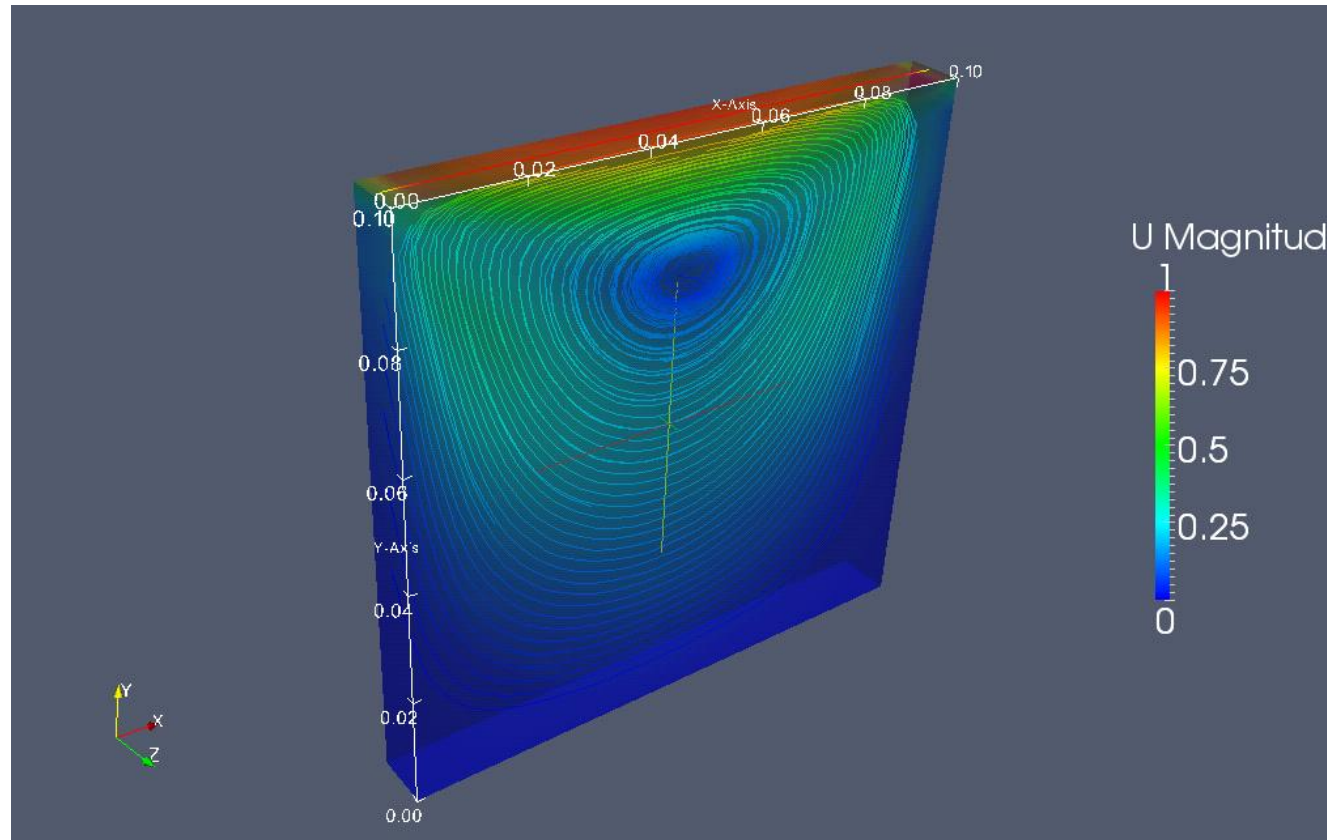# OpenFOAM Course

Assignment 1

Arash Eslamdoost

arash.eslamdoost@chalmers.se
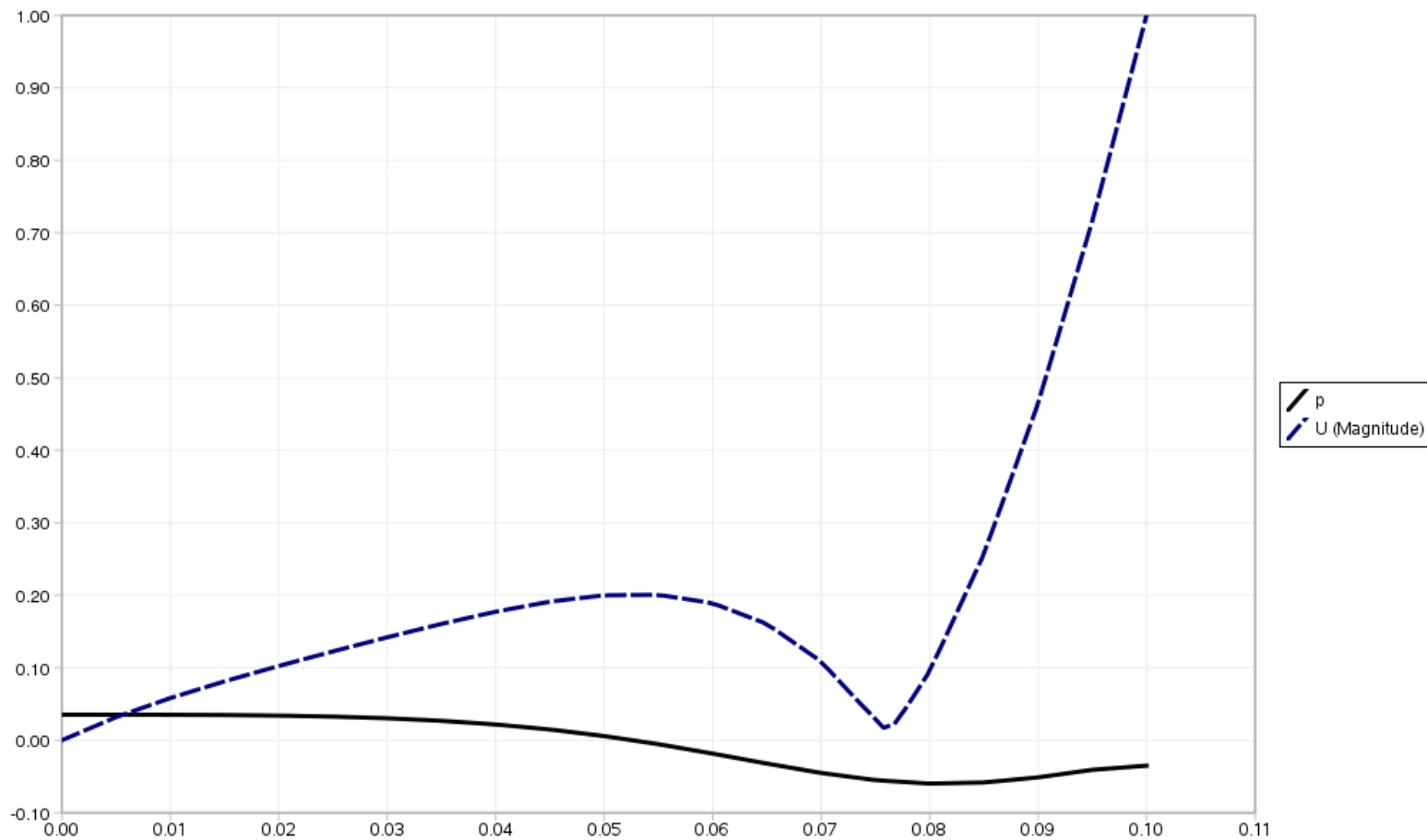
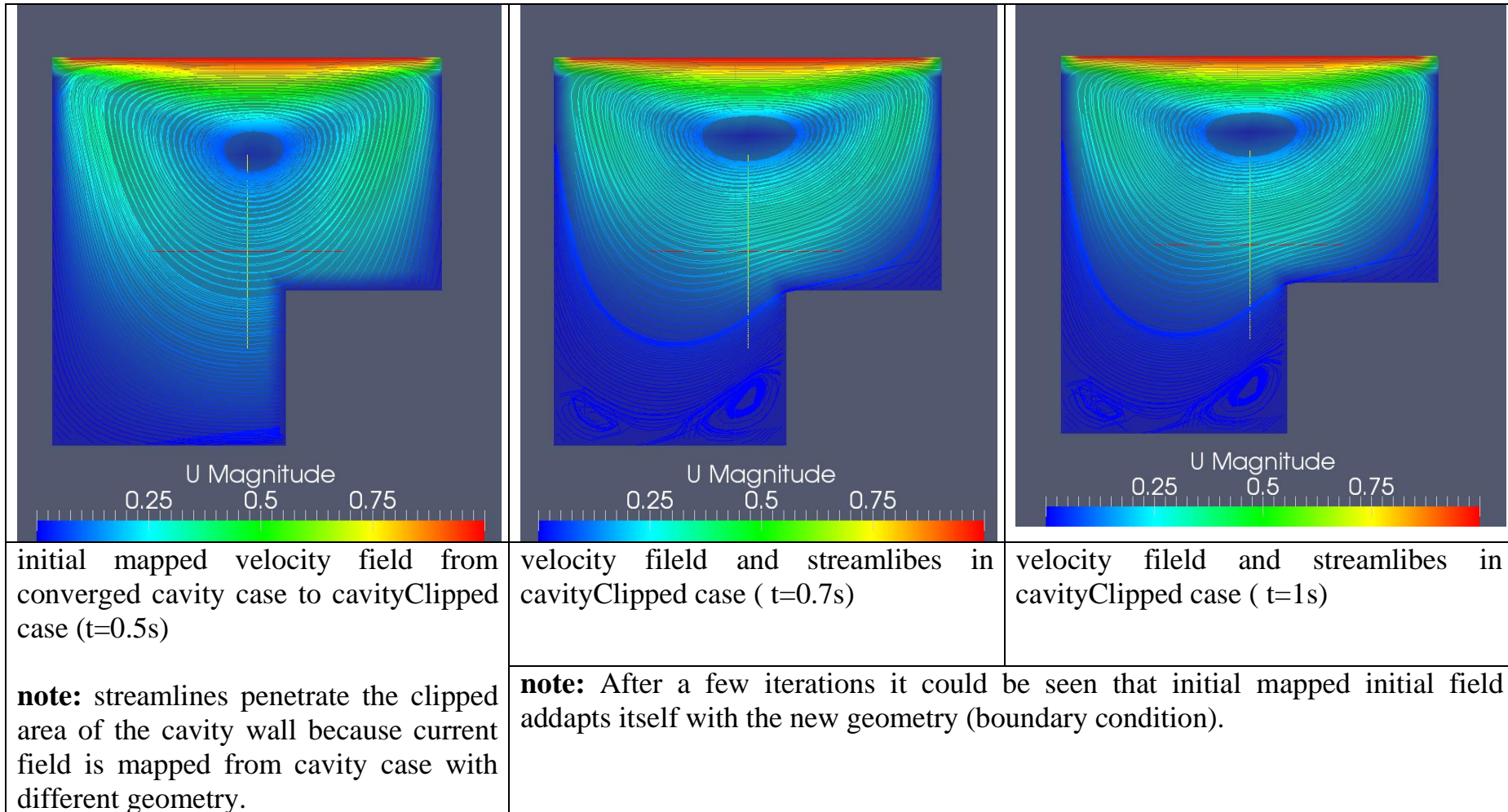**2009.10.05**

SOLVER: icoFoam
Case: cavity

Contour of velocity magnitude in a driven cavity and streamlines in it

Pressure [Pa] and Velocity [m/s] distribution exactly on the center-line of cavity from bottom fixed wall to the top moving wall
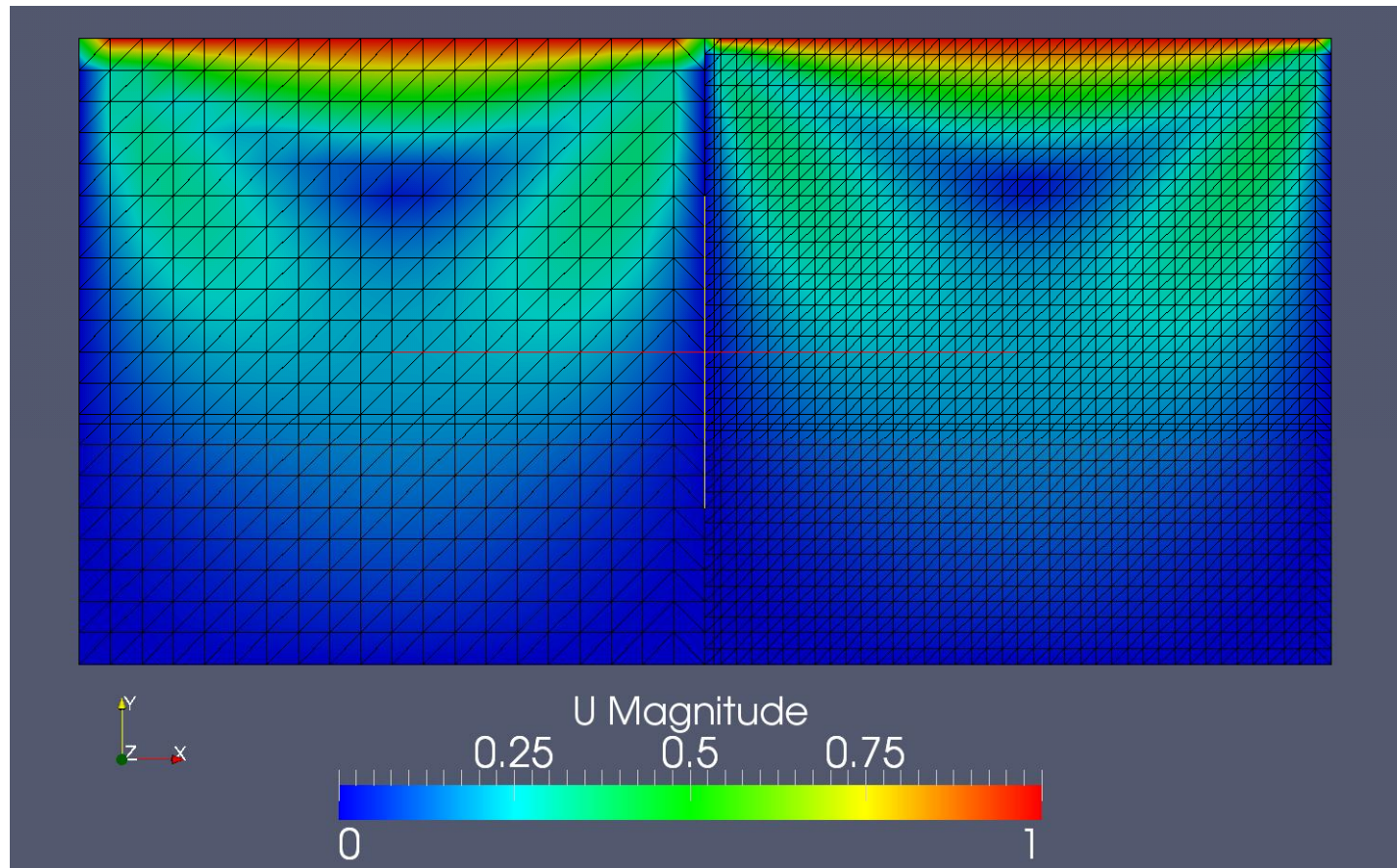
SOLVER: icoFoam
Case: cavityClipped

| initial mapped velocity field from converged cavity case to cavityClipped case (t=0.5s)	velocity fileld and streamlibes in cavityClipped case (t=0.7s)	velocity fileld and streamlibes in cavityClipped case (t=1s)
note: streamlines penetrate the clipped area of the cavity wall because current field is mapped from cavity case with different geometry.	**note:** After a few iterations it could be seen that initial mapped initial field addapts itself with the new geometry (boundary condition).	

SOLVER: icoFoam
Case: cavityFine

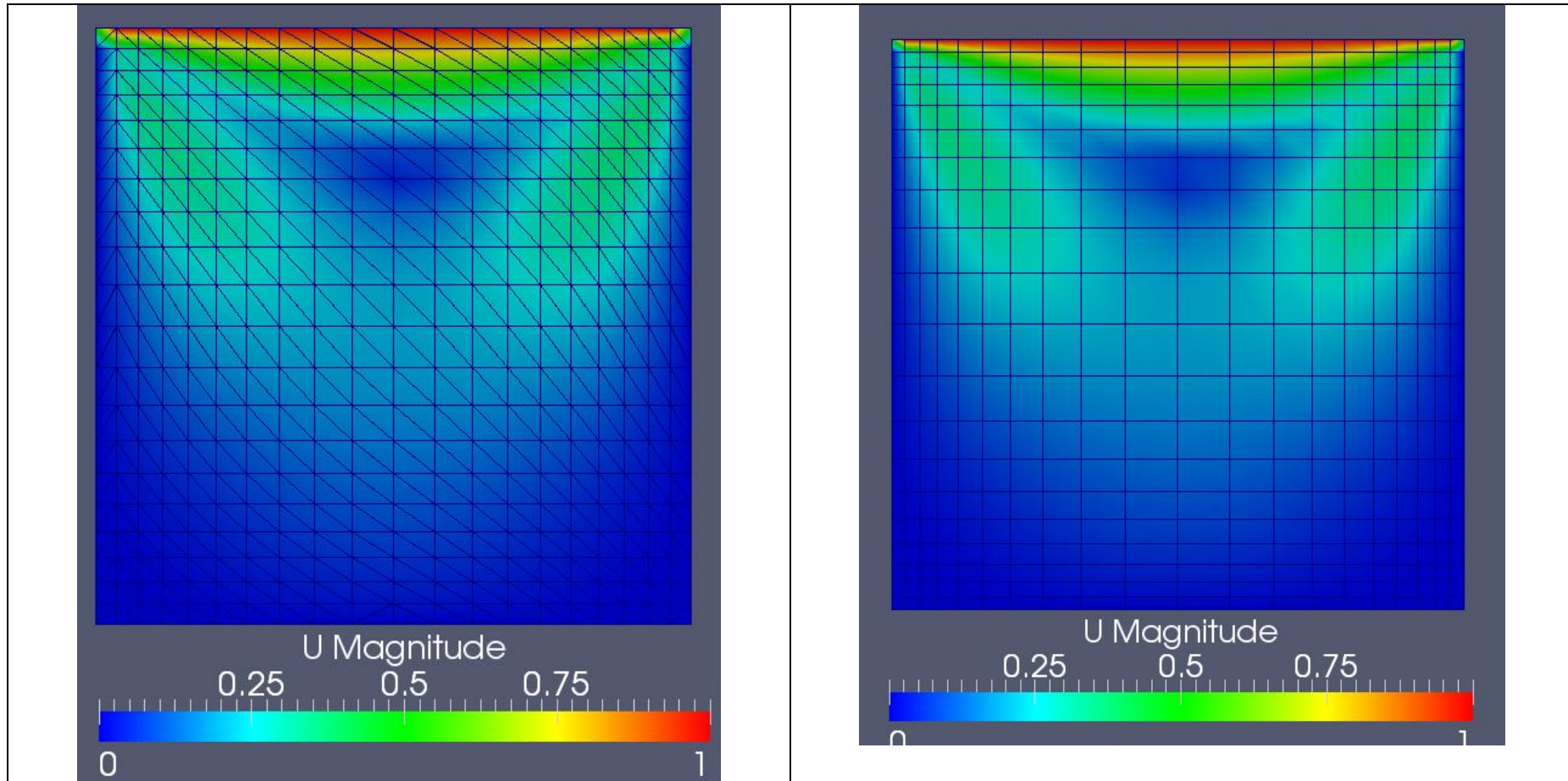Comparison of cavity case (left) with cavityFine (right) grid and converged velocity field
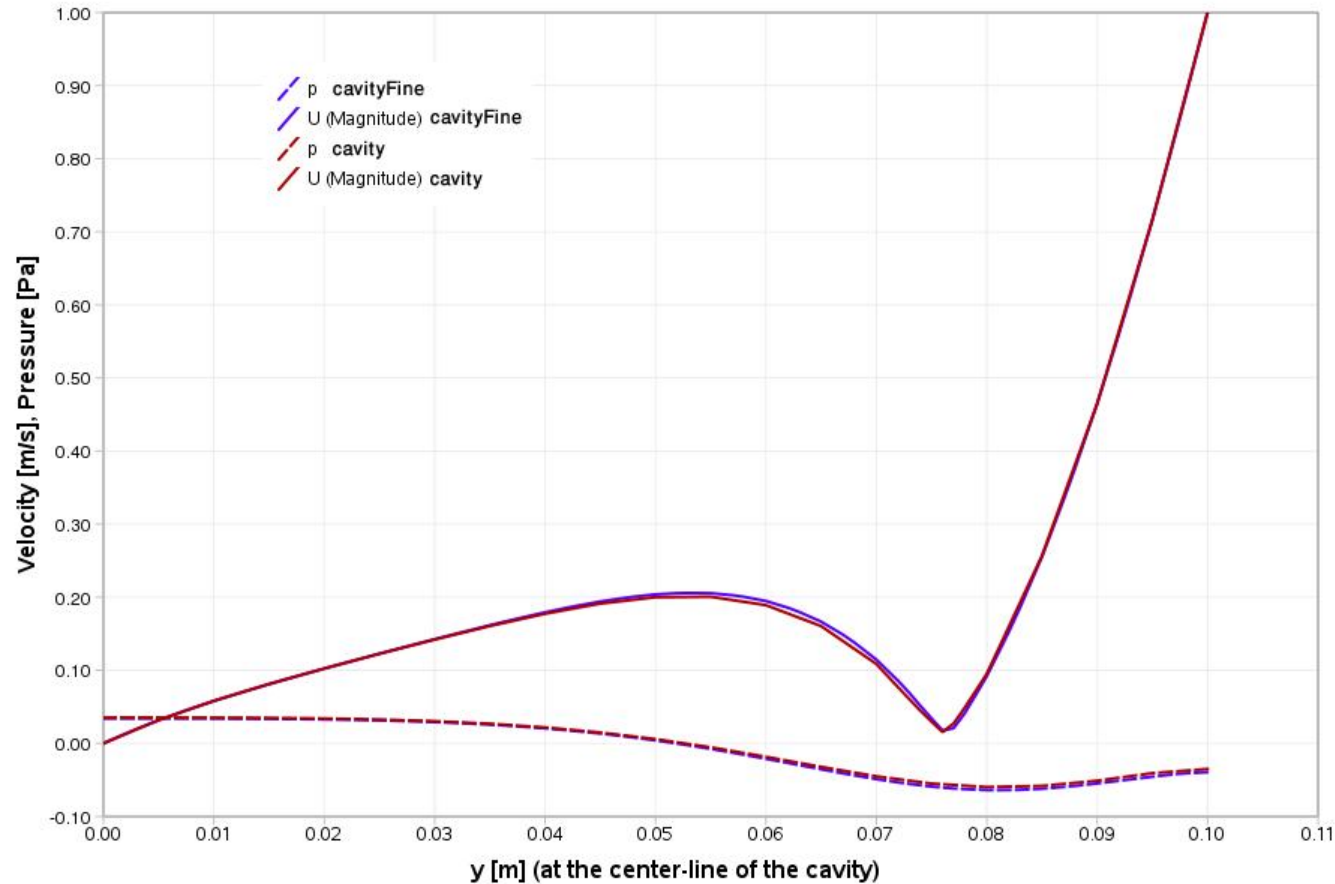
Comparison of Velocity and Pressure distribution along the cavity center-line for two similar cavities with different mesh resolution.

As it could be seen there is a slight difference between two grid resolution of cavity.
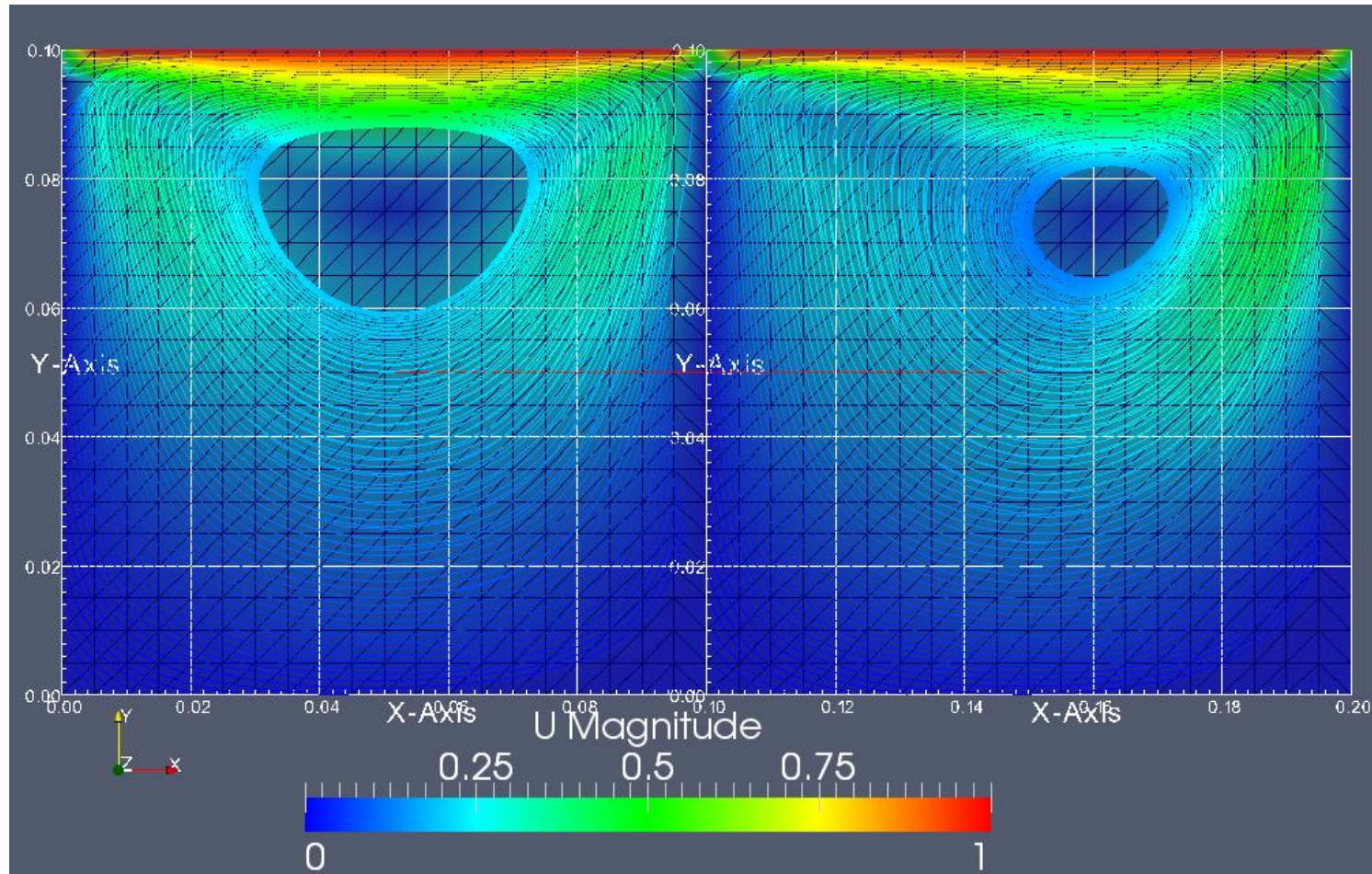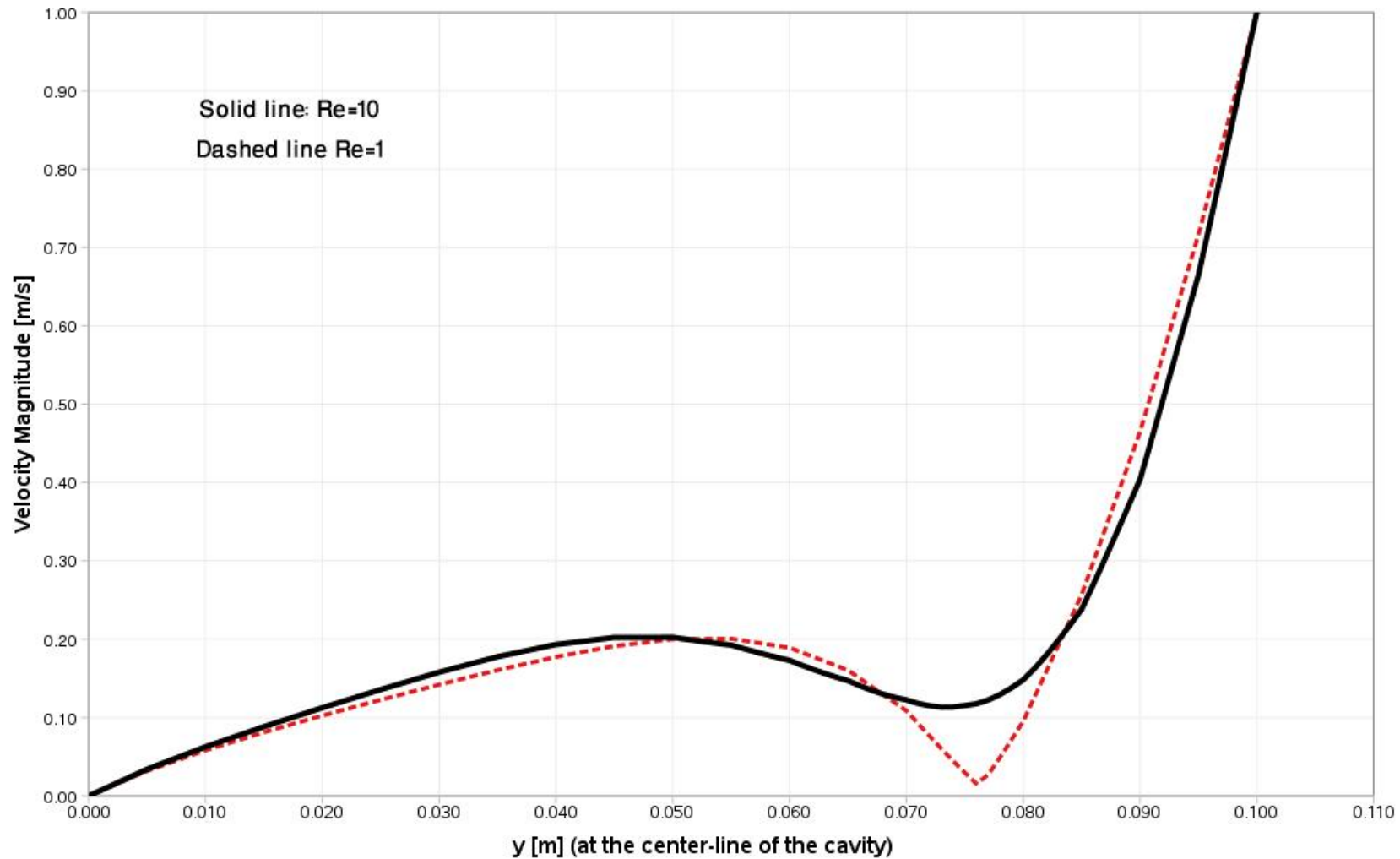
SOLVER: icoFoam
Case: cavityGrade

cavities with two different grading along x-axis (left : aspect ratio 2 and 0.5; right aspect ratio 4 and 0.25)
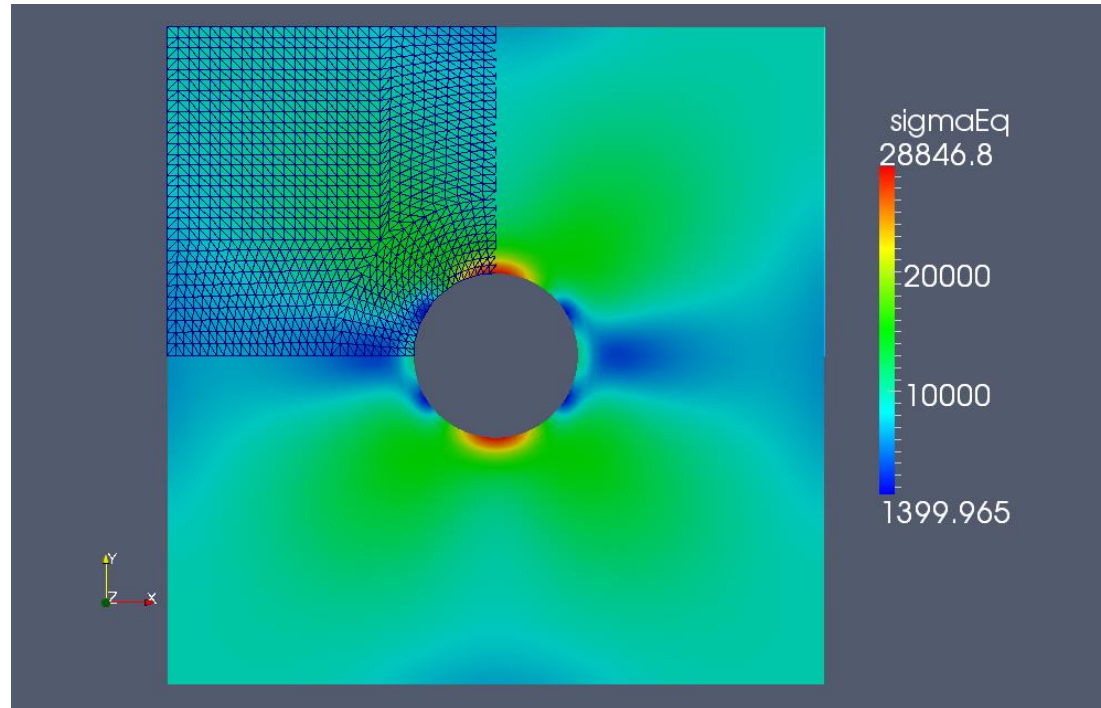
SOLVER: icoFoam
Case: cavityHighRe

cavity case with two different Re numbers. (left Re=1; right Re=10)

As it could be seen, increasing the Re number results in a different velocity field.

**SOLVER:** solidDisplacementFoam
Case: plateHole

Stress distribution on a plate with a hole at its center

Regarding the symmetry geometry and boundary conditions, only a quarter of the actual case is solved in this modeling (e.g. the quarter with grid shown or any other similar quarters); but, the whole geometry is plotted applying transform filter on the solved quarter.
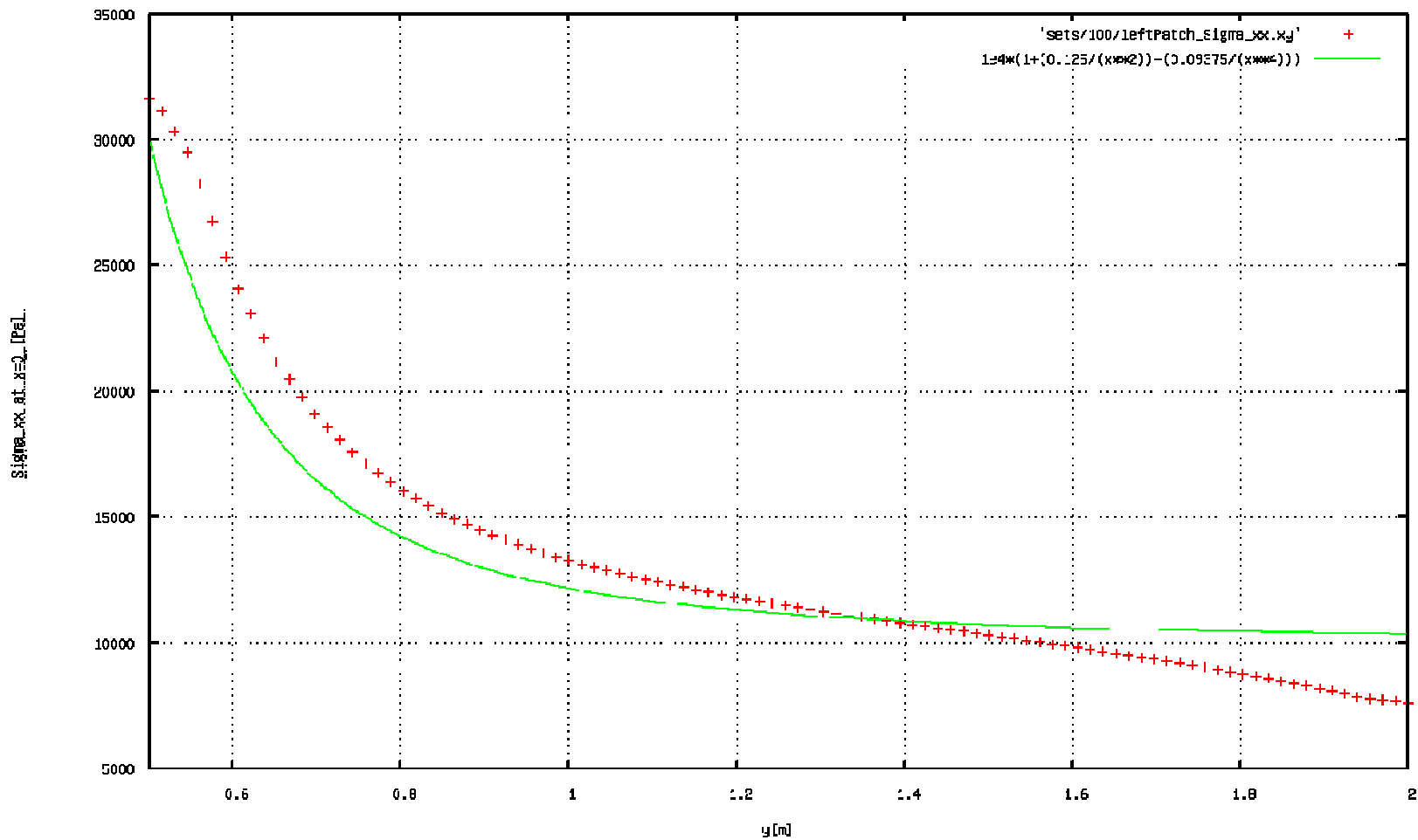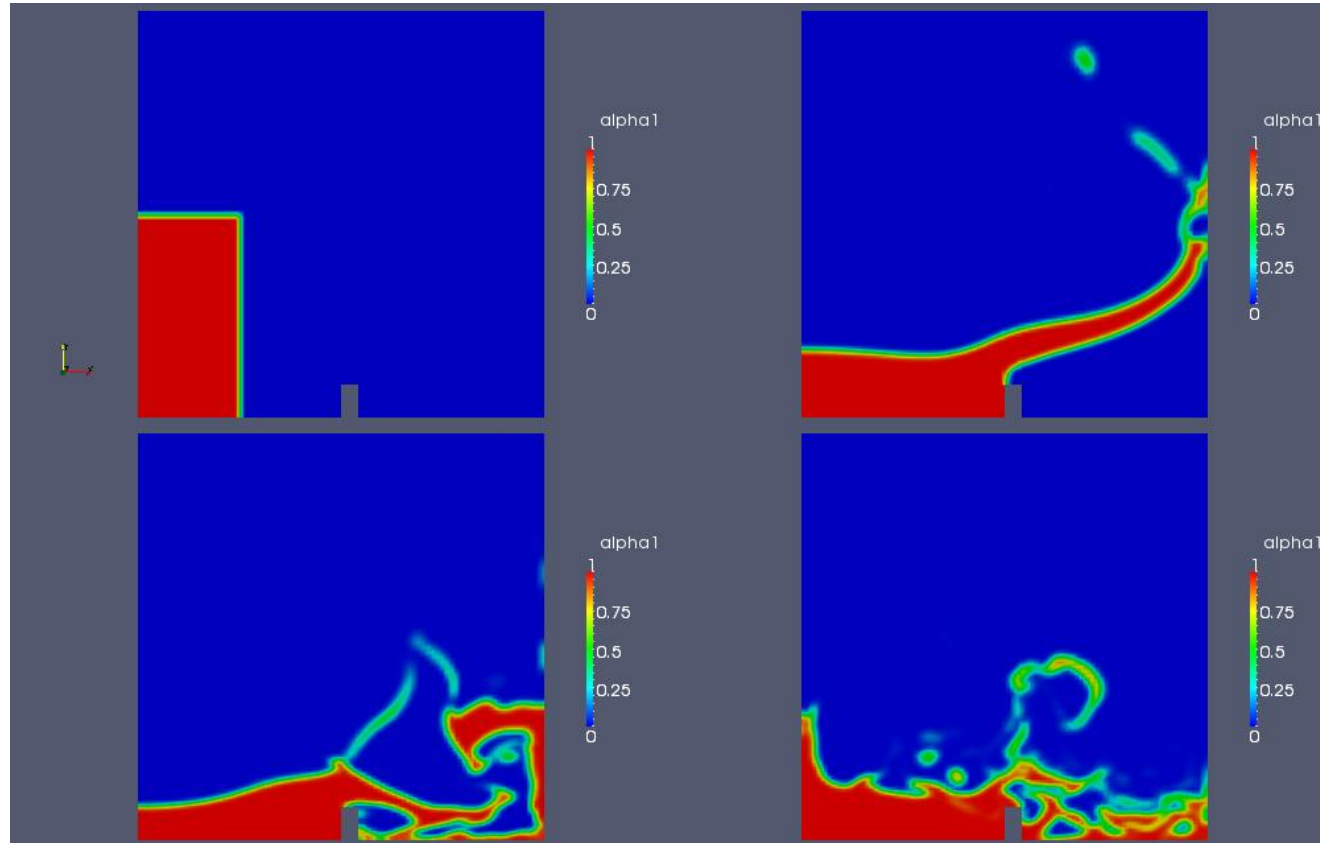
x-component of stress along center-line of the plate [dots for numerical solution and solid line for analytical solution]

SOLVER: interFoam
Case: damBreak
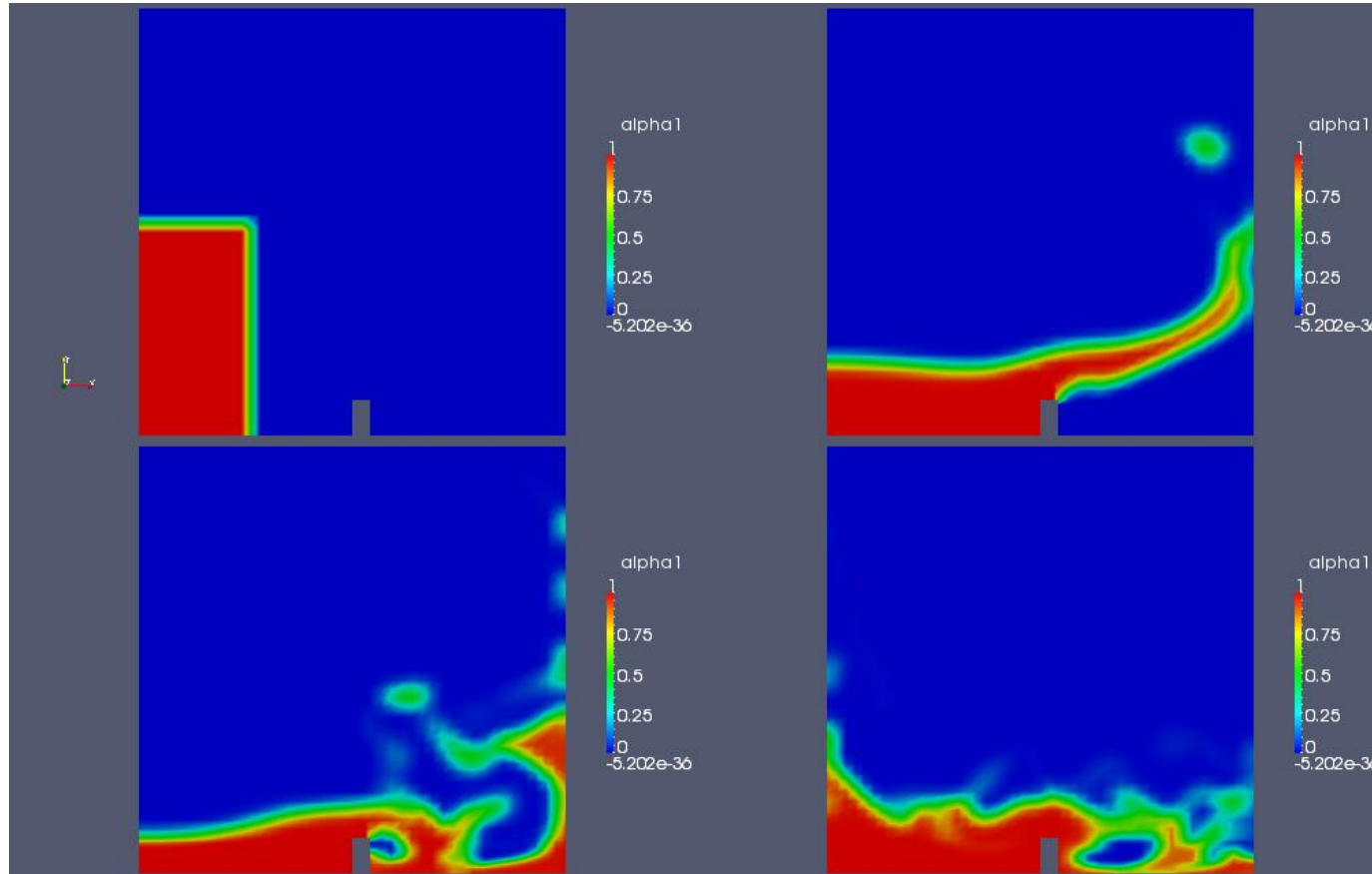
phase distribution during time after breaking the dam

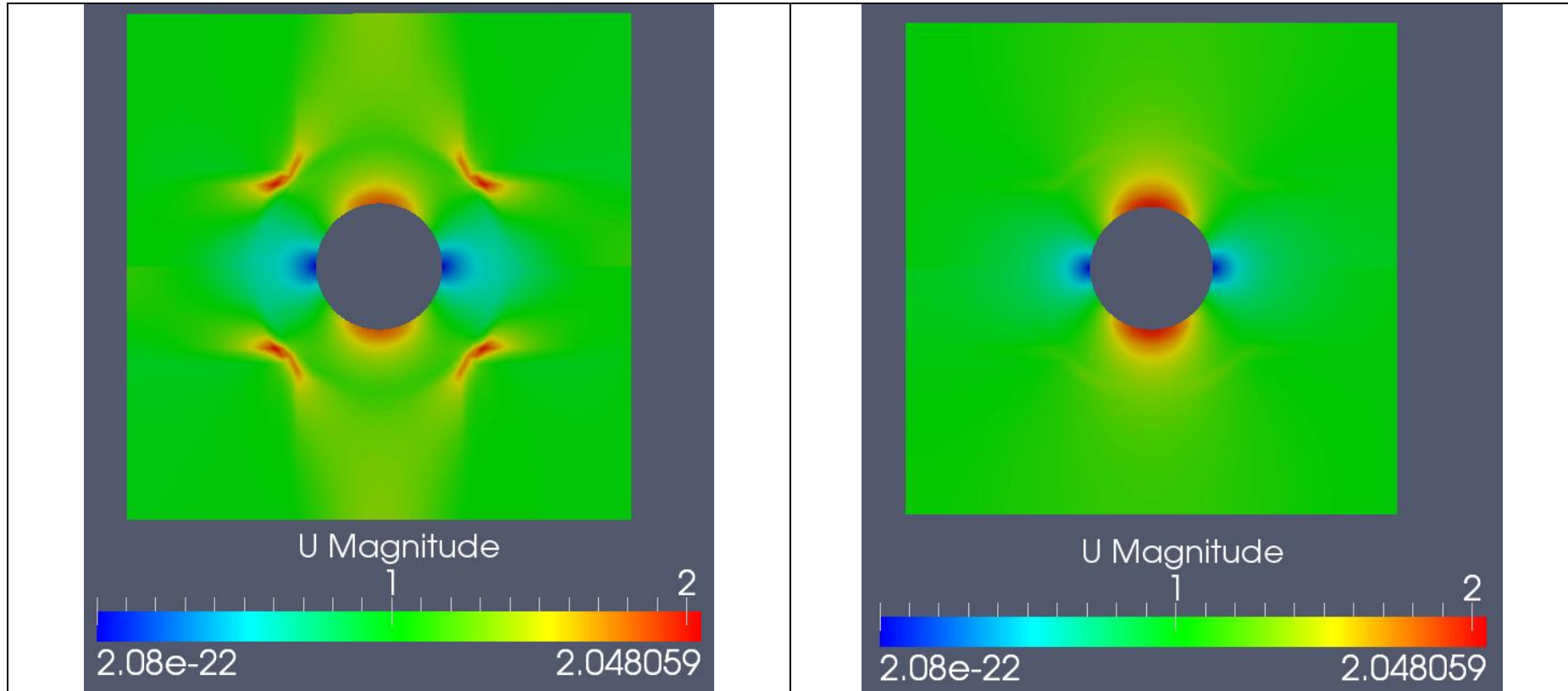**SOLVER:** interFoam
Case: damBreakFine

phase distribution during time after breaking the dam

Comparing to the previous case with coarse grid it could be seen that increasing the grid resolution makes it possible to capture a sharper interface.

SOLVER: potentialFoam
Case: cylinder

Plotting the case is done applying transform filter on the upper half of the cylinder case geometry and it has been rotated 180° around z-axis. Left picture is captured when there were not any non-orthogonality corrector function applied and the right picture in the final result when 3 steps of non-orthogonality correction are implemented.

SOLVER: simpleFoam
Case: pitzDaily
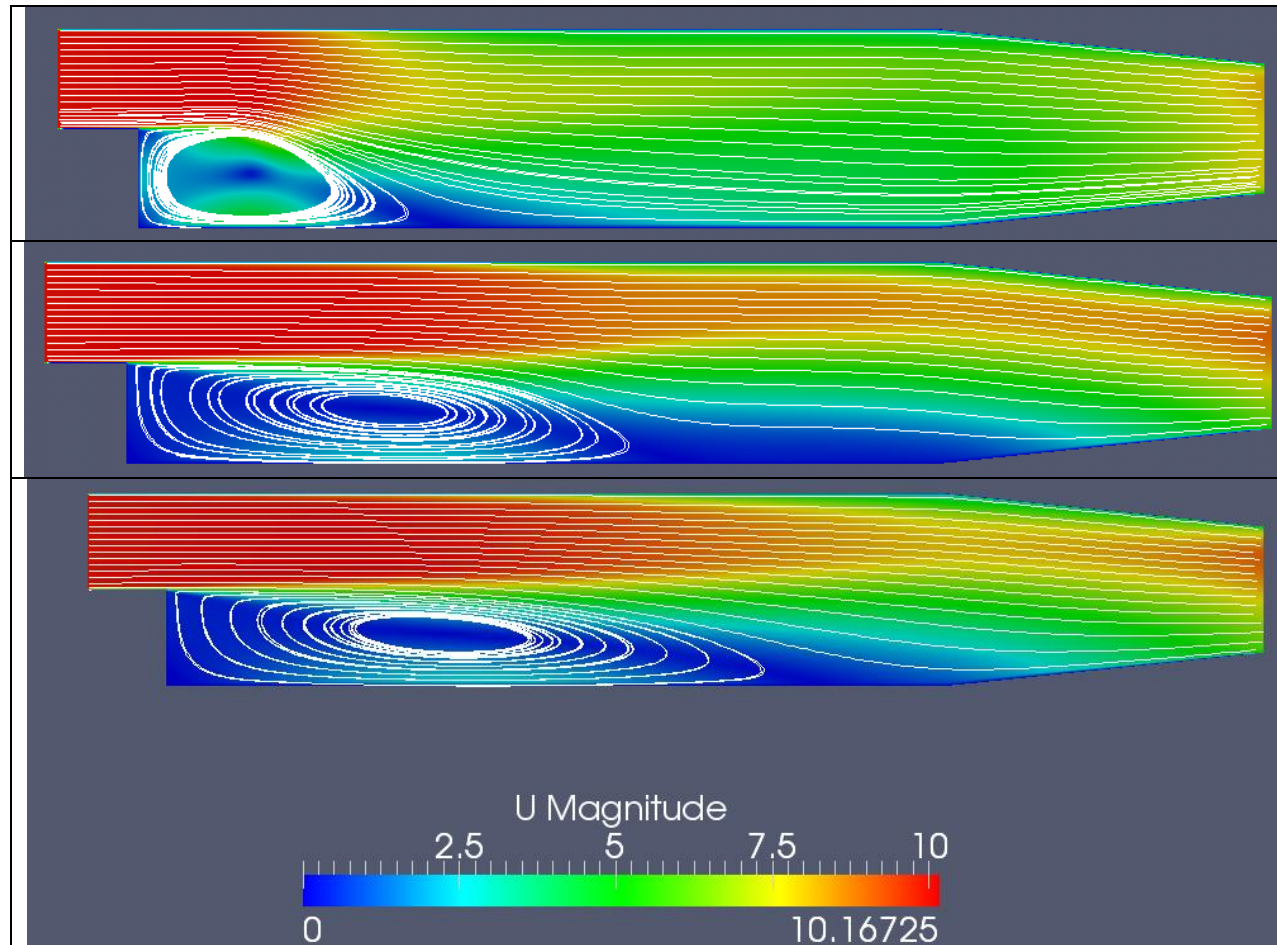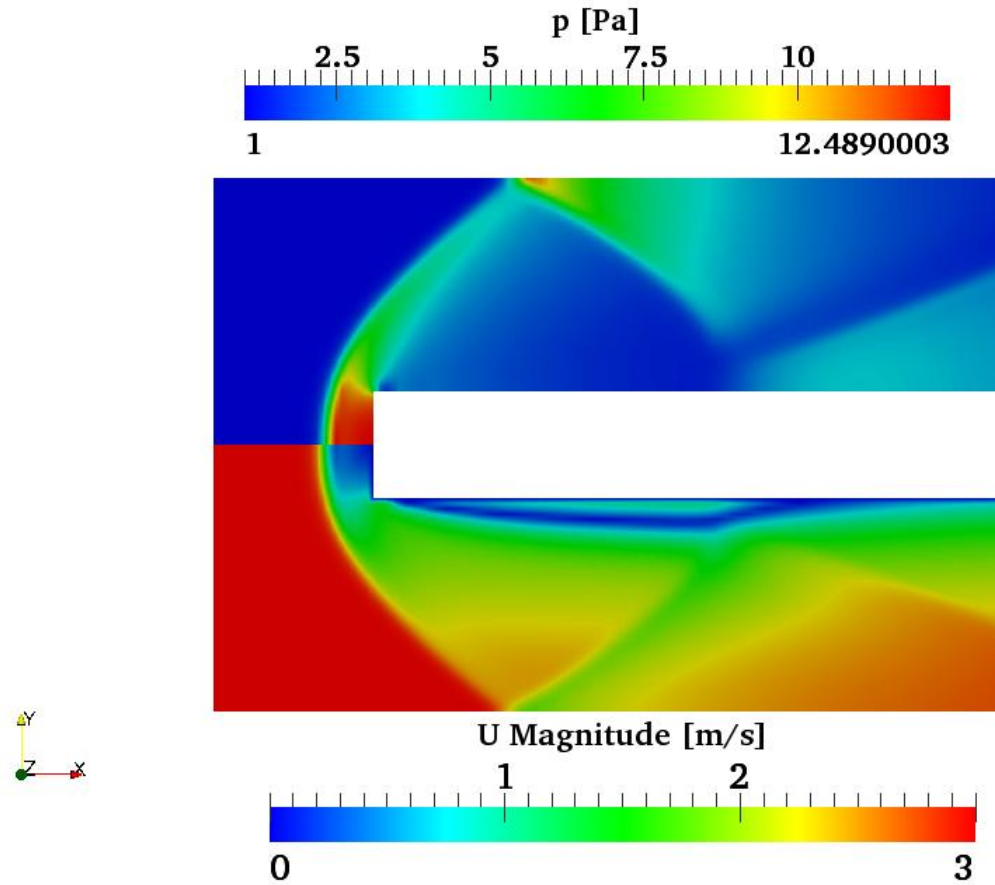

Velocity filed and streamlines during the convergence procedure
(From top to bottom iterations number are 100, 500 and 1000)
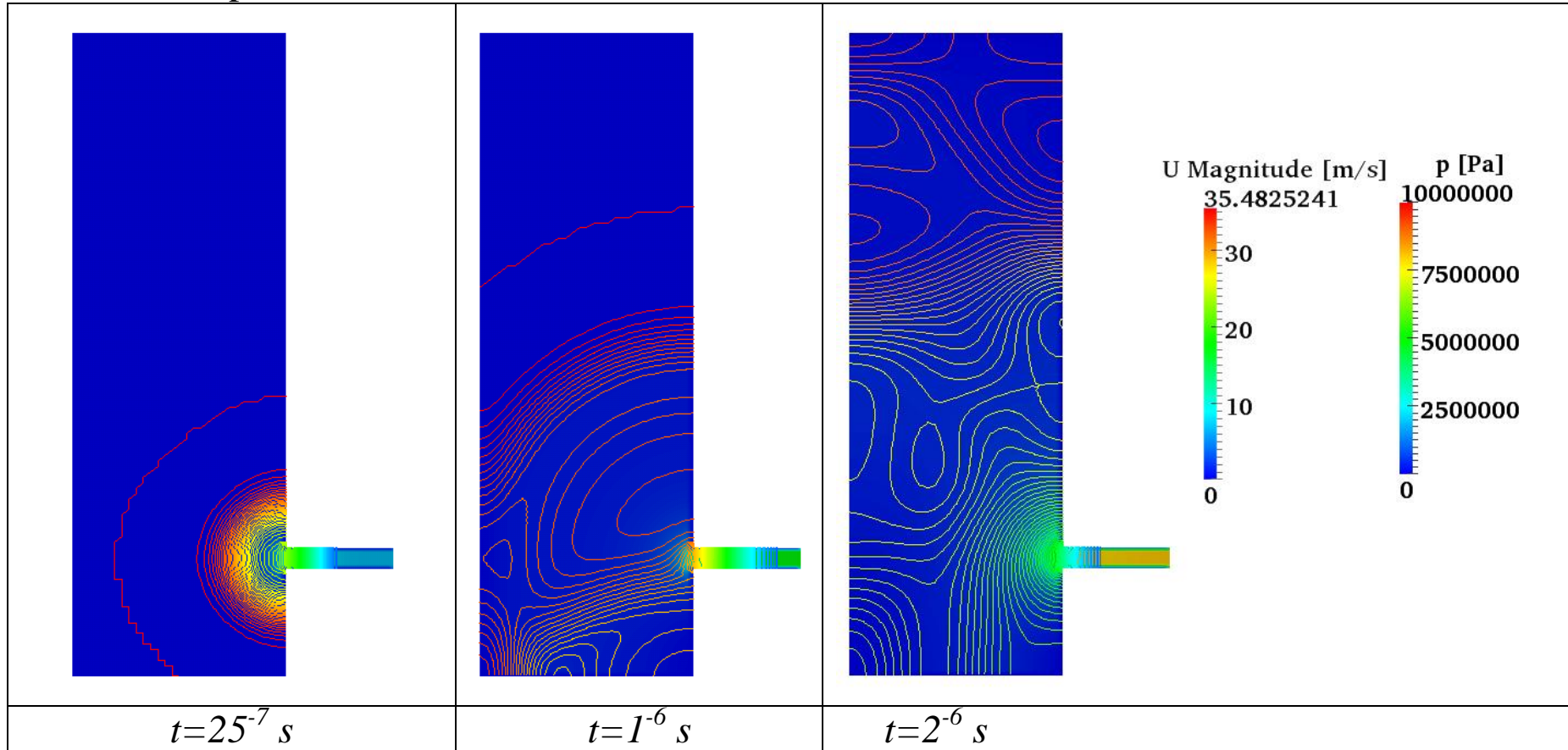
SOLVER: sonicFoam
Case: forwardStep

Contours of upper half display pressure field and the bellow half indicates the total velocity distribution. The regions where there are an abrupt gradient of pressure (velocity) show the location of shock occurrence.

SOLVER: sonicliquidFoam
Case: decompressionTank

| $t=25^{-7}$ s | $t=1^{-6}$ s | $t=2^{-6}$ s |

Pressure propagation in a water tank is plotted during various times by some iso-pressure lines. Velocity magnitude is presented as contour in the tank. It could be seen that actually there is not any detectable variation in velocity amount during the time in most regions but on the other hand, variation of pressure is more sensible.

SOLVER: sonicliquidFoam
Case: decompressionTankFine

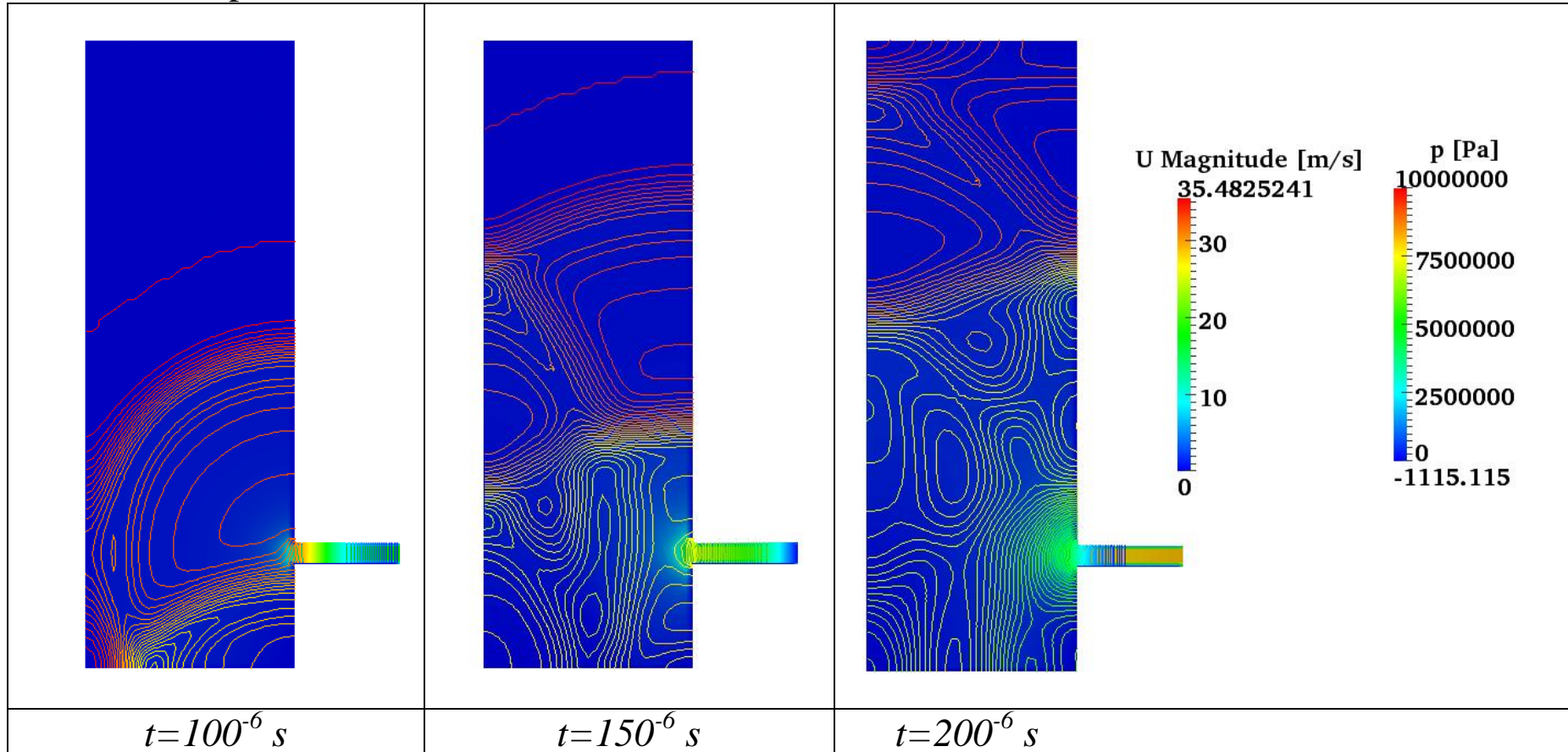| $t=100^{-6}$ s | $t=150^{-6}$ s | $t=200^{-6}$ s |

Pressure propagation in a water tank is plotted during various times by some iso-pressure lines. Velocity magnitude is presented as contour in the tank. It could be seen that actually there is not any detectable variation in velocity amount during the time in most regions but on the other hand, variation of pressure is more sensible. Comparing to the previo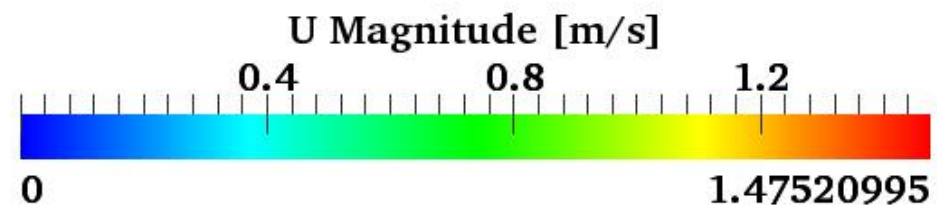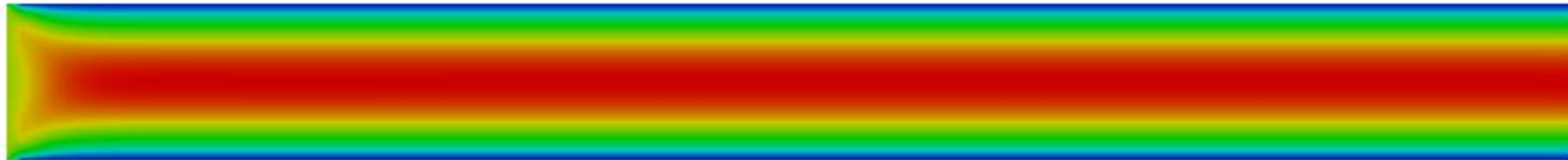us case with coarse grid, could be seen that pressure propagation and pressure waves in the flow field are captured more in detail.

SOLVER: mhdFoam
Case: hartmann

Comparing the velocity filed in presence (lower case) and absence (upper case) of a magnetic field.
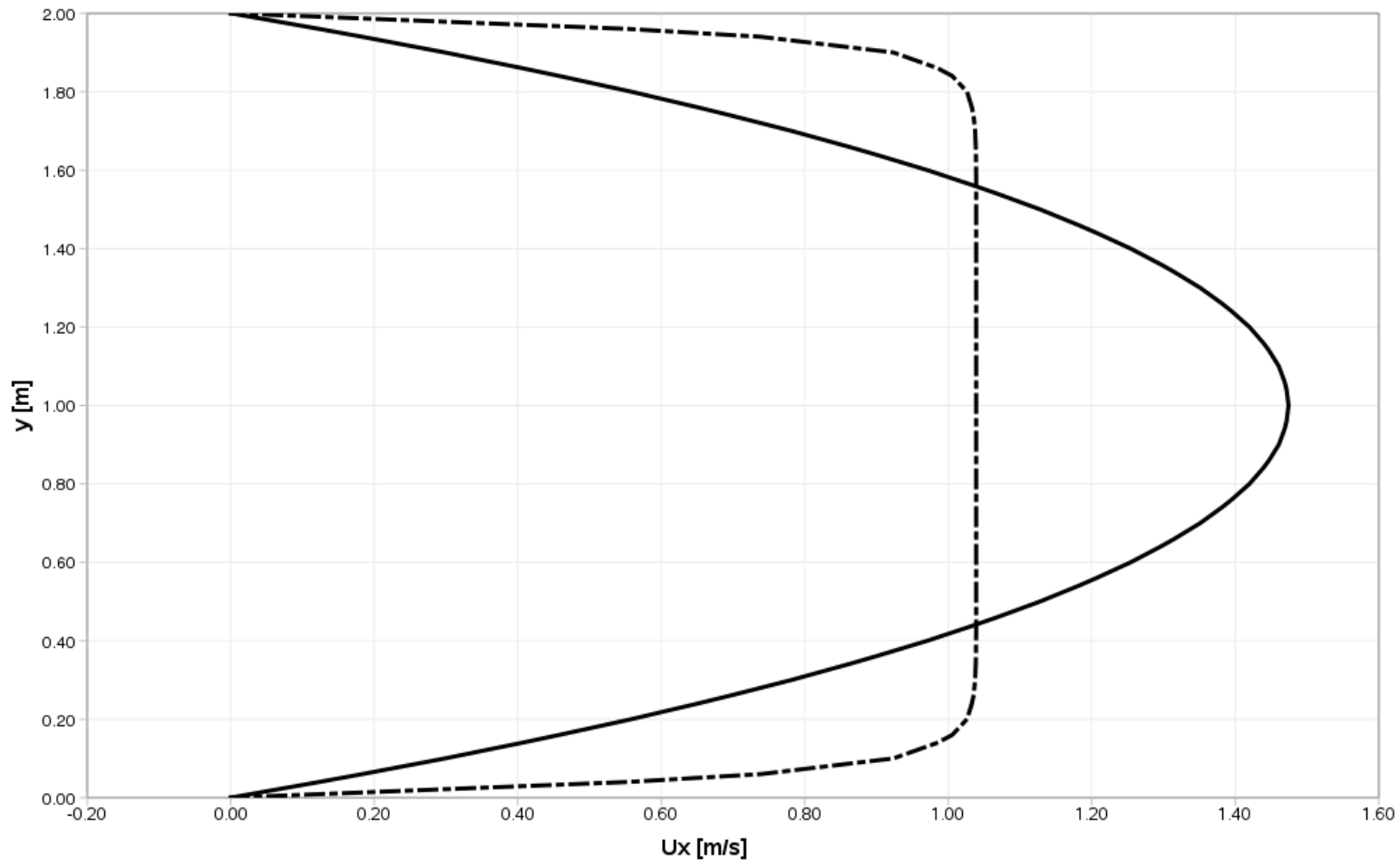
Comparing the velocity distribution along a line located at the center of the channel (x=10 m) from the bottom wall to the top wall in absence (solid line) and in presence (dashed line) of a magnetic field.

SOLVER: interFoam
Case: damBreakModified

This case is just an imaginary case to test the utilities of OpenFOAM. The case is based on damBreak original case. Just it has been tried to check the setFields code possibility for using variety of alfas as initial field. For this purpose initially a box is defined with alfa value of 1(left figure). Subsequently, to change the value of alfa inside the initial box another box is defined with alfa value of 0.5 (right figure). So, as it could be seen in the right figure it is possible to set different values initially to the field.
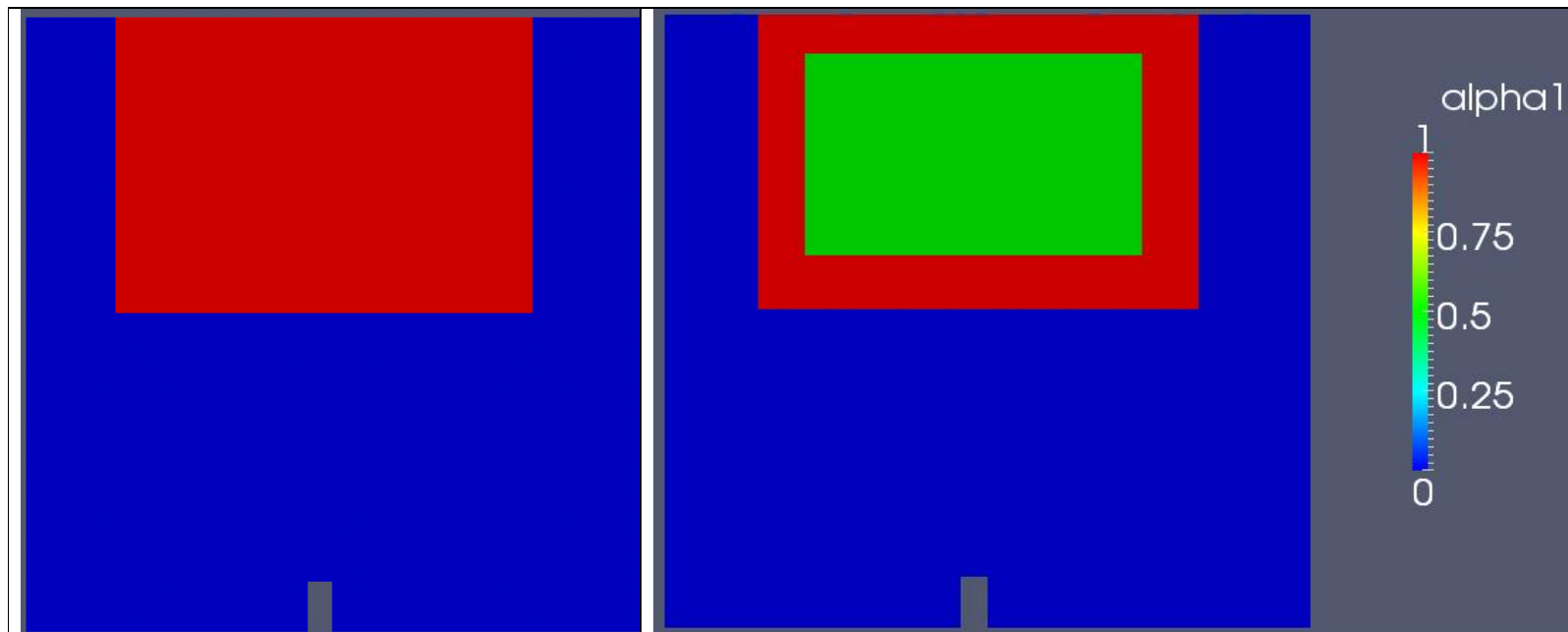
To do the discussed setFilds the following lines are inserted to the setFieldsDict:

```
defaultFieldValues
(
    volScalarFieldValue alpha1 0
);

regions
(
```

|	

<pre>boxToCell
    {
        box (0.1 0.3 -1) (0.5 0.6 1);
        fieldValues
        (
            volScalarFieldValue alpha1 1
        );
    }</pre>
	First block with alfa value of 1

<pre>   boxToCell
    {
        box (0.2 0.4 -1) (0.4 0.5 1);
        fieldValues
        (
            volScalarFieldValue alpha1 0
        );
    }</pre>
	Second block inside the first block with alfa value of 0.5

```
);
```

Moreover, to make the case more like the driven cavity case the upper-wall of this case has also changed to a moving wall with Ux=5 m/s.

```
dimensions      [0 1 -1 0 0 0 0];

internalField   uniform (0 0 0);

boundaryField
{
    leftWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    rightWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    lowerWall
    {
        type            fixedValue;
        value           uniform (0 0 0);
    }
    movingWall
    {

        type            fixedValue;
        value           uniform (5 0 0);
    }
    defaultFaces
    {
        type            empty;
    }
}
```

Results of the run are plotted in the following graphs. Contours show alfa value and vectors indicate velocity field.

t=0 s	t=0.05 s	t=0.5 s
t=2 s	t=3 s	t=4 s
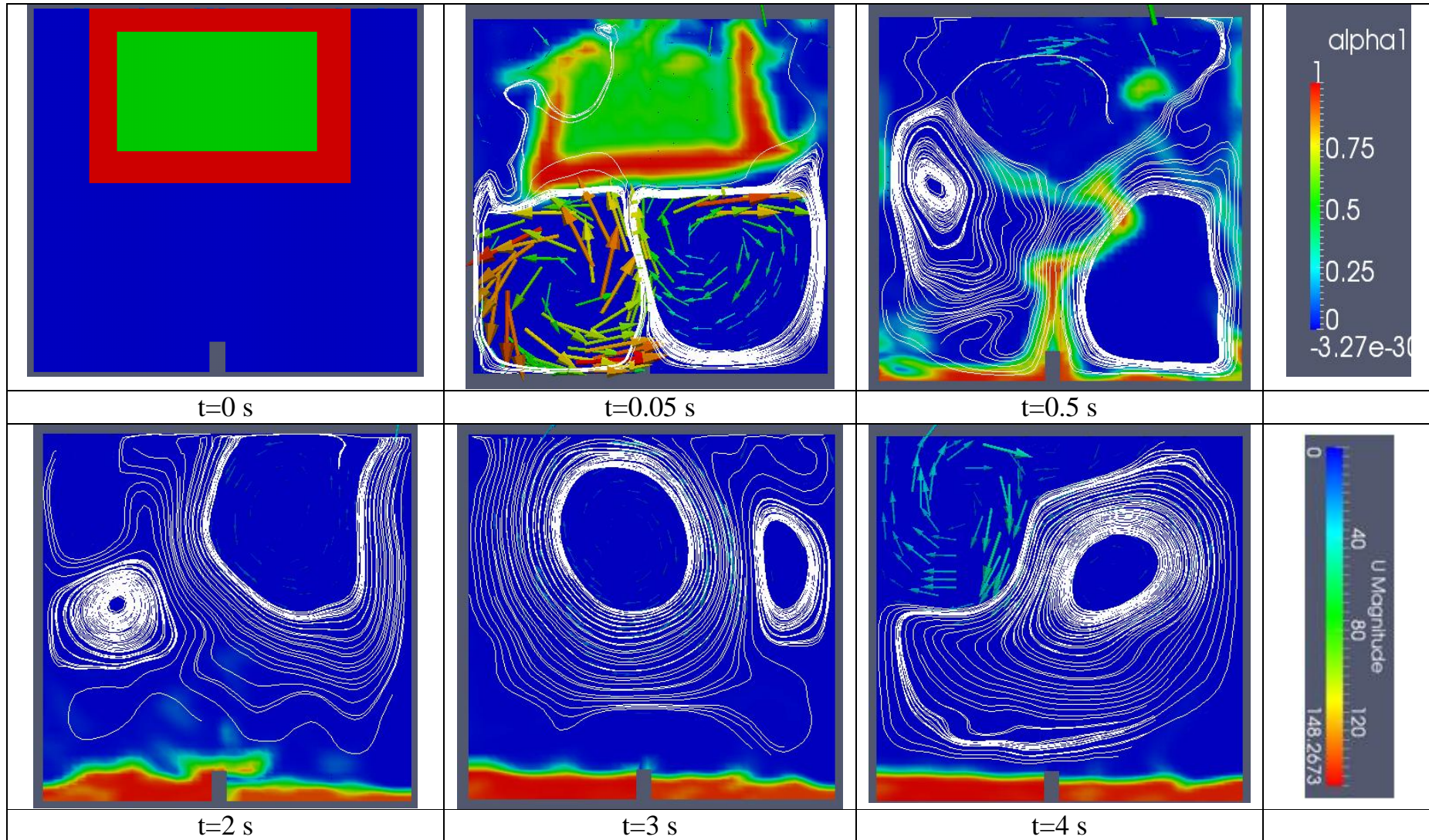
At t=0.05s just after the block of water find itself floating in the air and starts to collapse and falls down two main vortices are formed in the cavity just below the floating water block. The largest value of velocity (~ 148 m/s) also occurs in this time step and suddenly after this time step reduces. After t=2s almost the effects of collapsed water block disappears and the flow field is mainly affected by the upper moving wall and a couple of vortices initiate and start circling in the field.