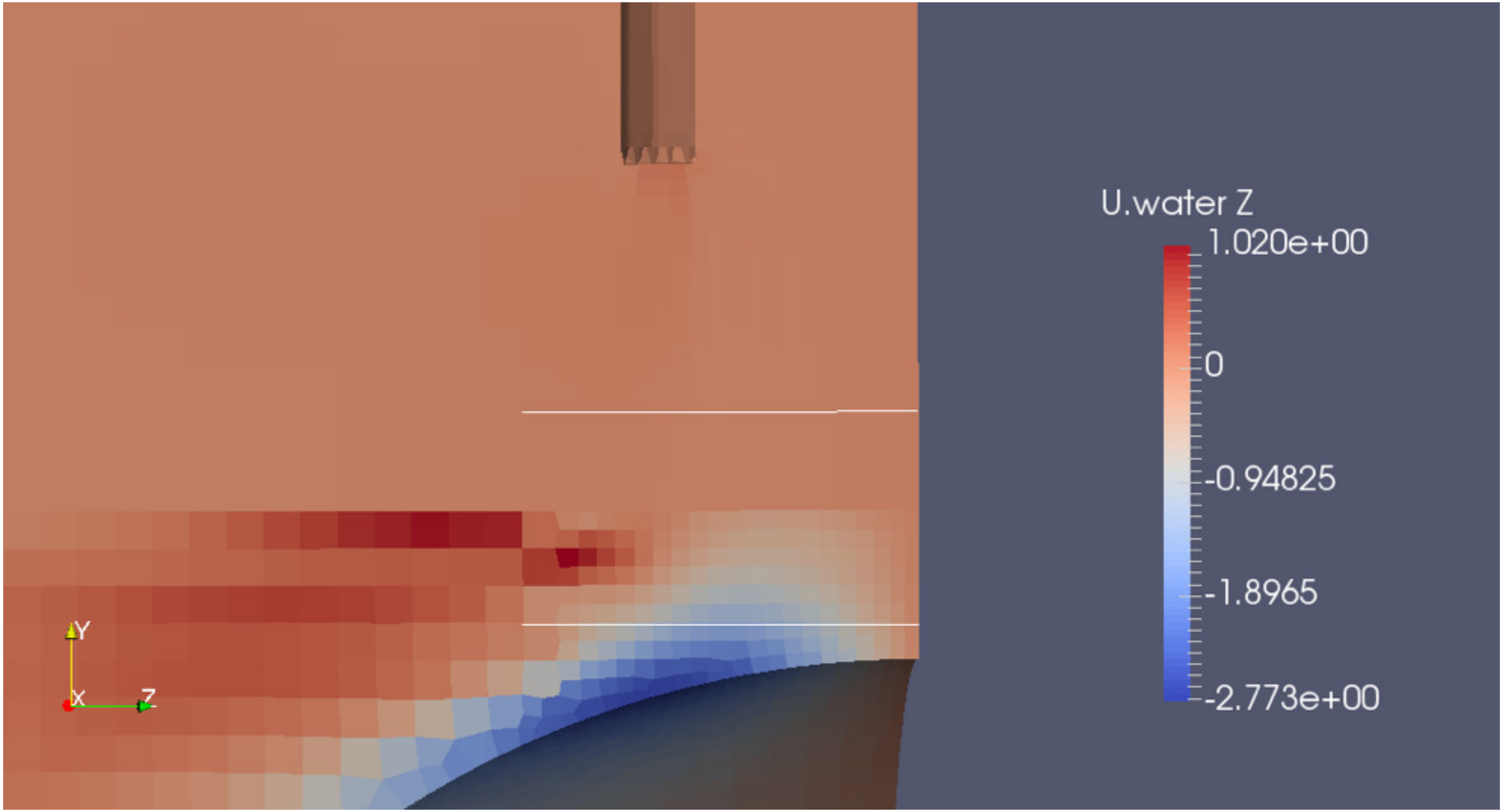

Cone- 10mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
0.80
0.70
0.60
0.50
0.40
0.30
0.20
0.10
0.00
-0.10
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
U PIV
U CFD
V CFD
V CFD

Hemisphere velocity contours



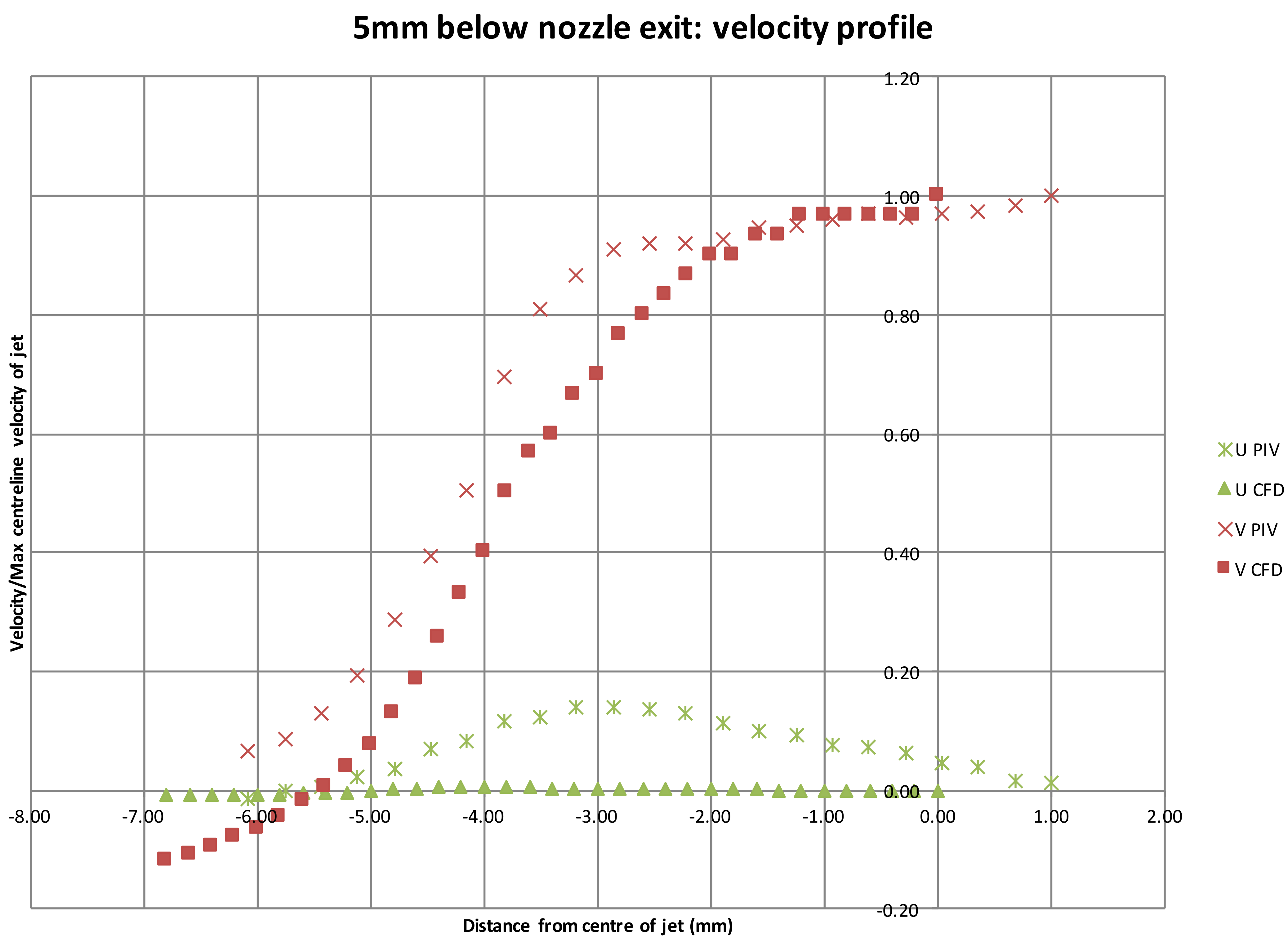
Not interpolating ‘U’ component correctly upstream

Error in Hemisphere



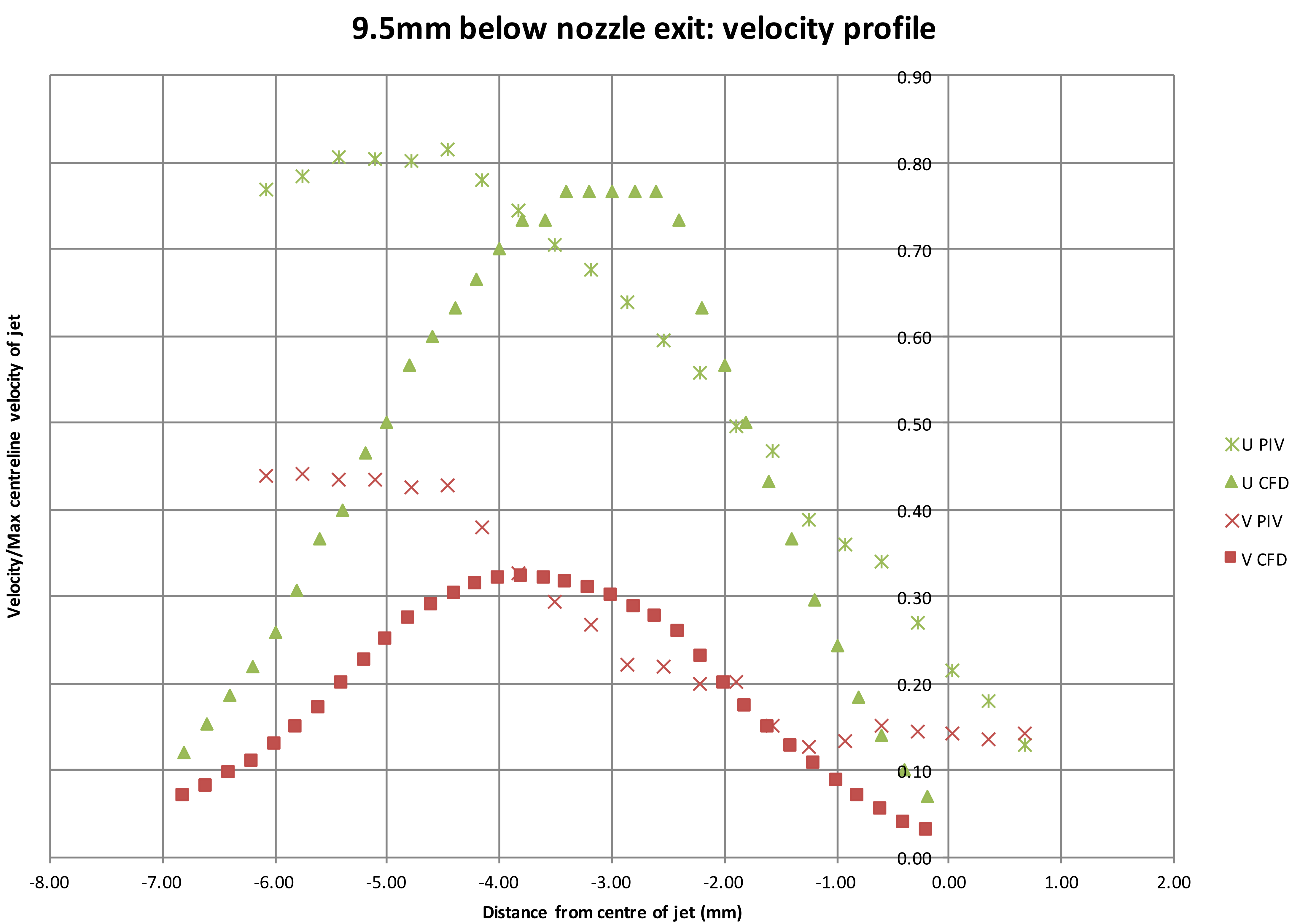
U.water Z is the horizontal velocity component
There is almost no UZ in region0

Hemisphere

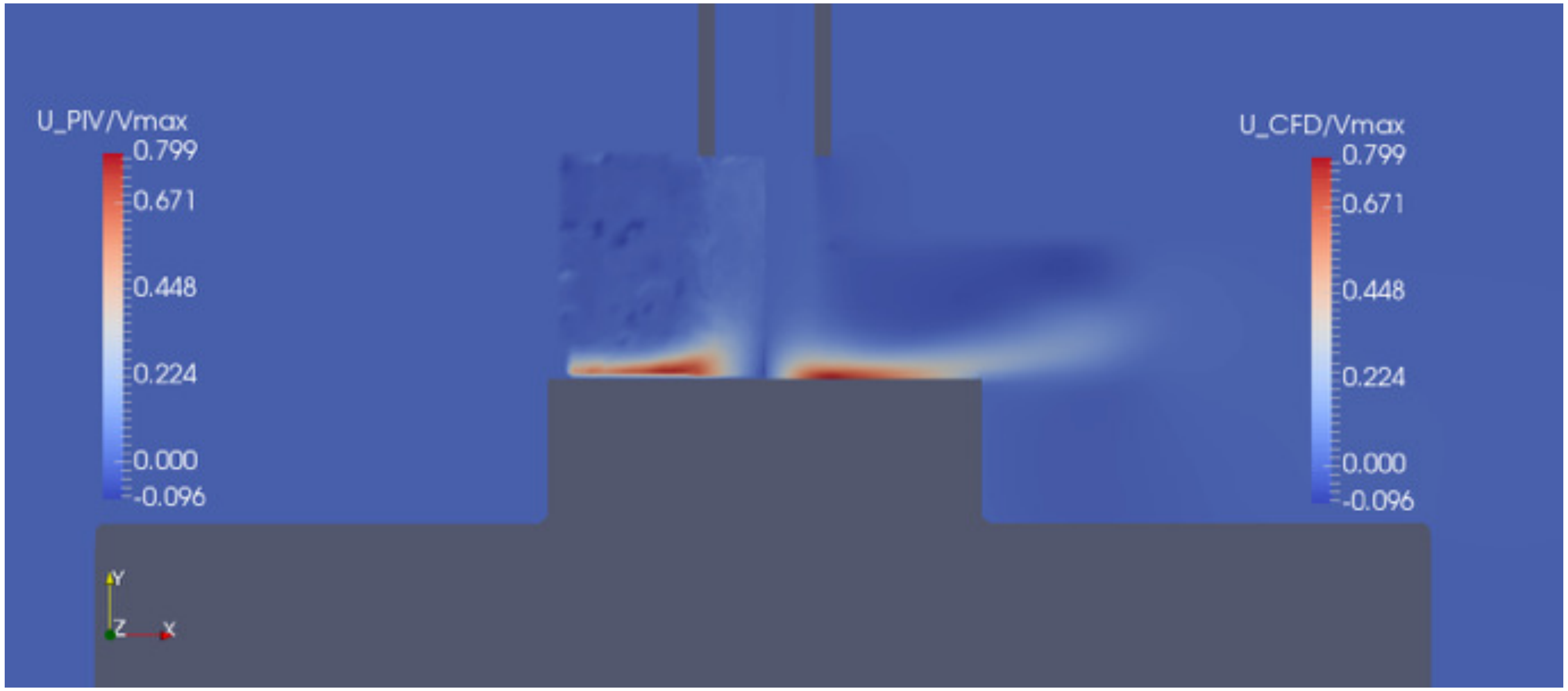

5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Hemisphere

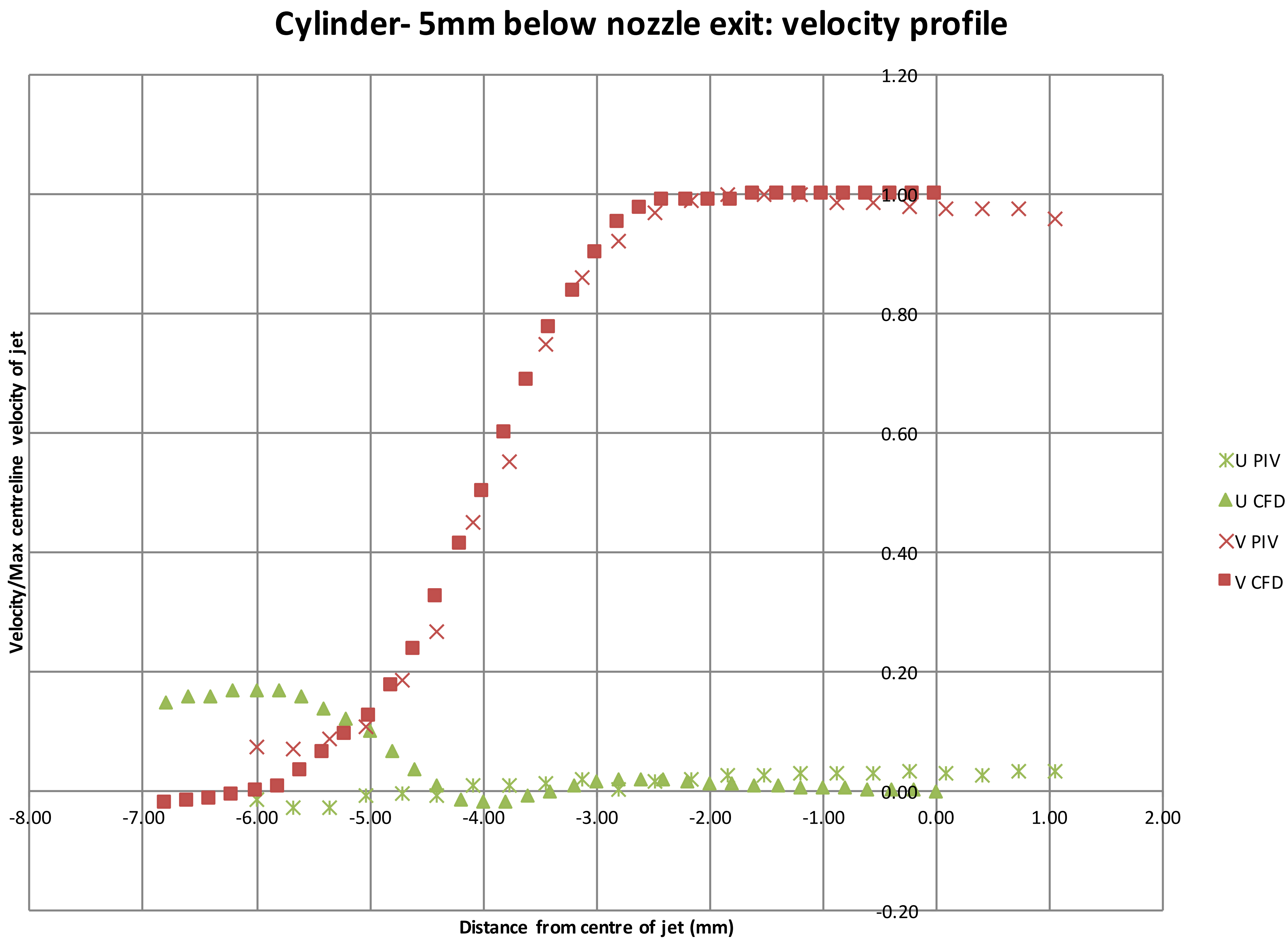
University of Strathclyde Glasgow



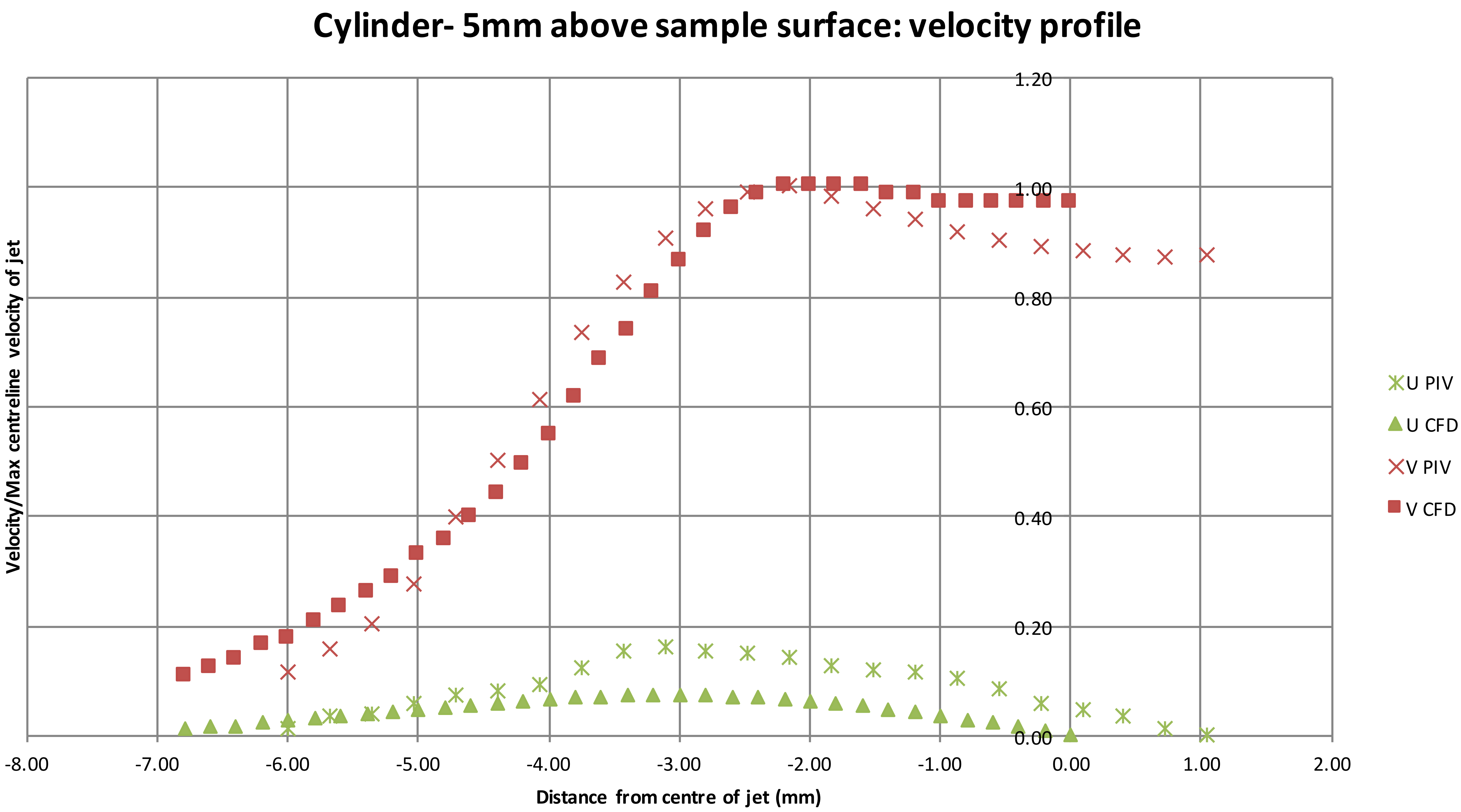
Cylinder velocity contours



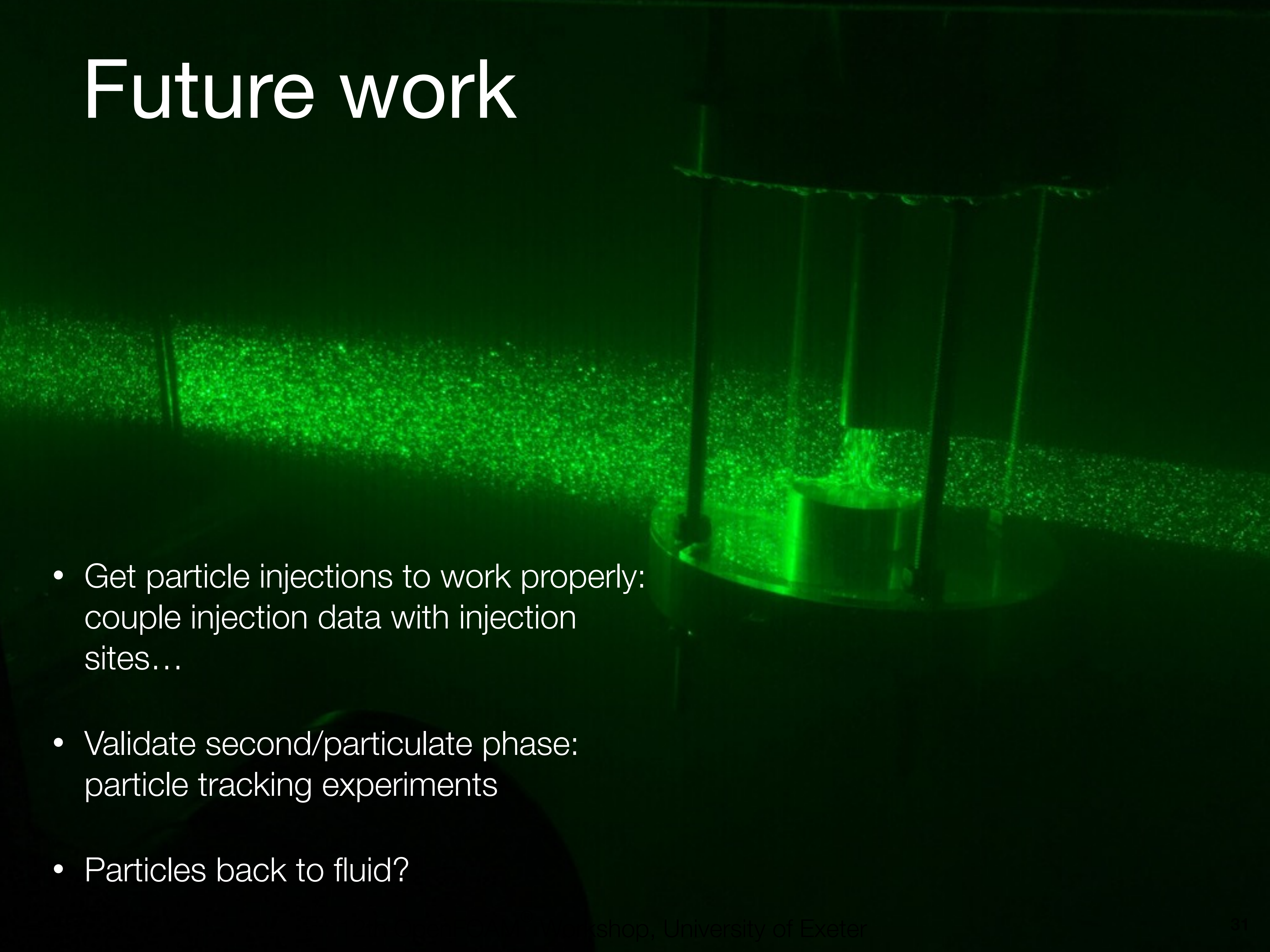
Cylinder


Cylinder- 5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Cylinder


Cylinder- 5mm above sample surface: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Future work



- Get particle injections to work properly: couple injection data with injection sites…

- Validate second/particulate phase: particle tracking experiments

- Particles back to fluid?

Conclusion

Work still in progress but…

- Fluid phase shown to work on different geometries
- Solver should dramatically reduce computational time compared to pure EL
- Particle data should still be present near walls, where required
- Enable better design of mining equipment

Thank you

alasdair.mackenzie.100@strath.ac.uk

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