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

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

Ball mill video

Impeller

Before

After

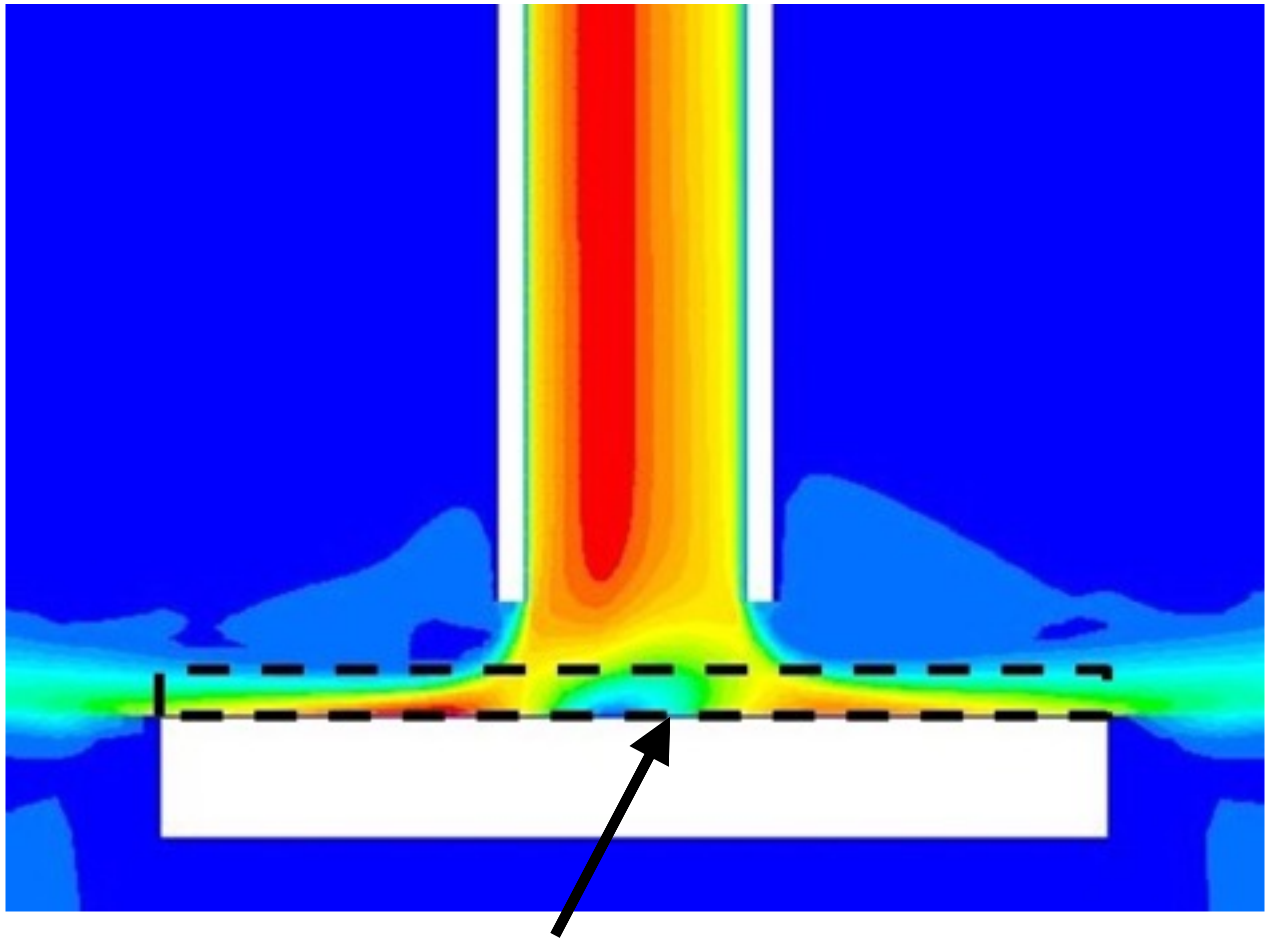
It could be as little as 2 weeks of continuous running for this to happen

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

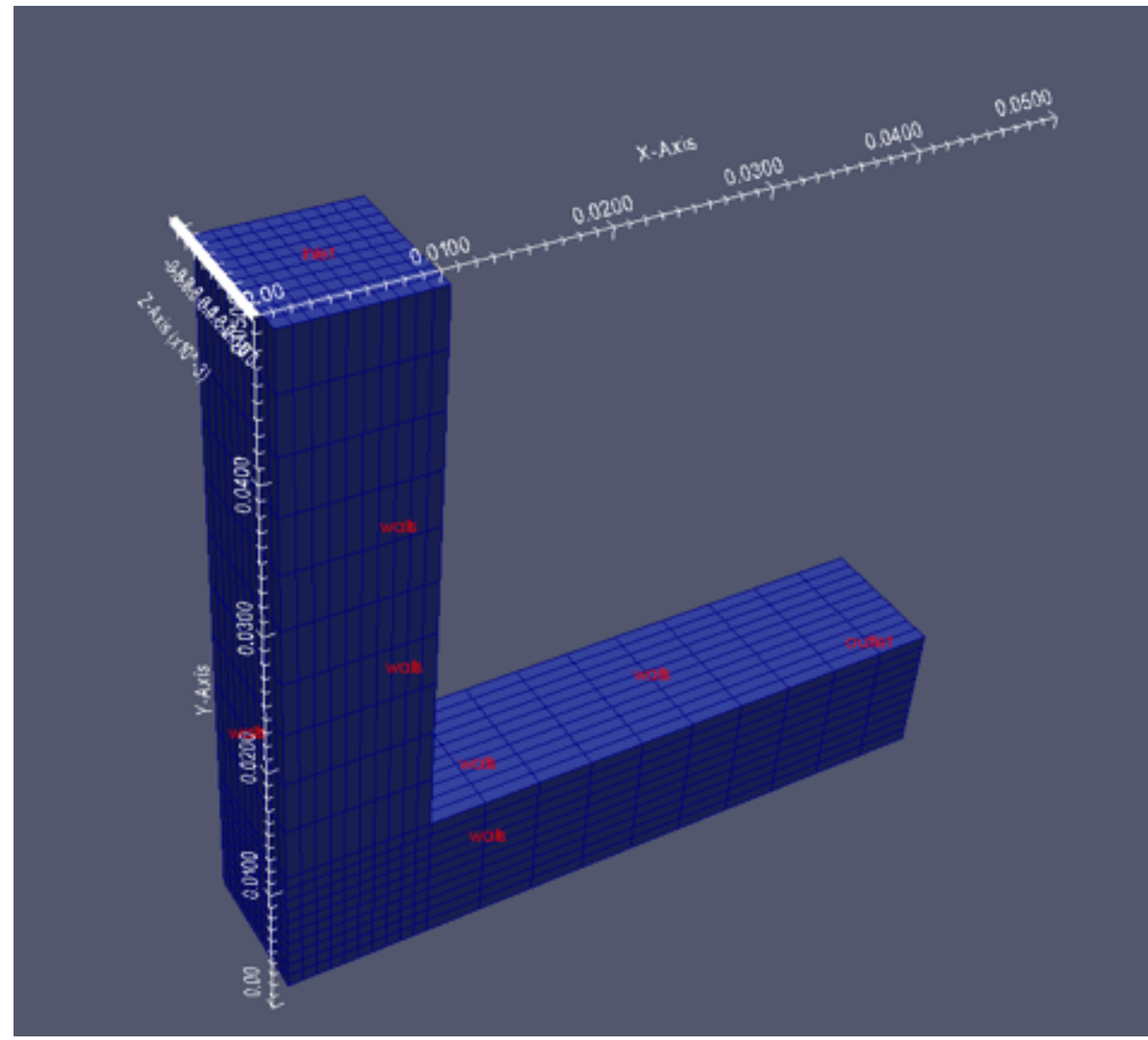


Dotted region where particles are necessary for impact data

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: <u>http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2016/AlasdairMackenzie/tutorial1.pdf</u>



Geometry shown with sizes in metres

Description of Solvers

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

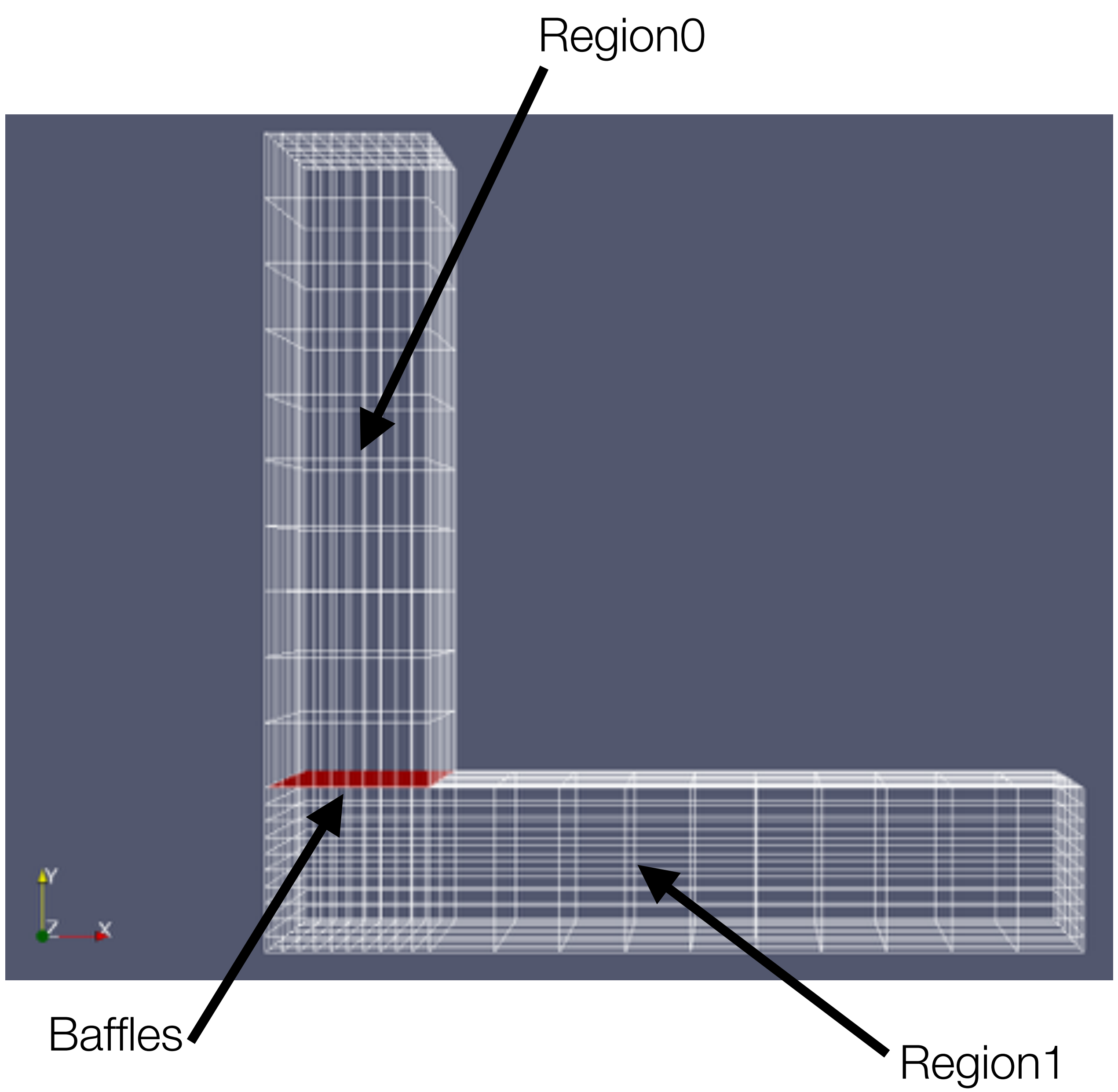
Combining the solvers

University of Strathclyde Glasgow

- 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…

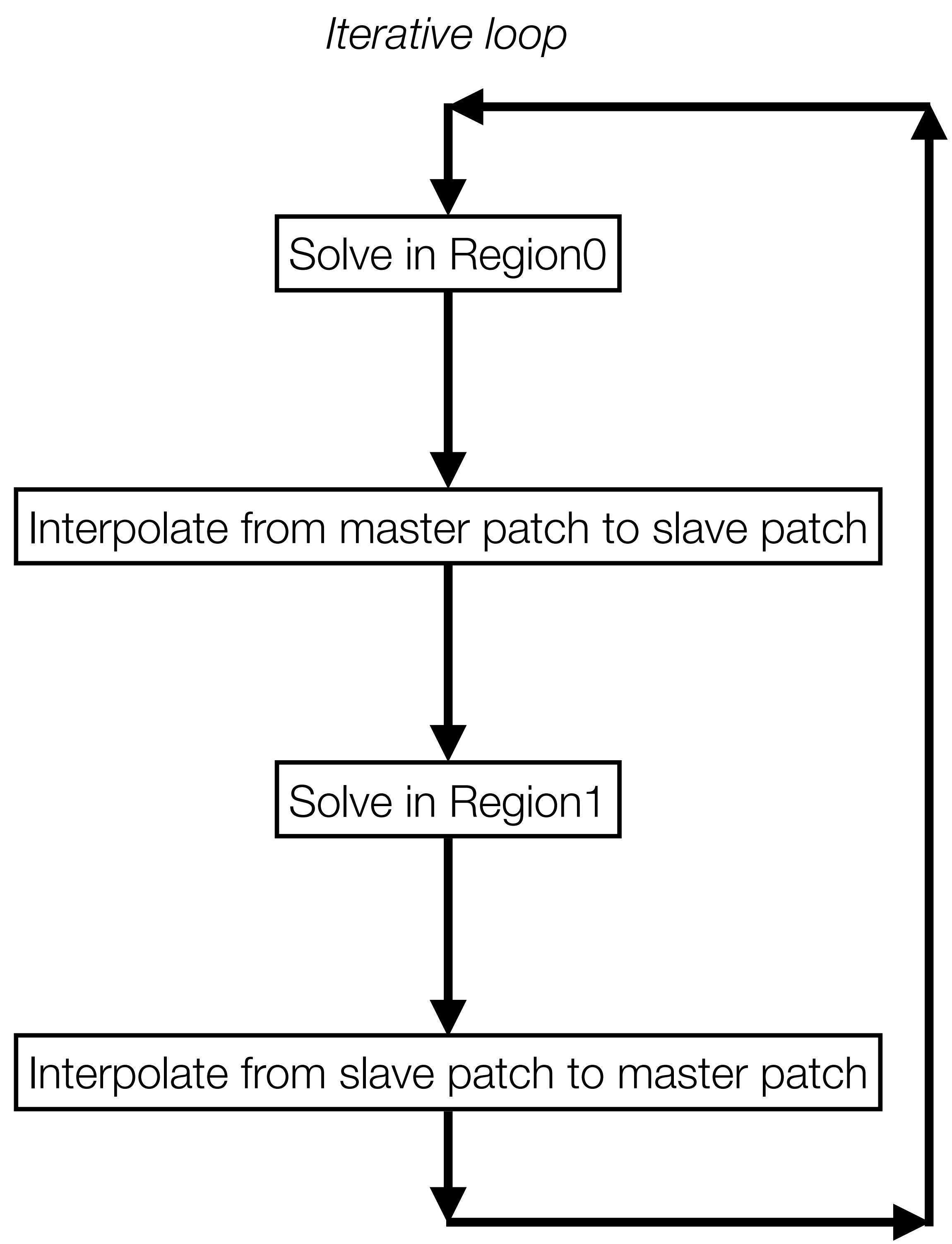
Baffles + Regions

- 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

- 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

```
/* (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

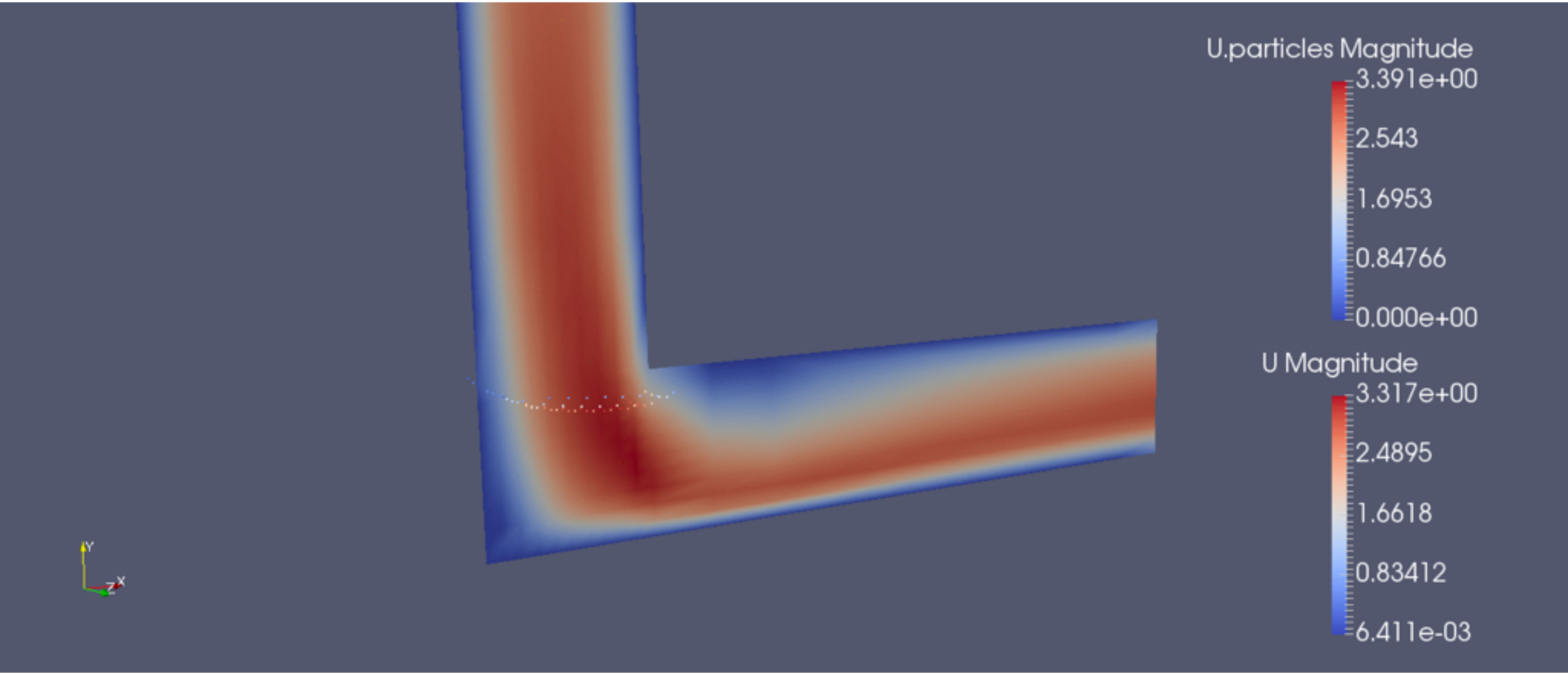
DPMFoam injection

```
    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.

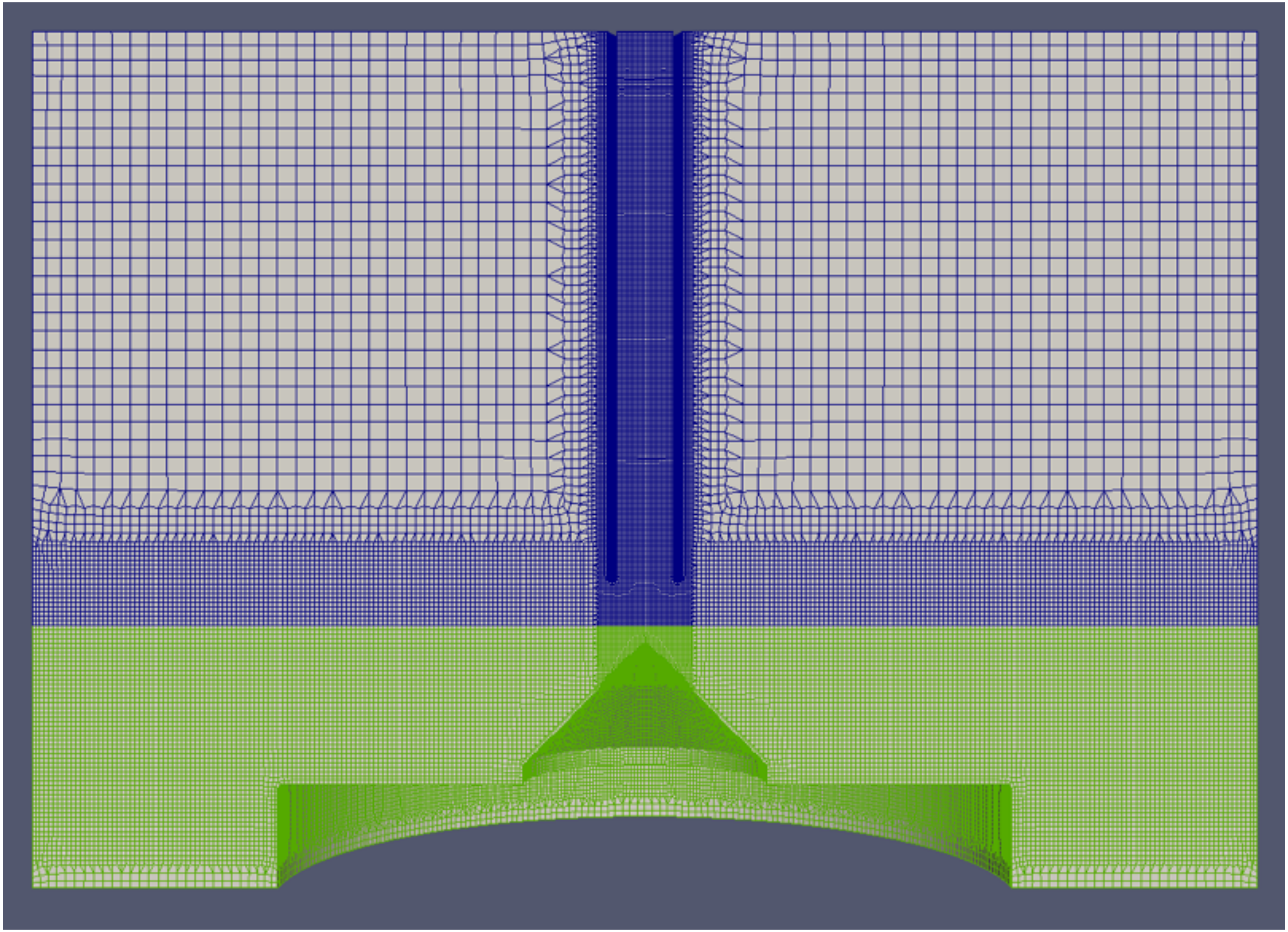
- Number of parcels/cell = (alpha particles * area of cell * normal velocity component to cell boundary face) / (volume of particle * number of particles/parcel * number of time-steps/second)

Velocity contours

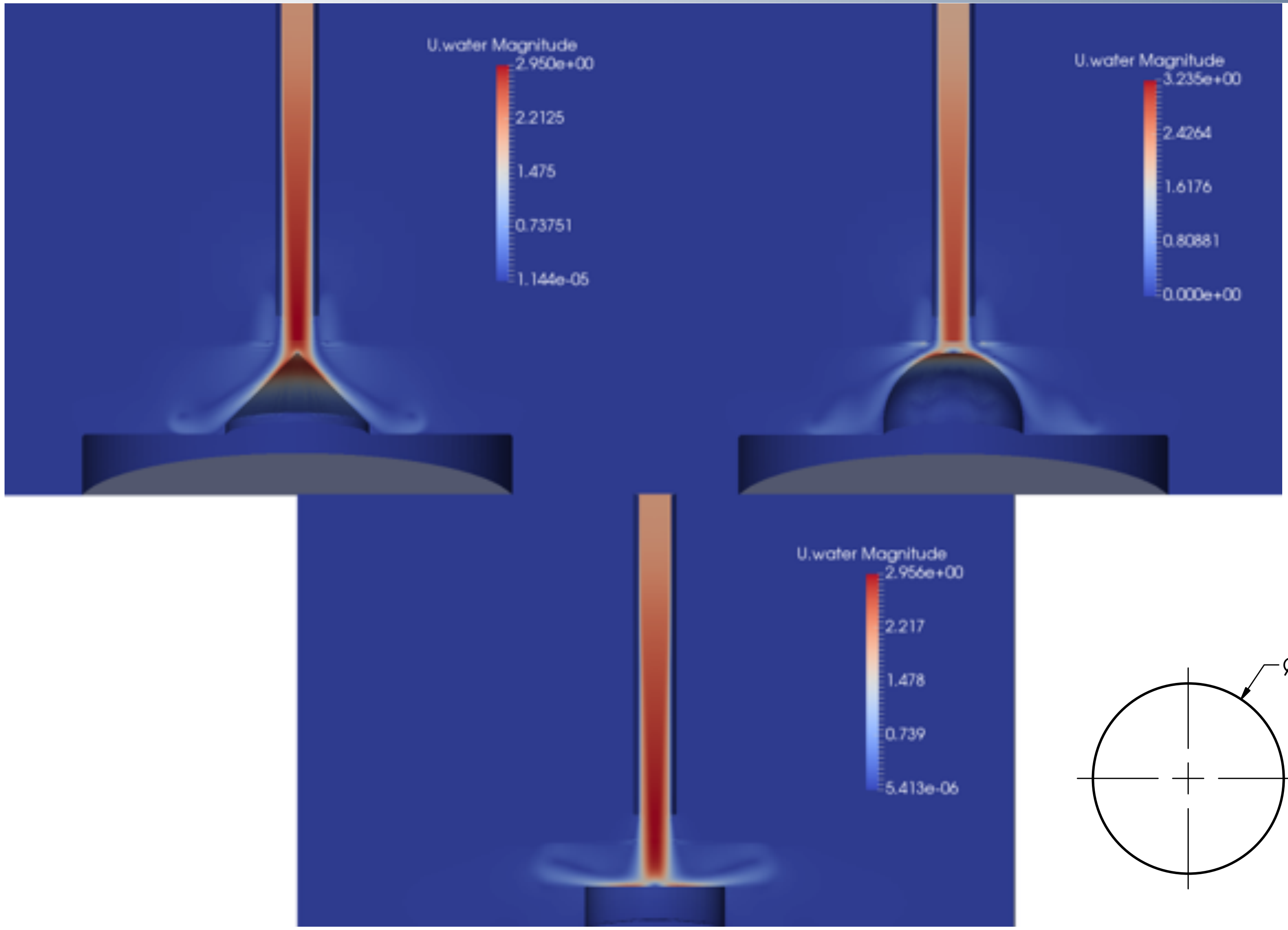


- 2D slice through Z normal. Particles injected from slave patch

Real geometry setup

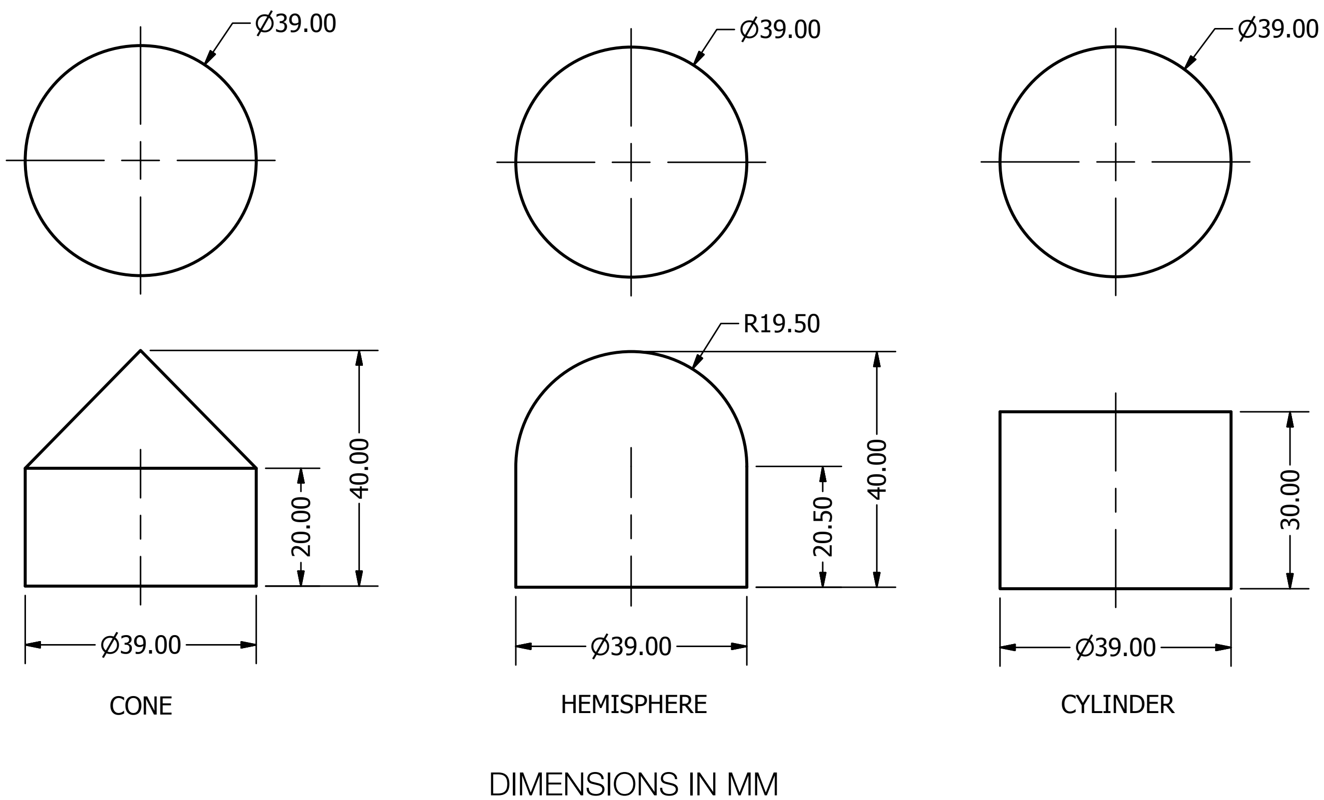


3 sample geometries



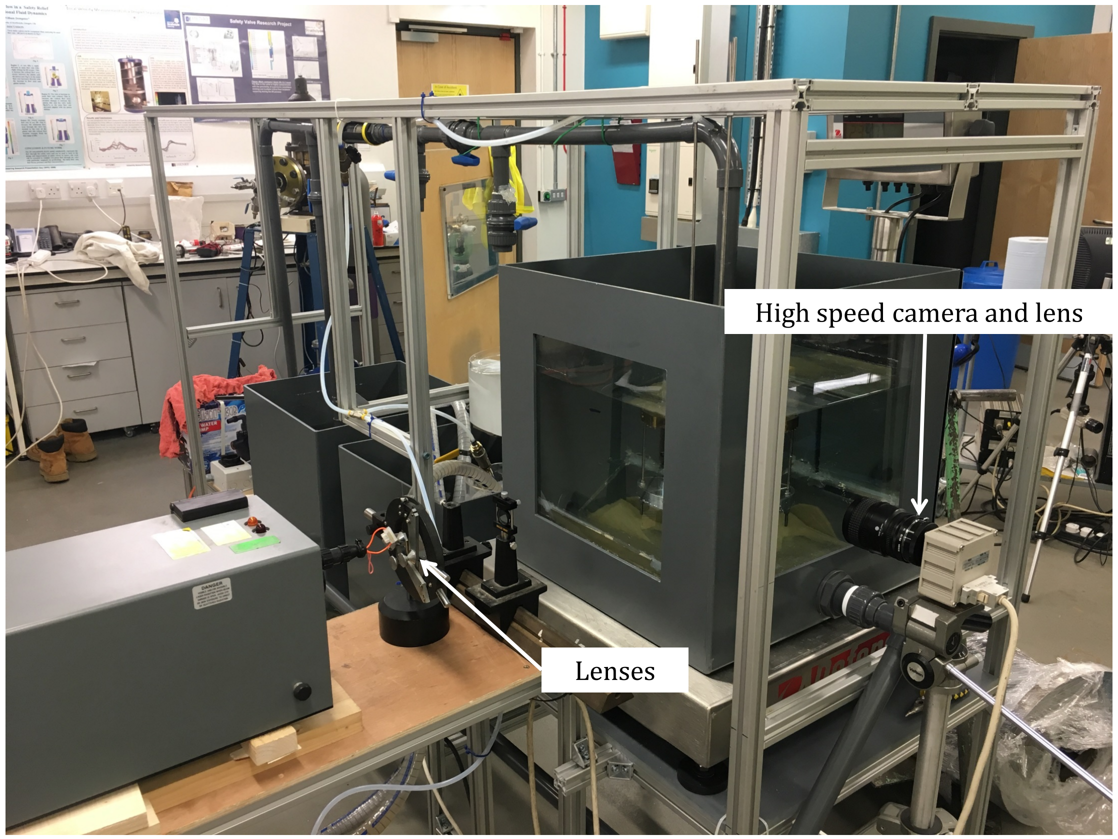
New solver was tested
on the shown geometries

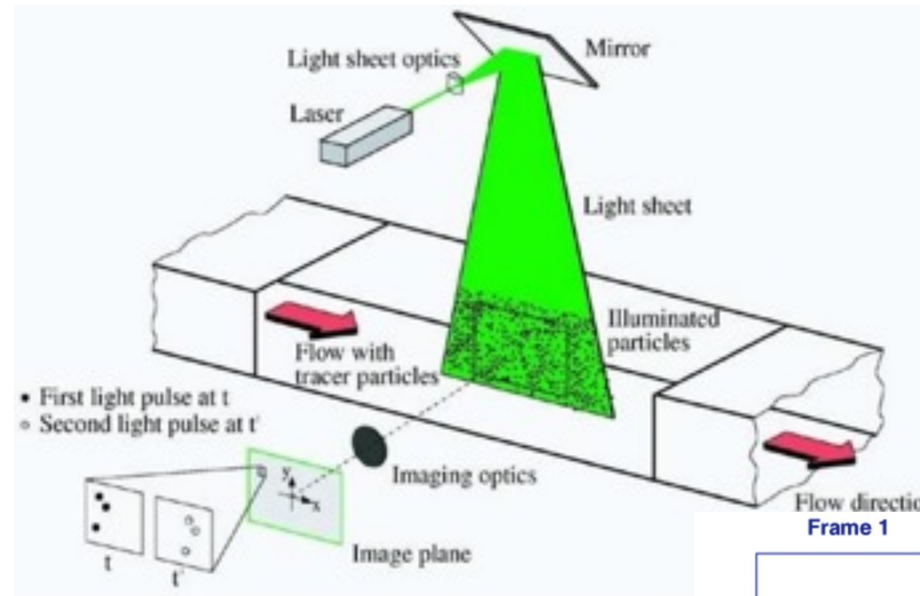
Particle Image Velocimetry
was carried out for validation

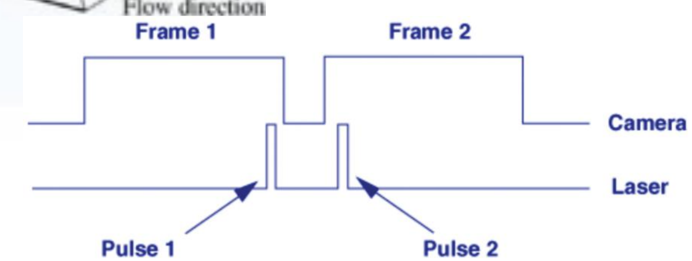


Mass flow inlet (~2m/s)
K-Omega SST turbulence model used
Only first phase compared **(so far)**

Experimental setup







<u>Particle Image Velocimetry</u>

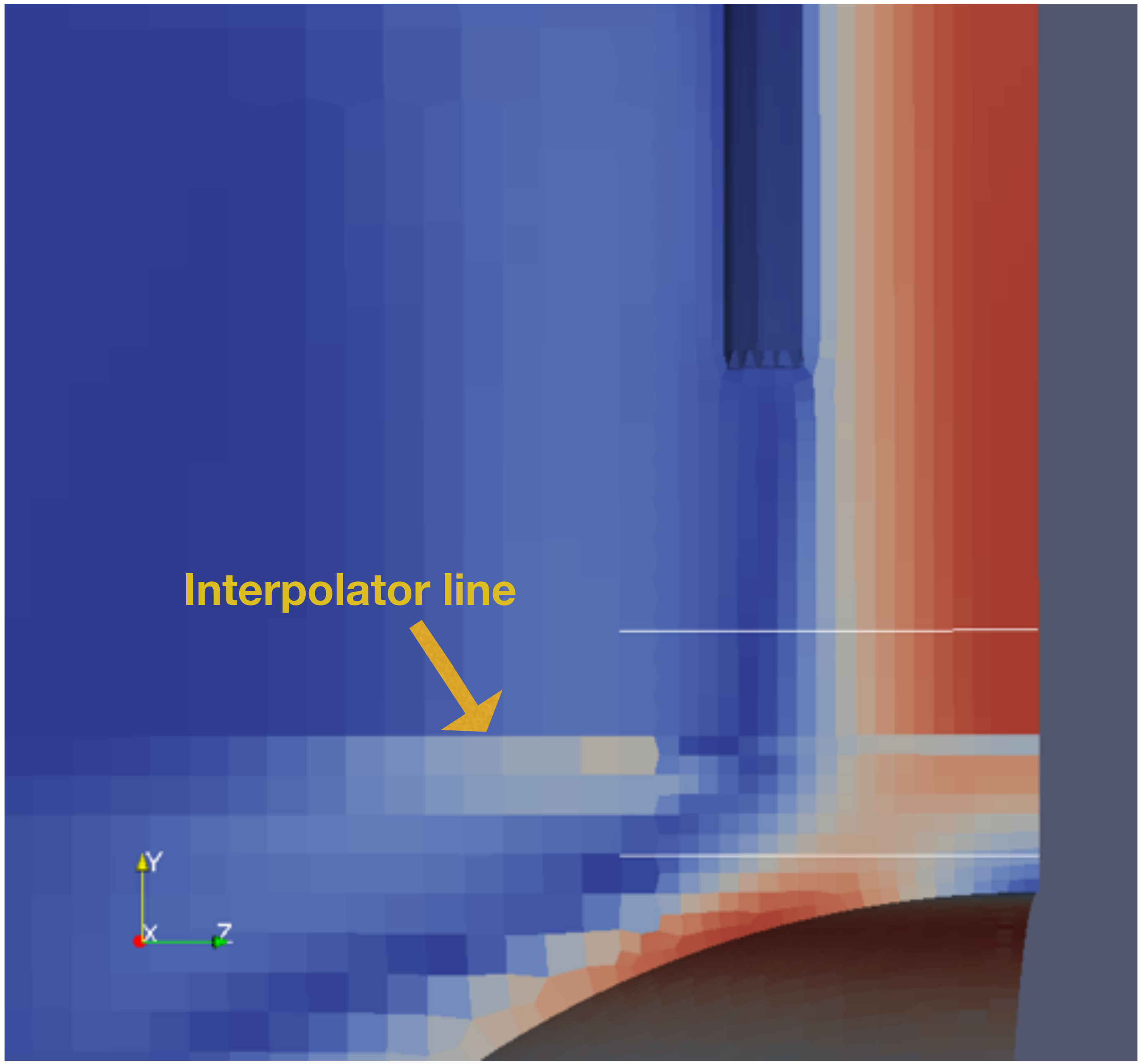
Frame straddling used by laser ΔT=67μs

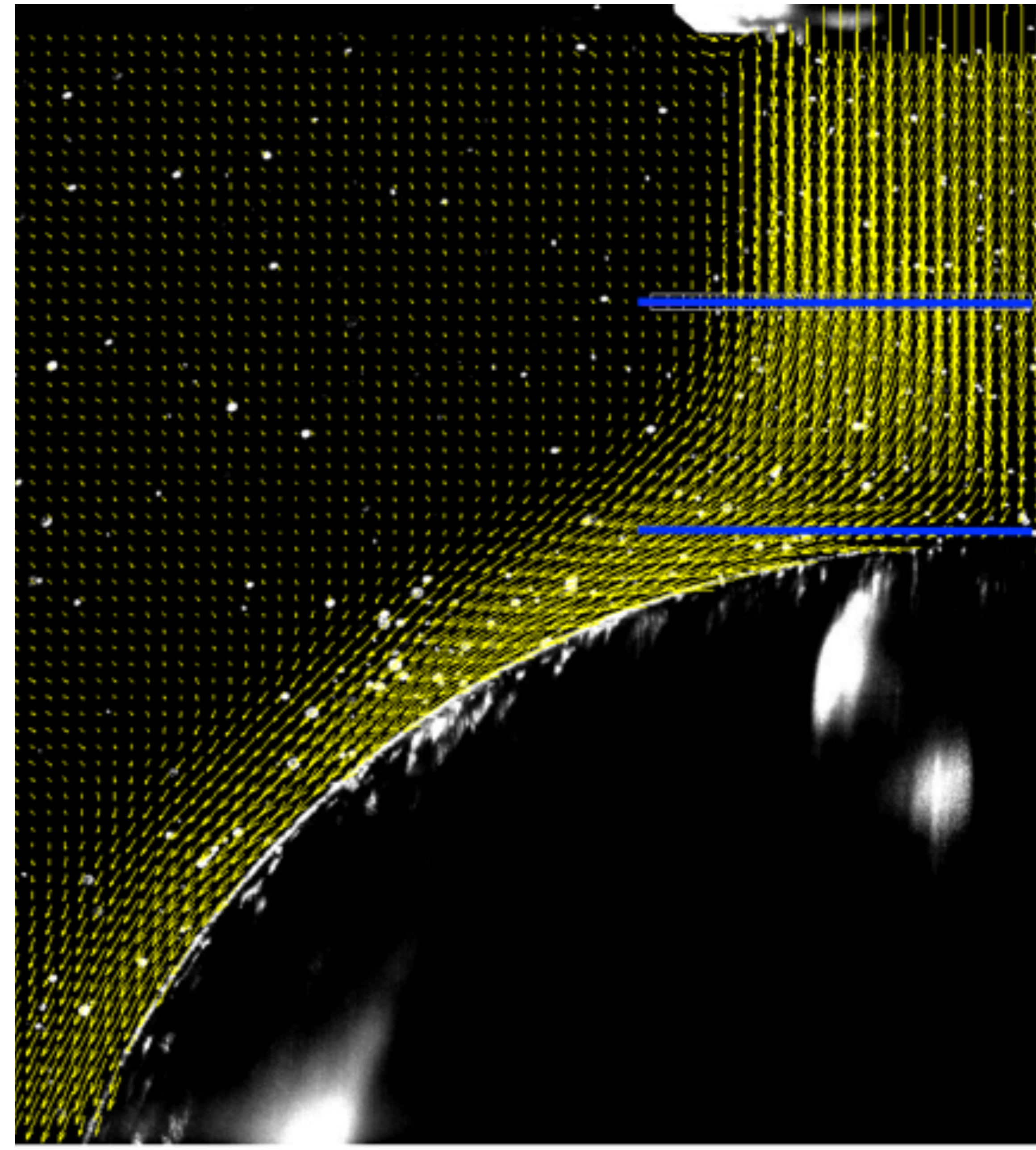
DantecDynamics laser system

250 images used, 125 image pairs

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

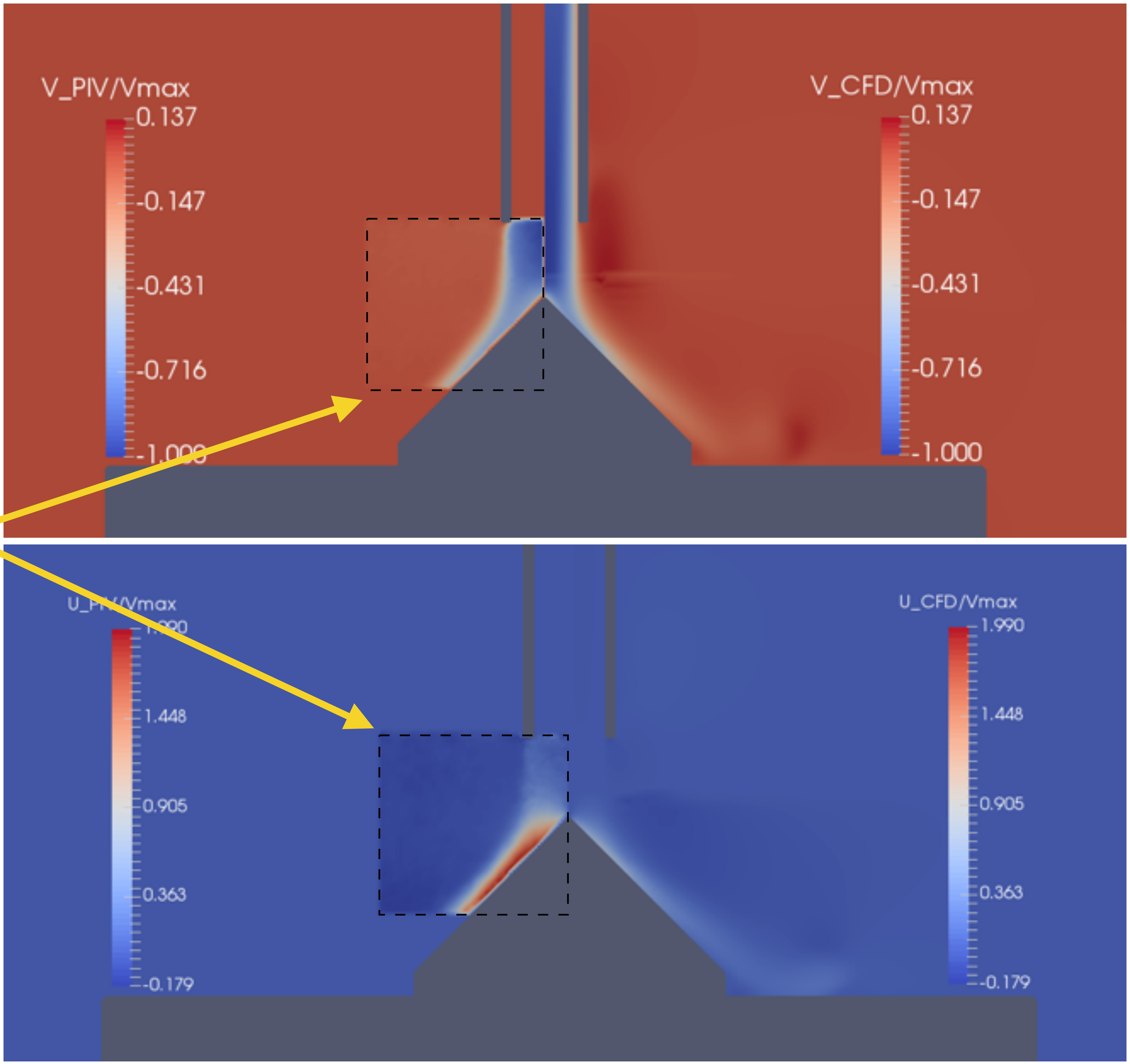
Comparison of data





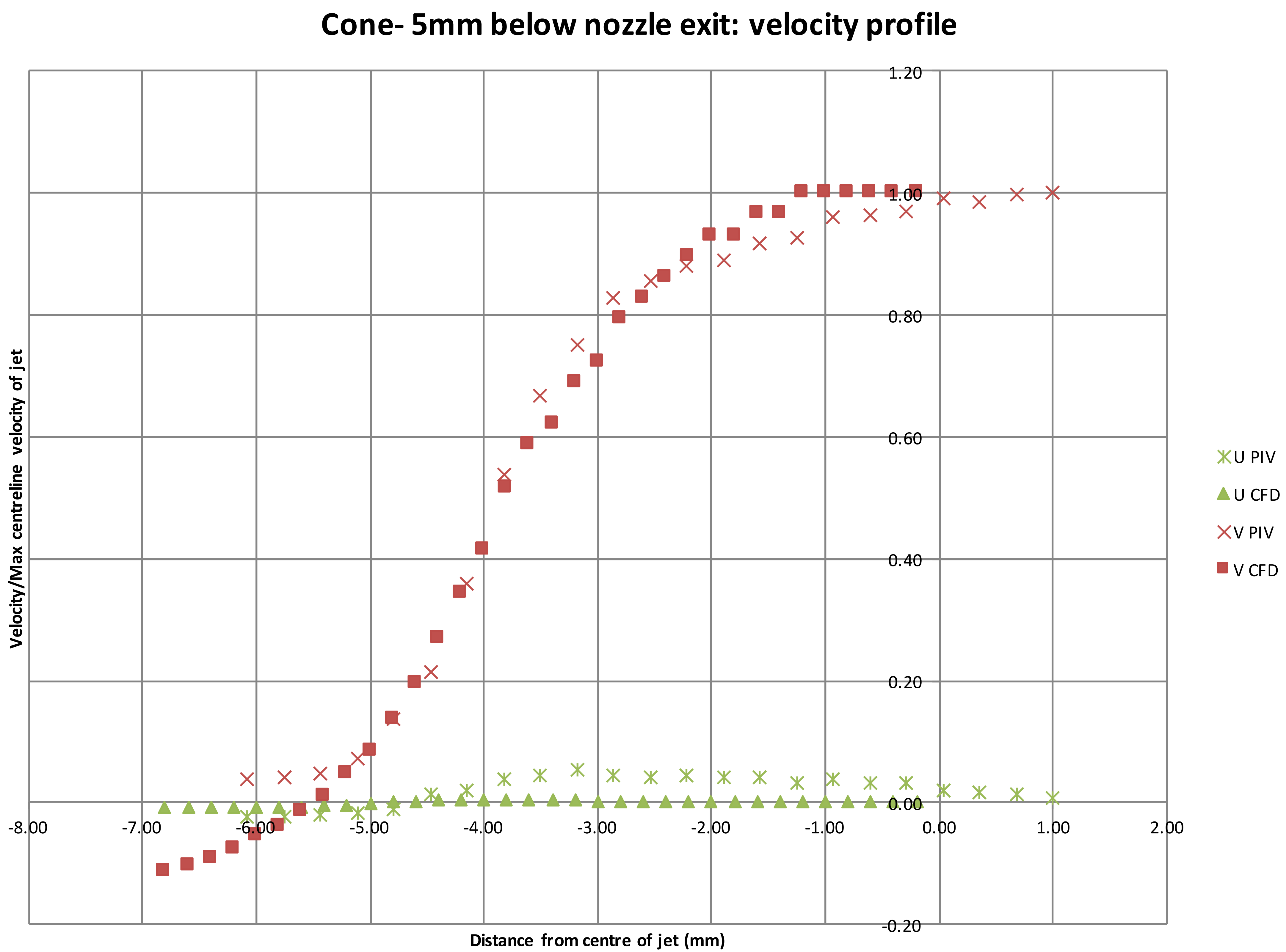
Lines show where data is taken from:
top is 5mm from nozzle, bottom is 9.5/10mm from nozzle
Interpolator is in between sample lines

Cone velocity contours



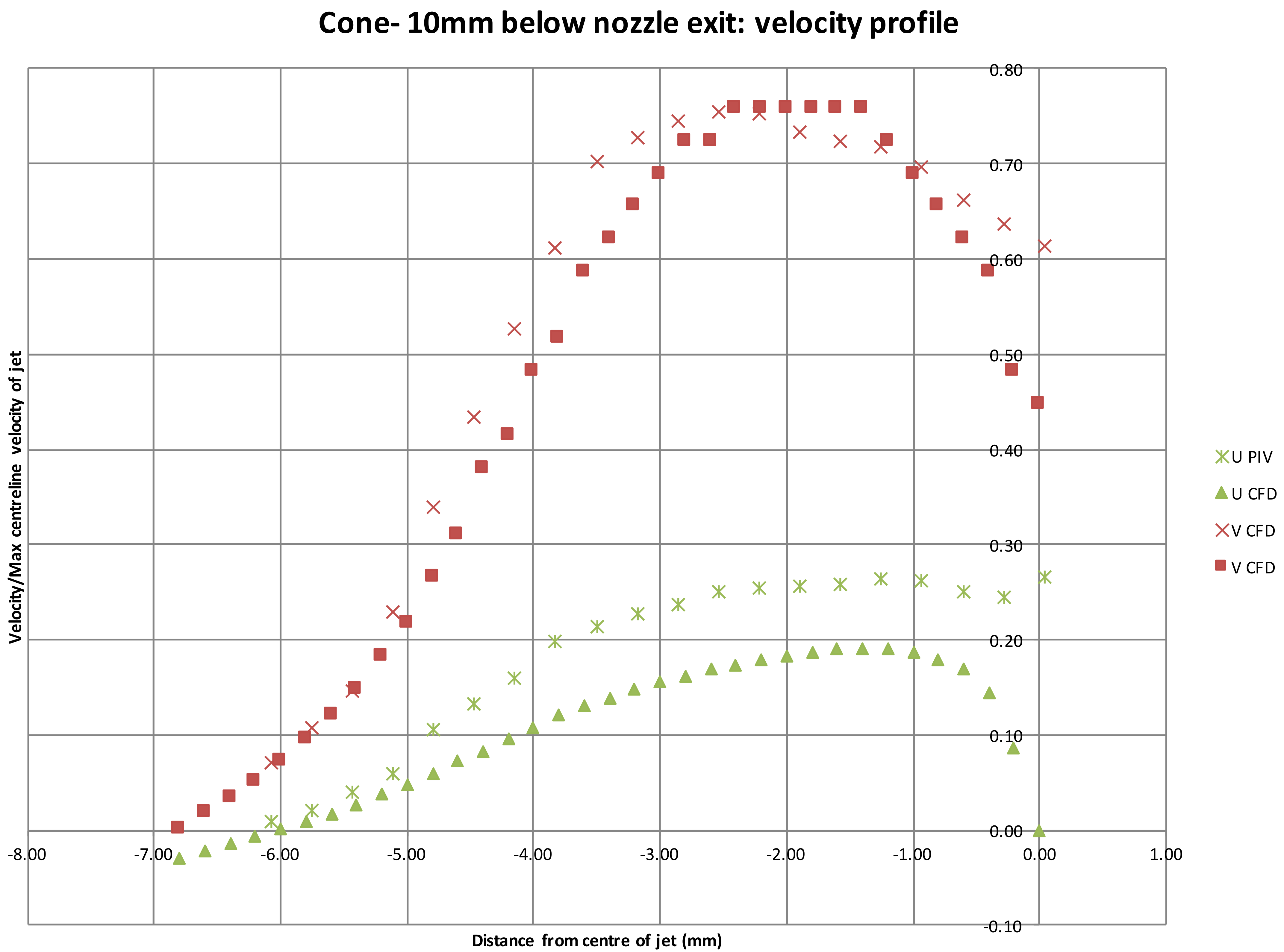
PIV data
(same for
other slides)

Cone

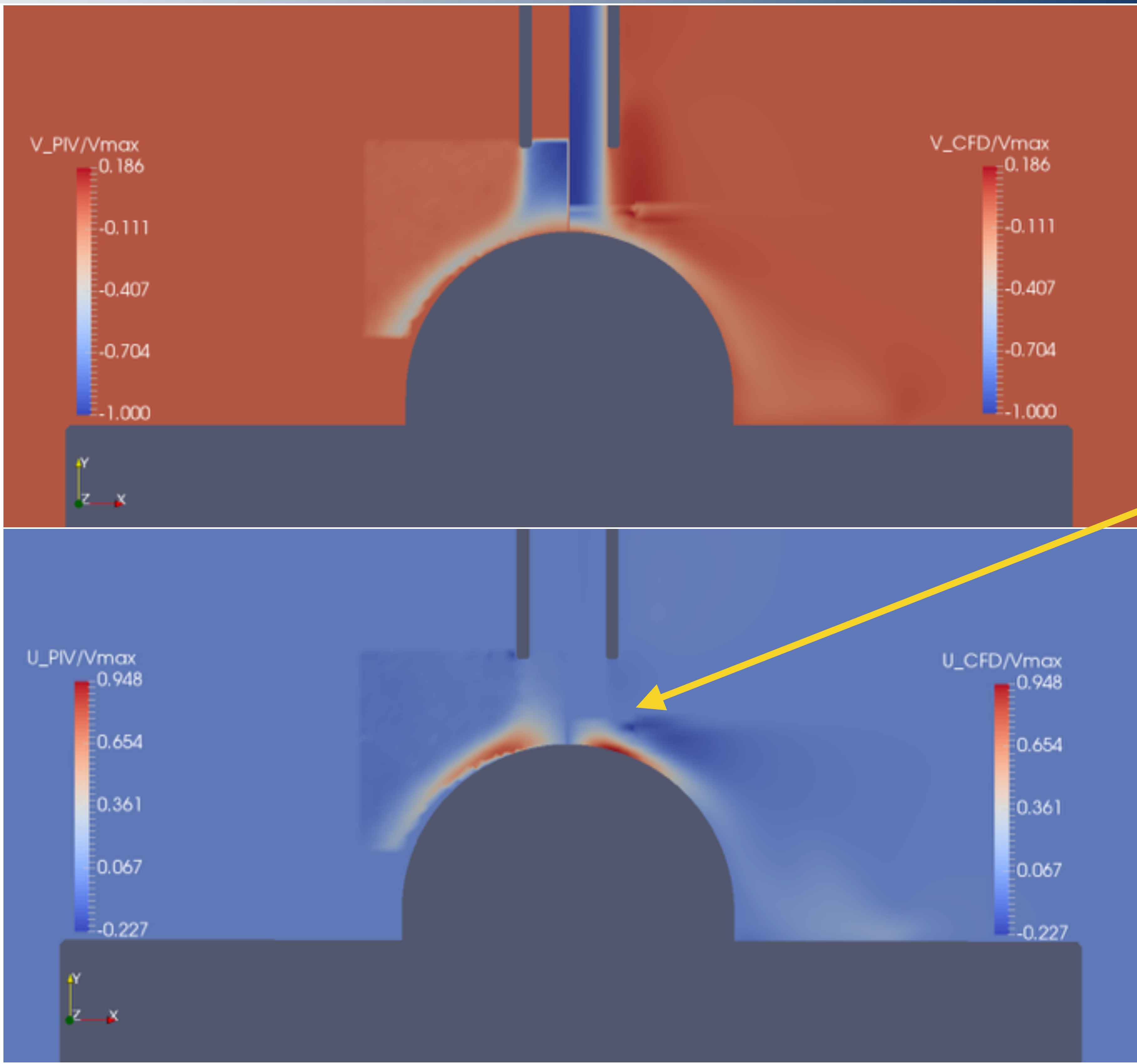


Cone- 5mm below nozzle exit: velocity profile
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
U PIV
U CFD
V PIV
V CFD

Cone

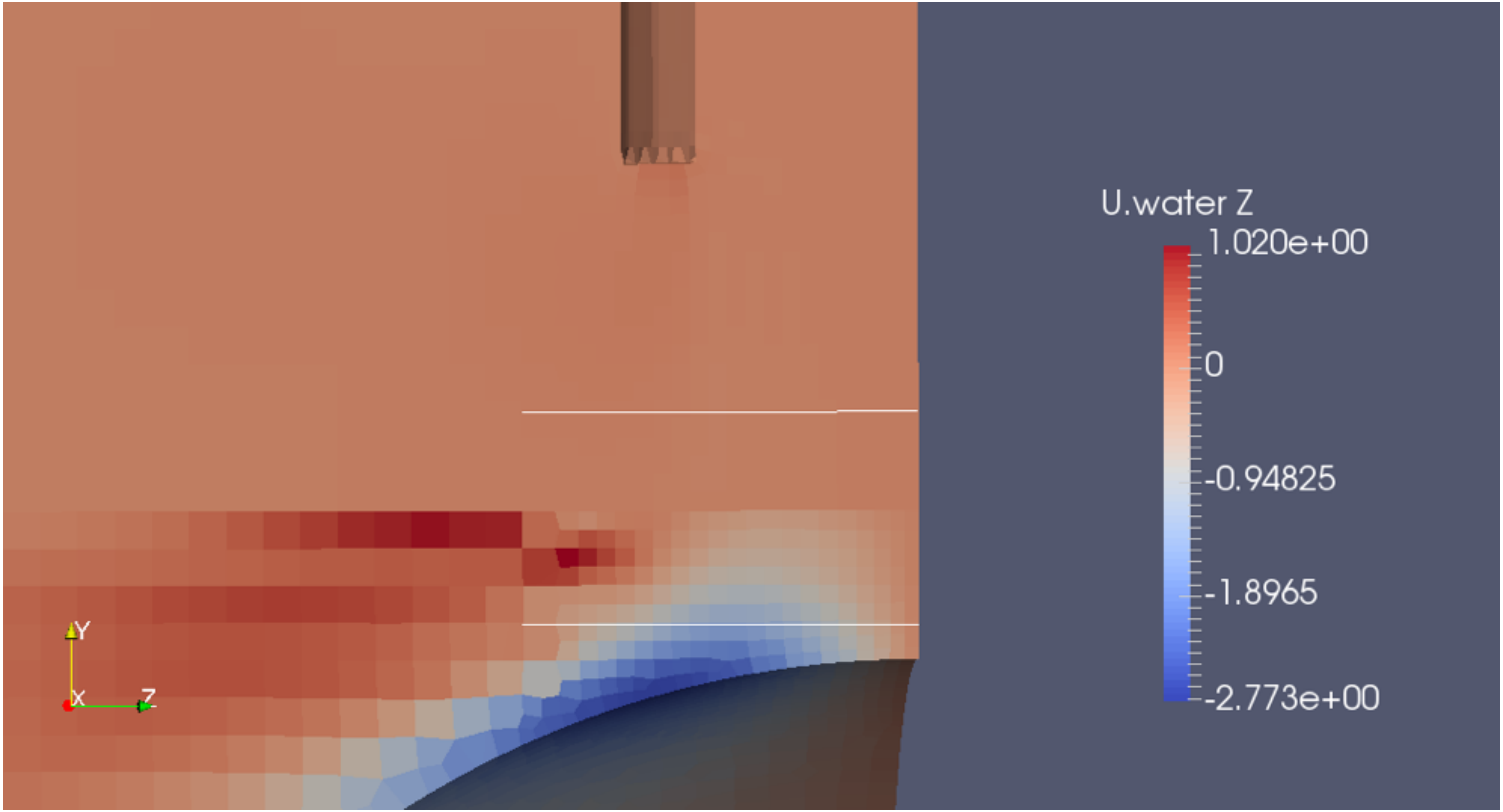

Cone- 10mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
0.80
0.70
0.60
0.50
0.40
0.30
0.20
0.10
0.00
-0.10
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
U PIV
U CFD
V CFD
V CFD

Hemisphere velocity contours



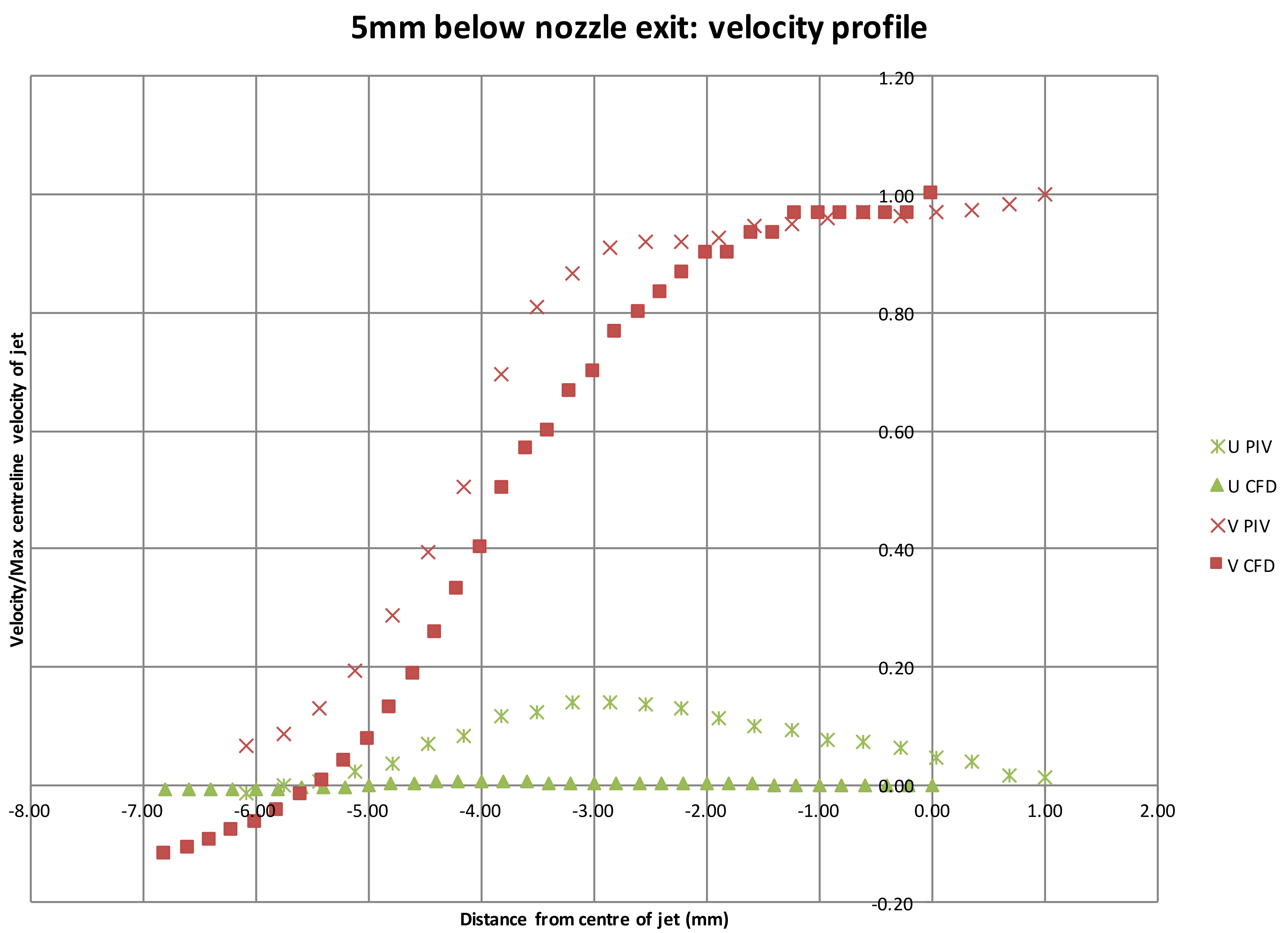
Not interpolating 'U' component correctly upstream

Error in Hemisphere



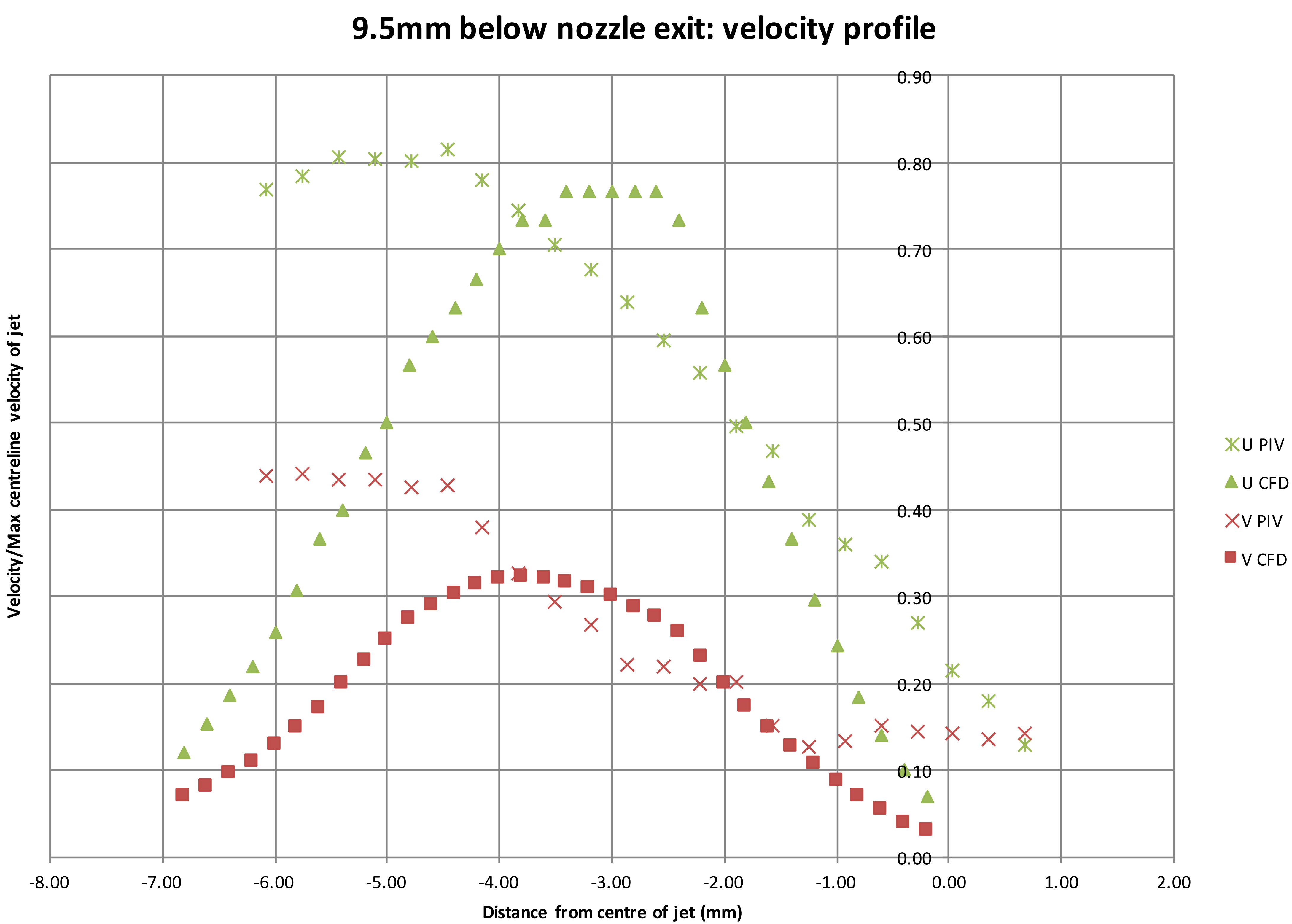
U.water Z is the horizontal velocity component
There is almost no UZ in region0

Hemisphere



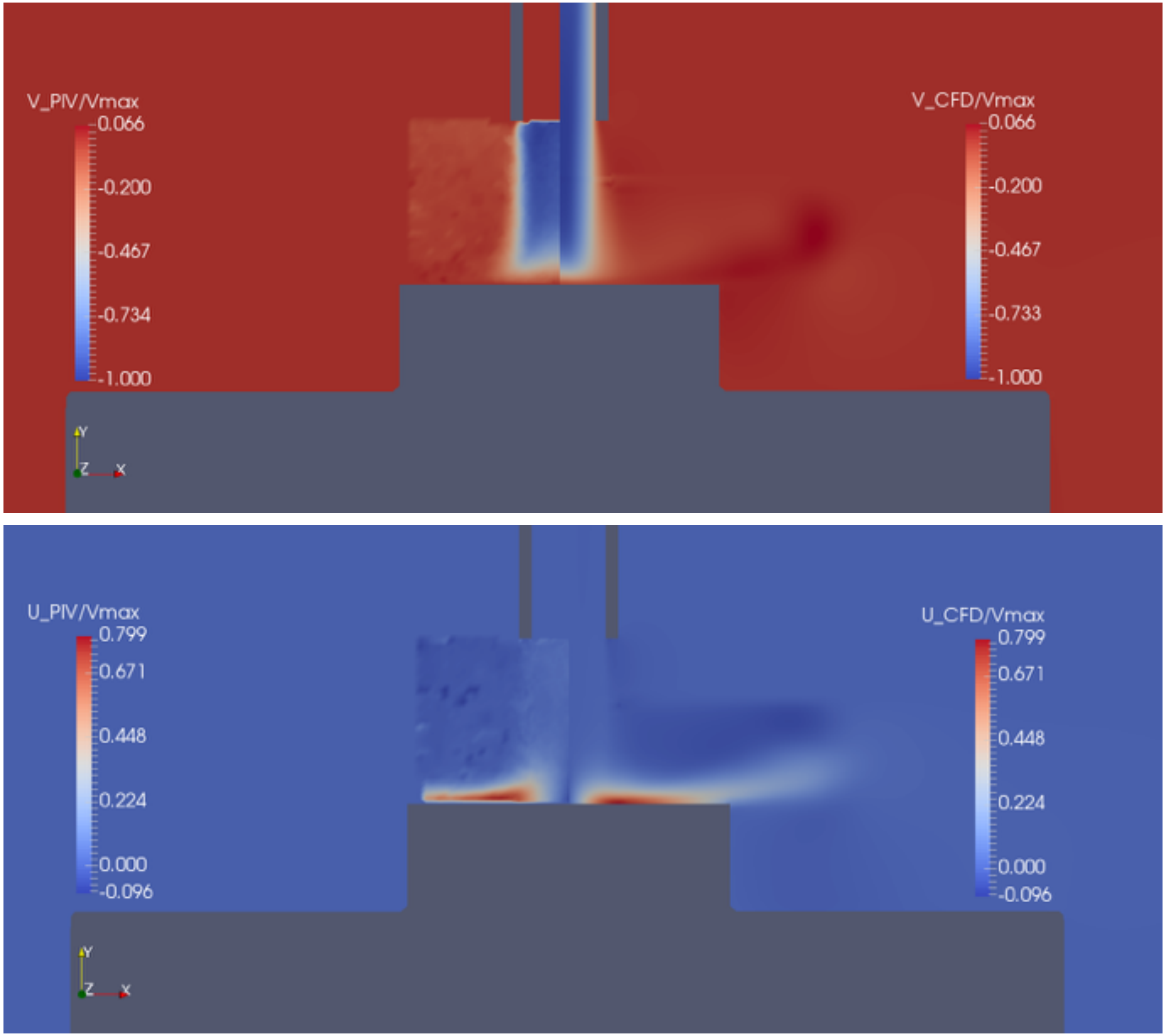
5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Hemisphere

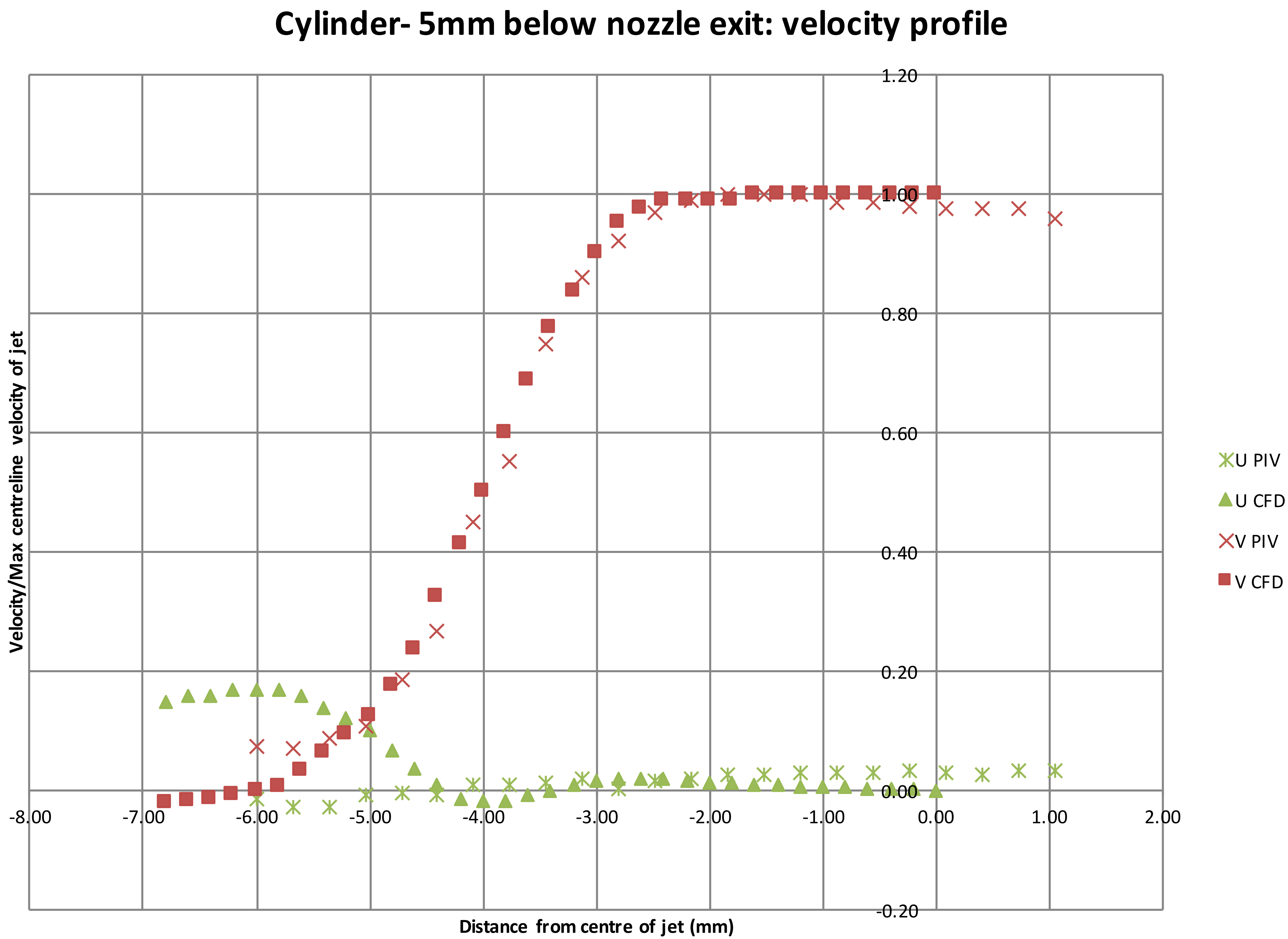


9.5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
0.90
0.80
0.70
0.60
0.50
0.40
0.30
0.20
0.10
0.00
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

Cylinder velocity contours

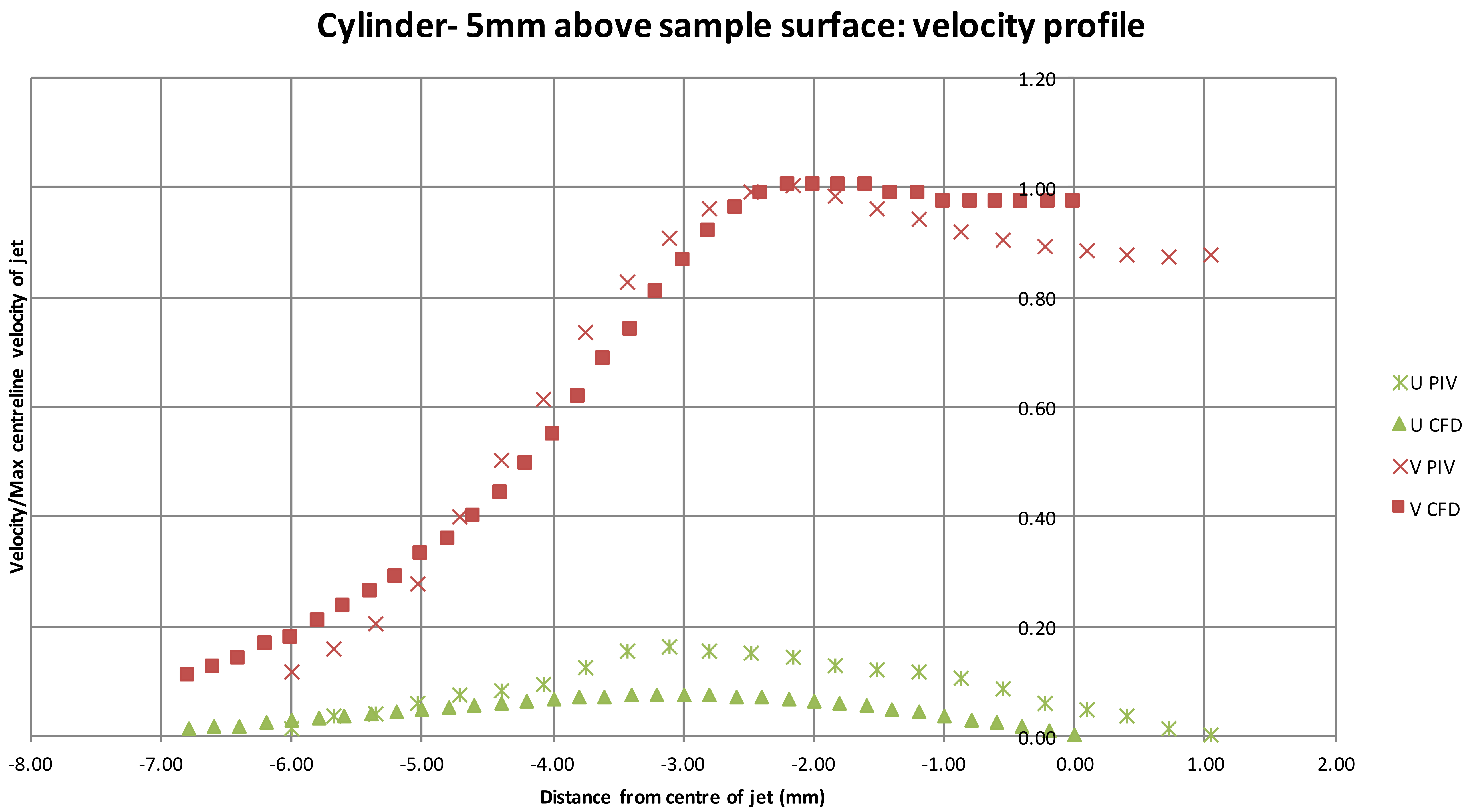


Cylinder

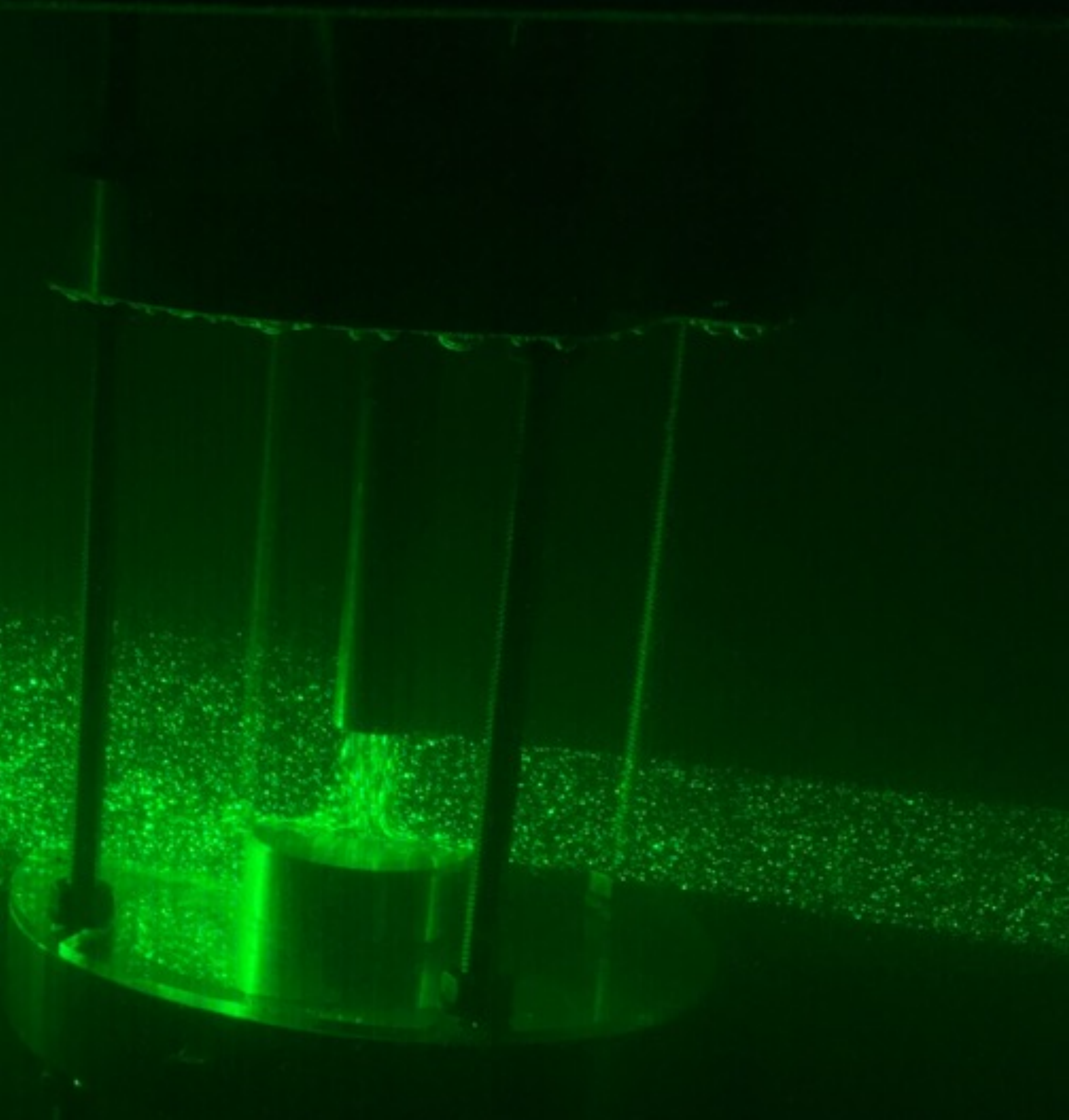


Cylinder- 5mm below nozzle exit: velocity profile
Velocity/Max centreline velocity of jet
Distance from centre of jet (mm)
1.20
1.00
0.80
0.60
0.40
0.20
0.00
-0.20
-8.00
-7.00
-6.00
-5.00
-4.00
-3.00
-2.00
-1.00
0.00
1.00
2.00
U PIV
U CFD
V PIV
V CFD

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

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

Thank you

alasdair.mackenzie.100@strath.ac.uk

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