OpenFOAM Course

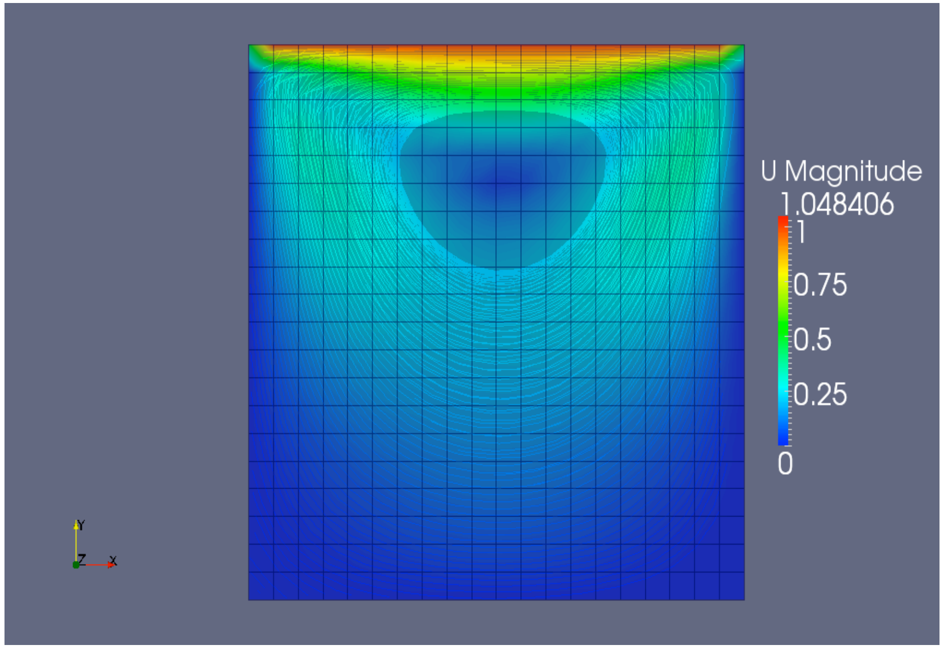
Assignment 1

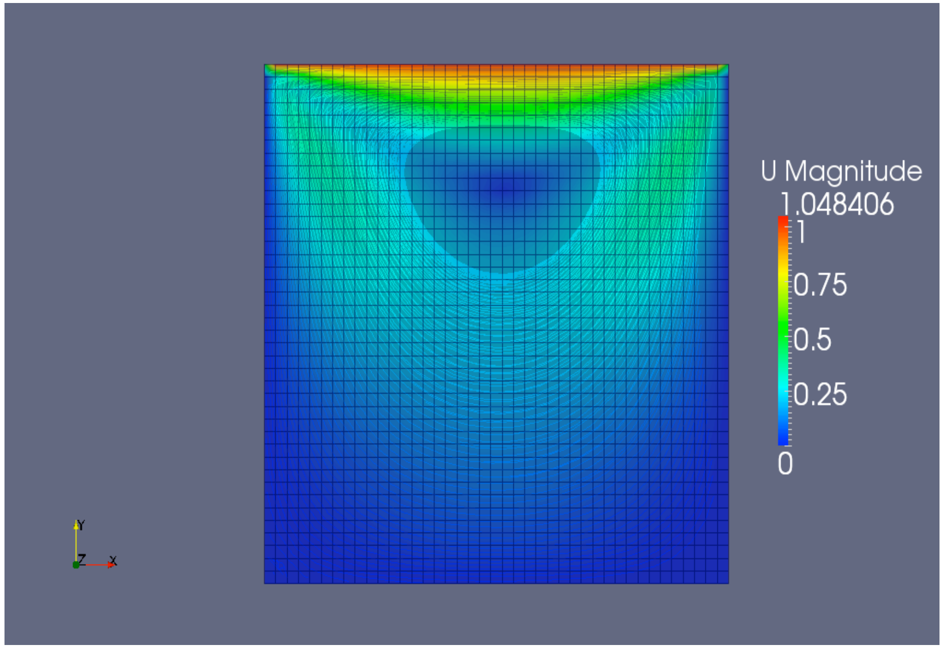
Ehsan Yasari
yasari@chalmers.se
Combustion Division of Applied Mechanics Department

2010-09-07

Solver : **IcoFoam**: Transient solver for incompressible, laminar flow of Newtonian fluids

Case: Cavity, CavityFine

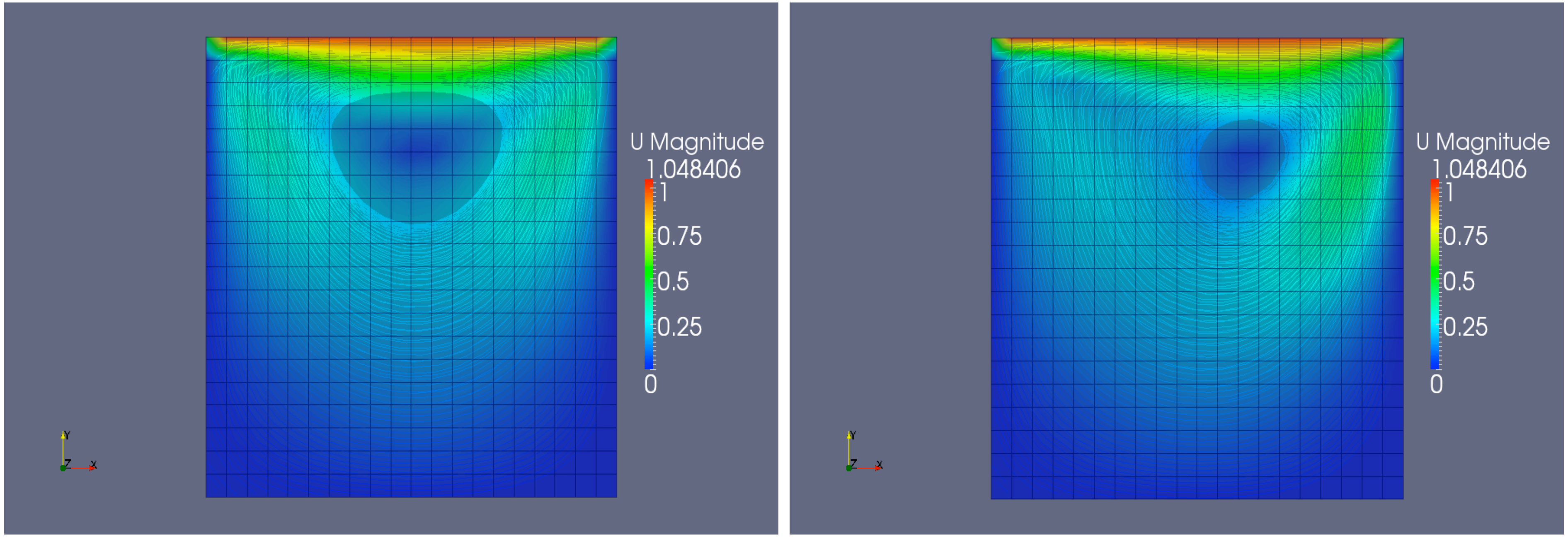




Contour of the velocity and the streamline for the coarse mesh(left) and the fine mesh(right)

Solver : **IcoFoam**: Transient solver for incompressible, laminar flow of Newtonian fluids

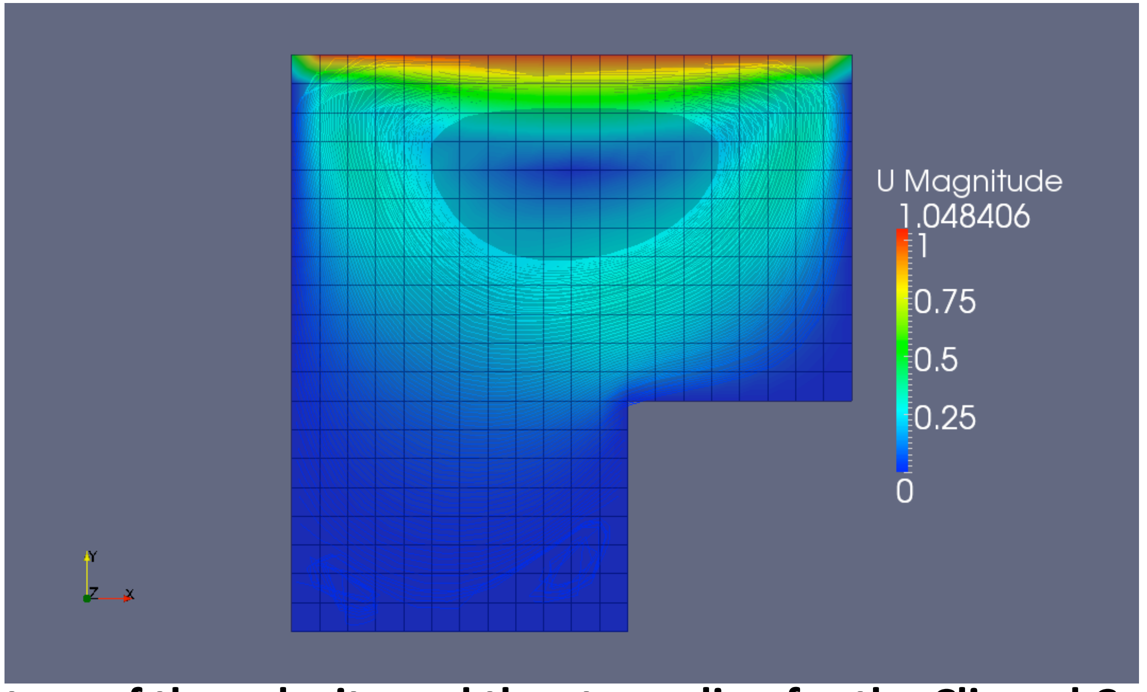
Case: Cavity, CavityHighRe



Contour of the velocity and the streamline for different Reynolds numbers
(left: Re=1 right: Re=10)

Solver : **IcoFoam**: Transient solver for incompressible, laminar flow of Newtonian fluids

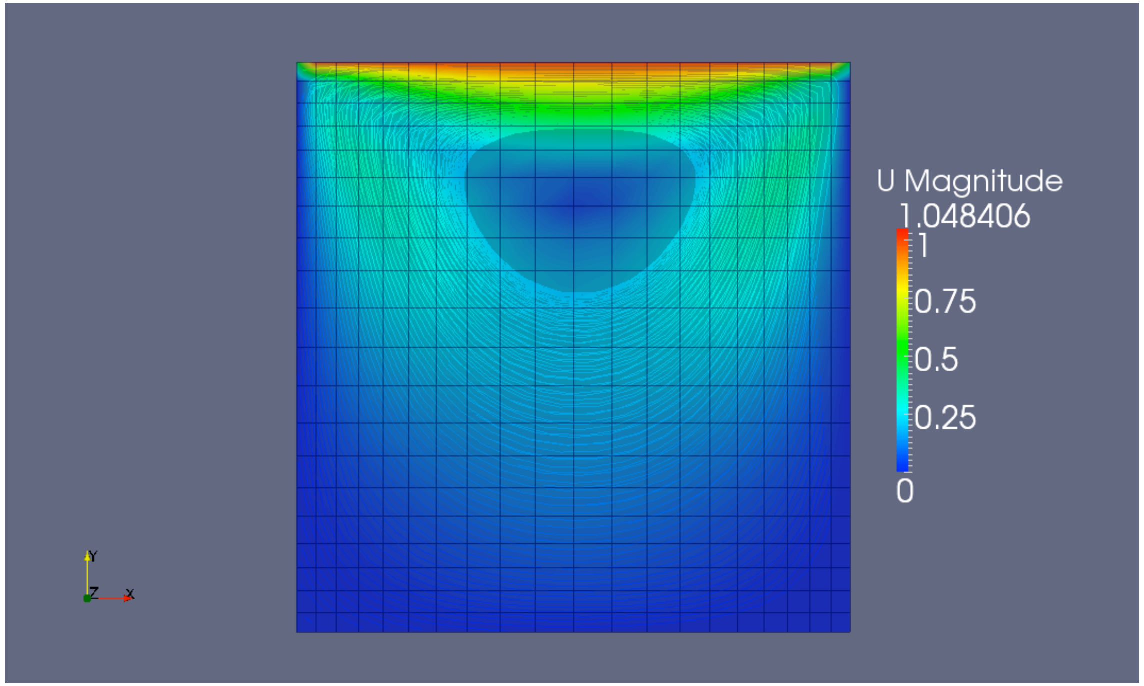
Case: Cavity Clipped



Contour of the velocity and the streamline for the Clipped Cavity

Solver : **IcoFoam**: Transient solver for incompressible, laminar flow of Newtonian fluids

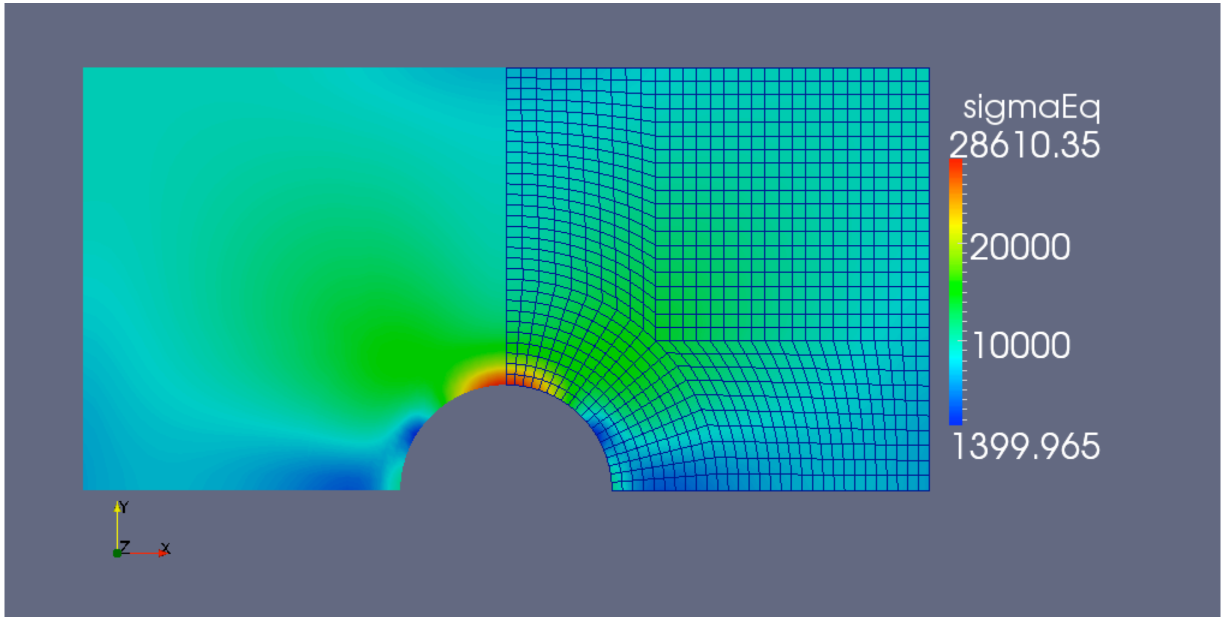
Case: CavityGrade



Contour of the velocity and the streamline for the CavityGrade

Solver : **solidDisplacementFoam**: Transient segregated finite-volume solver of linear-elastic, small-strain deformation of a solid body, with optional thermal diffusion and thermal stresses

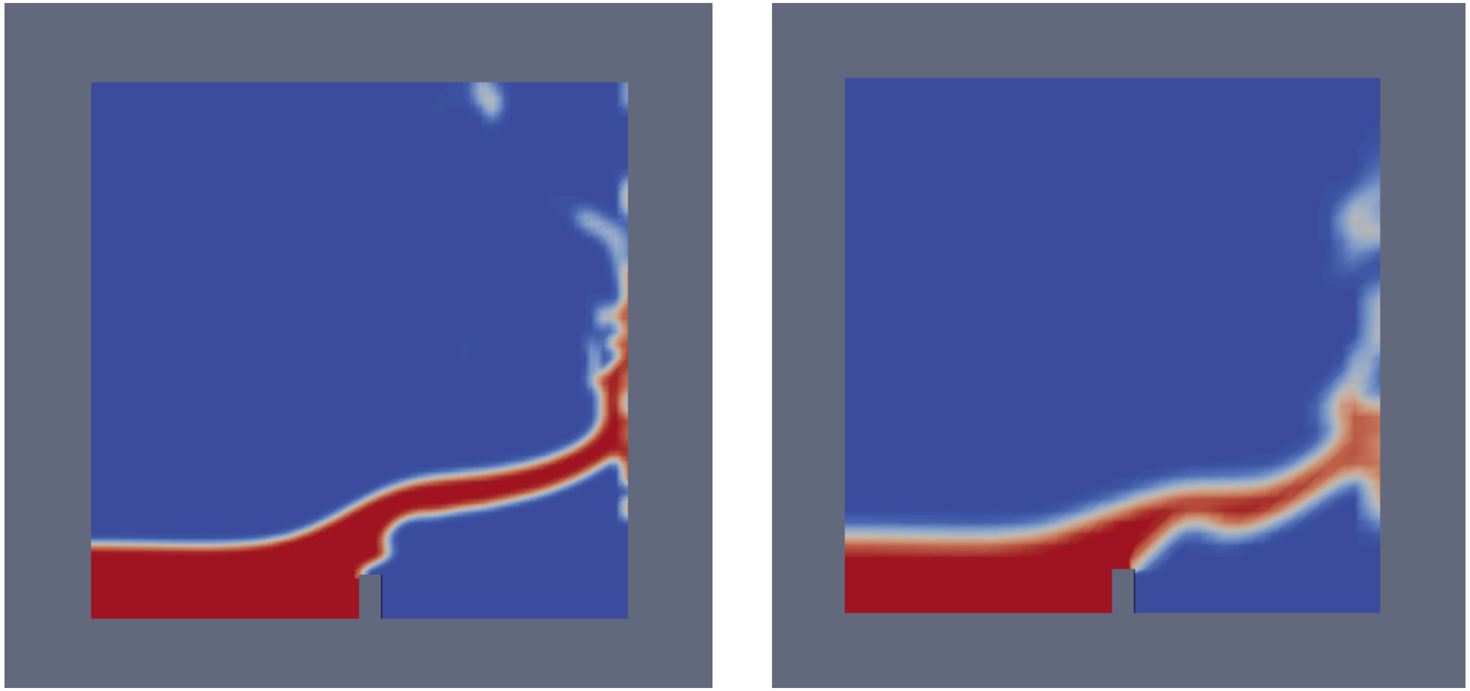
Case: plateHole



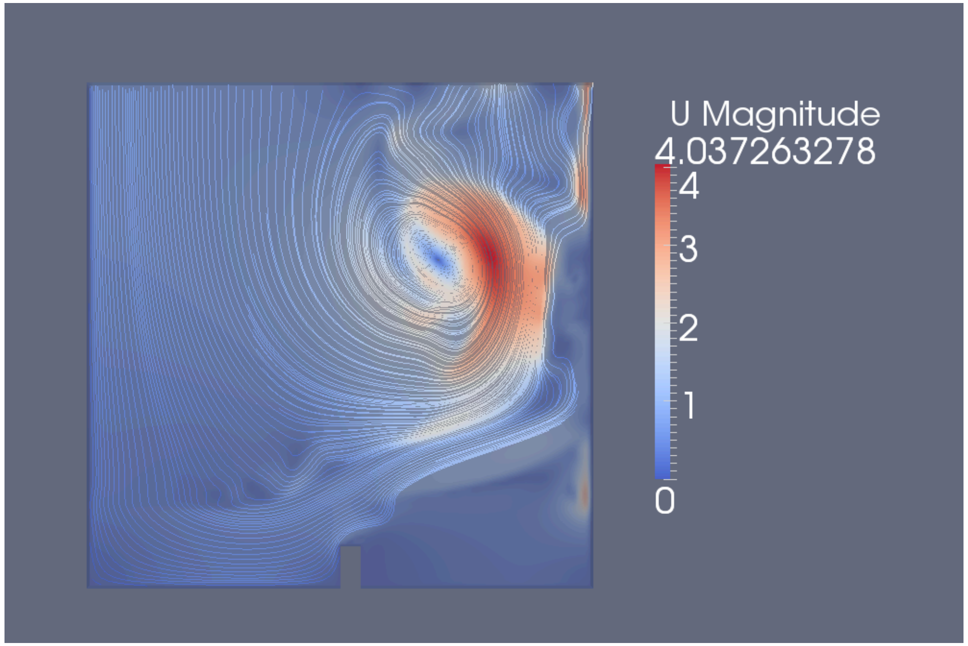
Stress distribution on a flat plate with a circular hole at its center

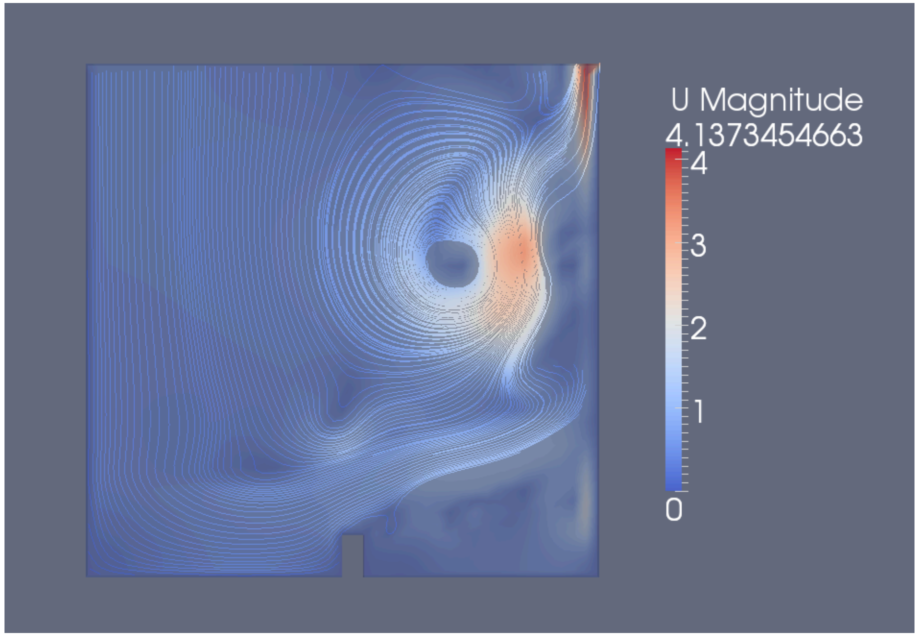
Solver : **interFoam**: Solver for 2 incompressible, isothermal immiscible fluids using a VOF (volume of fluid) phase-fraction based interface capturing approach

Case: damBreak. damBreakFine



phase distribution after breaking the dam for fine (left) and coarse mesh (right)

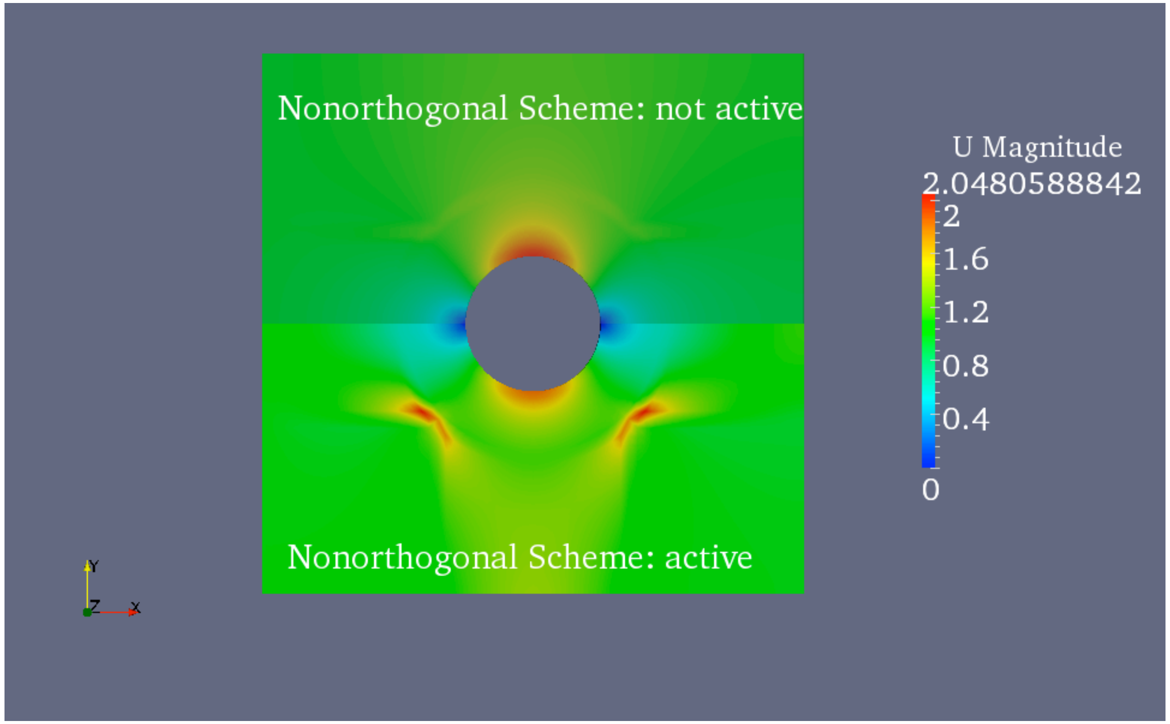




Velocity streamline after breaking the dam for fine (left) and coarse mesh (right)

Solver:potentialFoam:: Simple potential flow solver can be used to generate starting fields for full Navier-Stokes codes

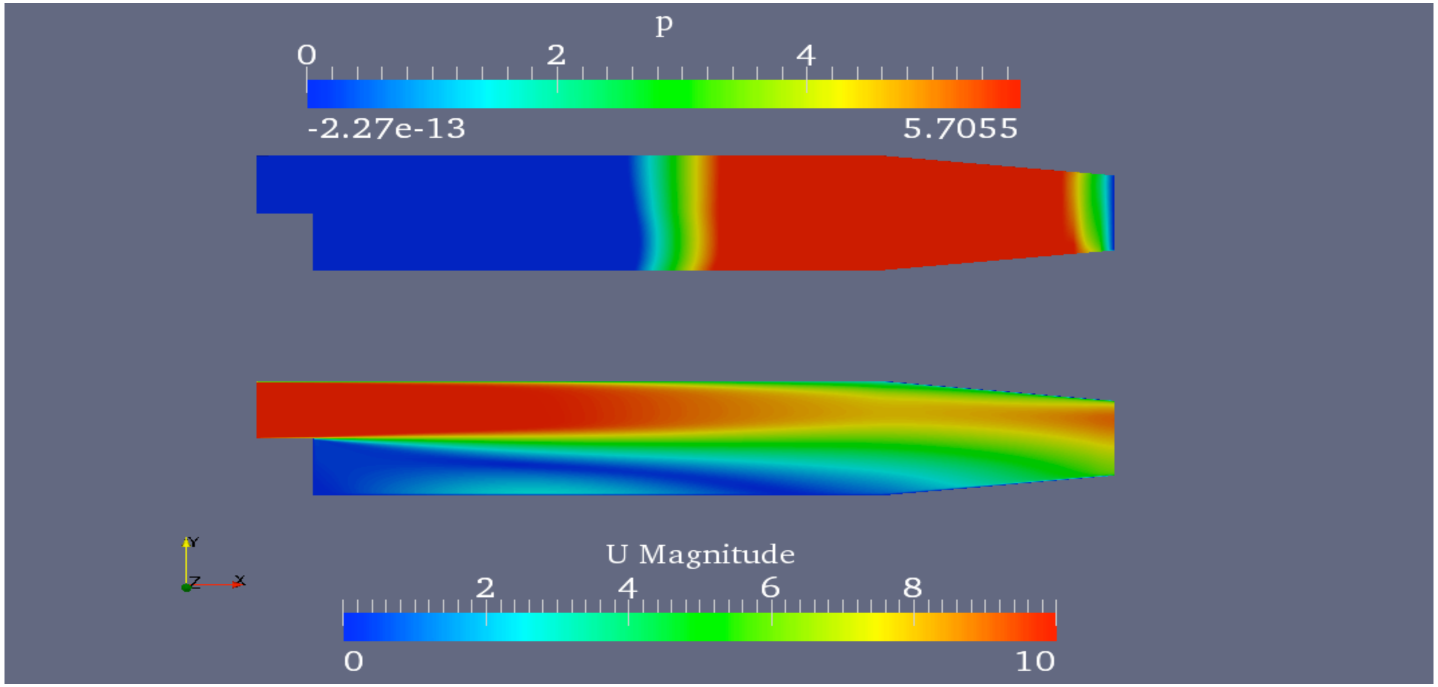
Case: cylinder



Comparison the effects of non-orthogonality corrector option

Solver: **simpleFoam:** Steady-state solver for incompressible, turbulent flow

Case: pitzDaily



Contour of Velocity Field and Pressure Field

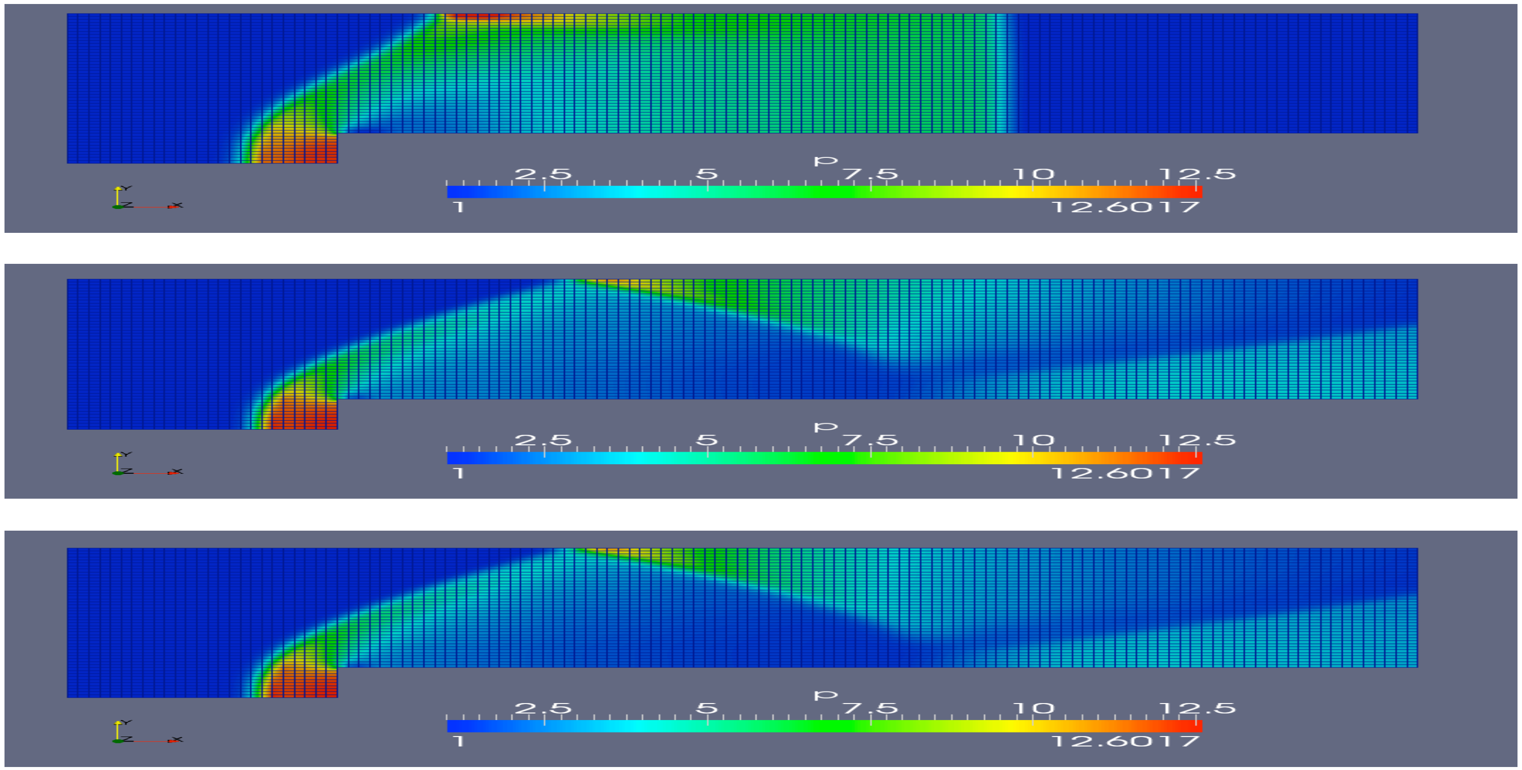
Solver: **sonicFoam:** Transient solver for trans-sonic/supersonic, laminar or turbulent flow of a compressible gas

Case: forwardStep

t1

t2

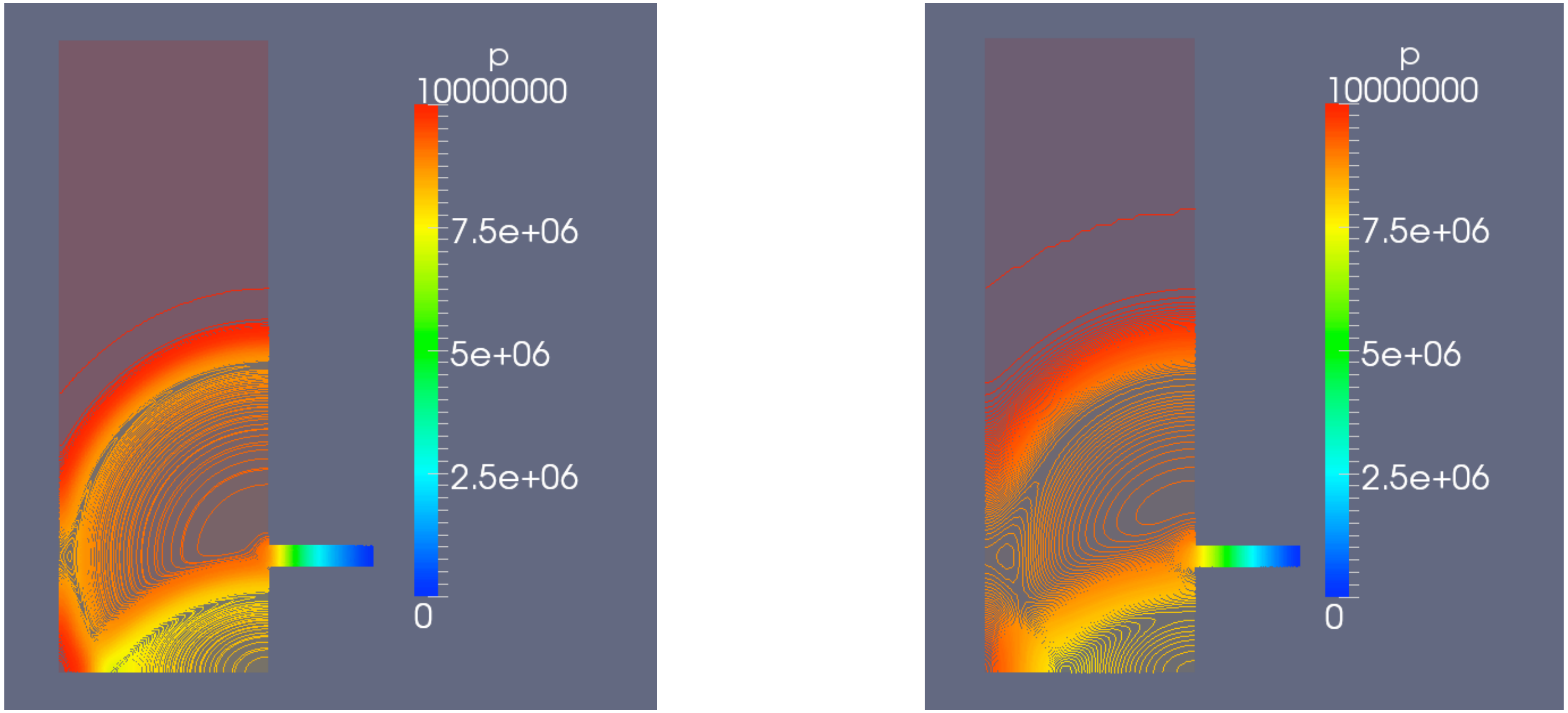
t3



Pressure contour in forward step for various time step (t3>t2>t1)

Solver: **sonicLiquidFoam:** Transient solver for trans-sonic/supersonic, laminar flow of a compressible liquid

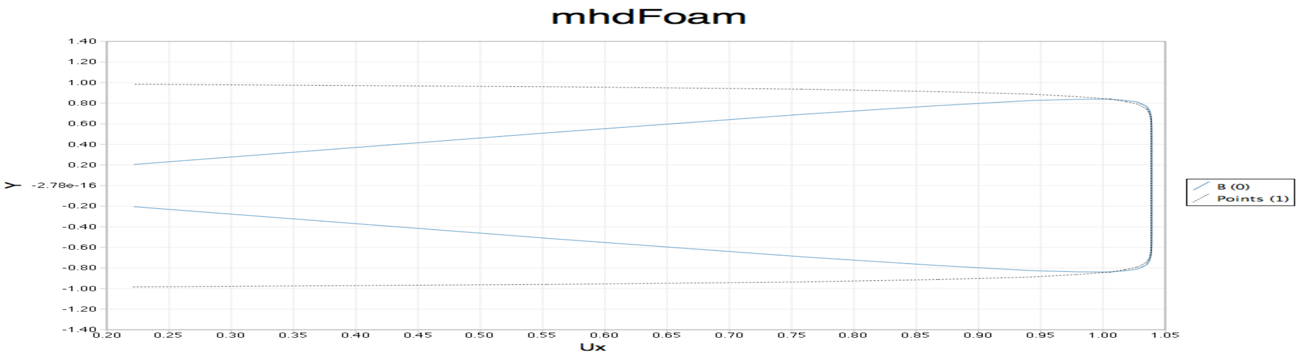
Case: decompressionTank

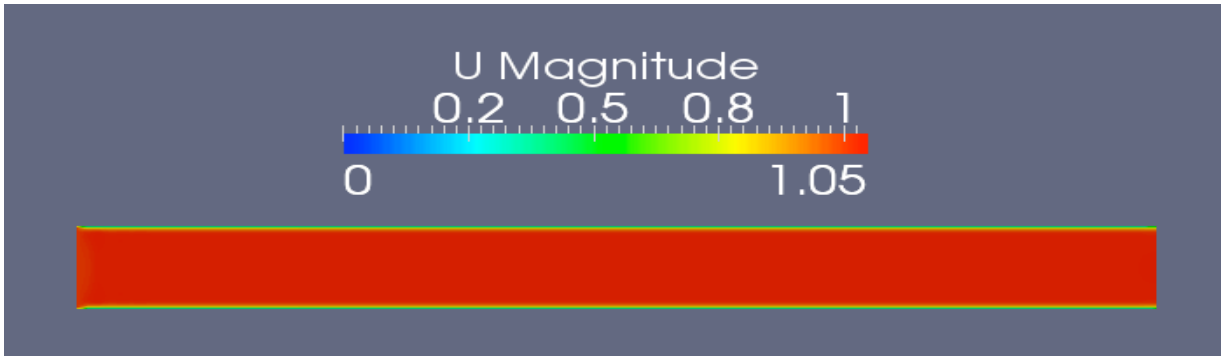


Isosurface pressure for fine mesh(right) and coarse mesh(left)

Solver: **mhdFoam:** Molecular dynamics solver for fluid dynamics

Case: hartmann



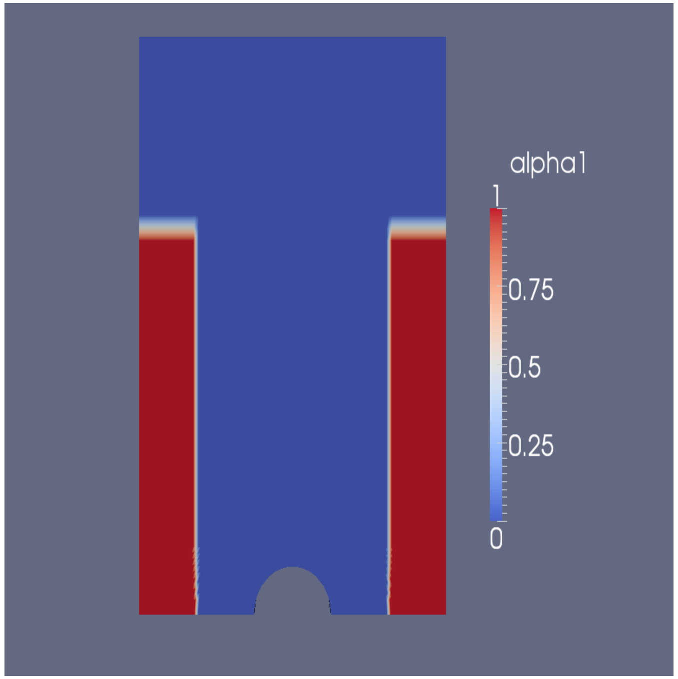


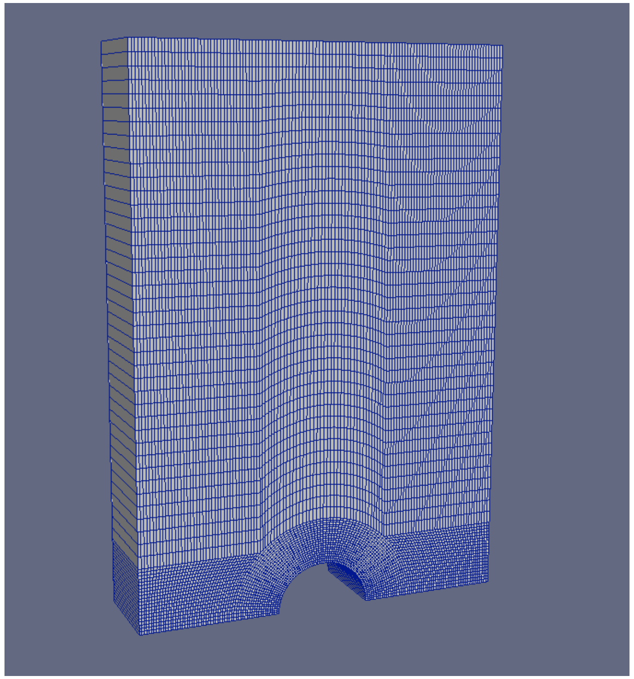
Solver: : **interFoam**

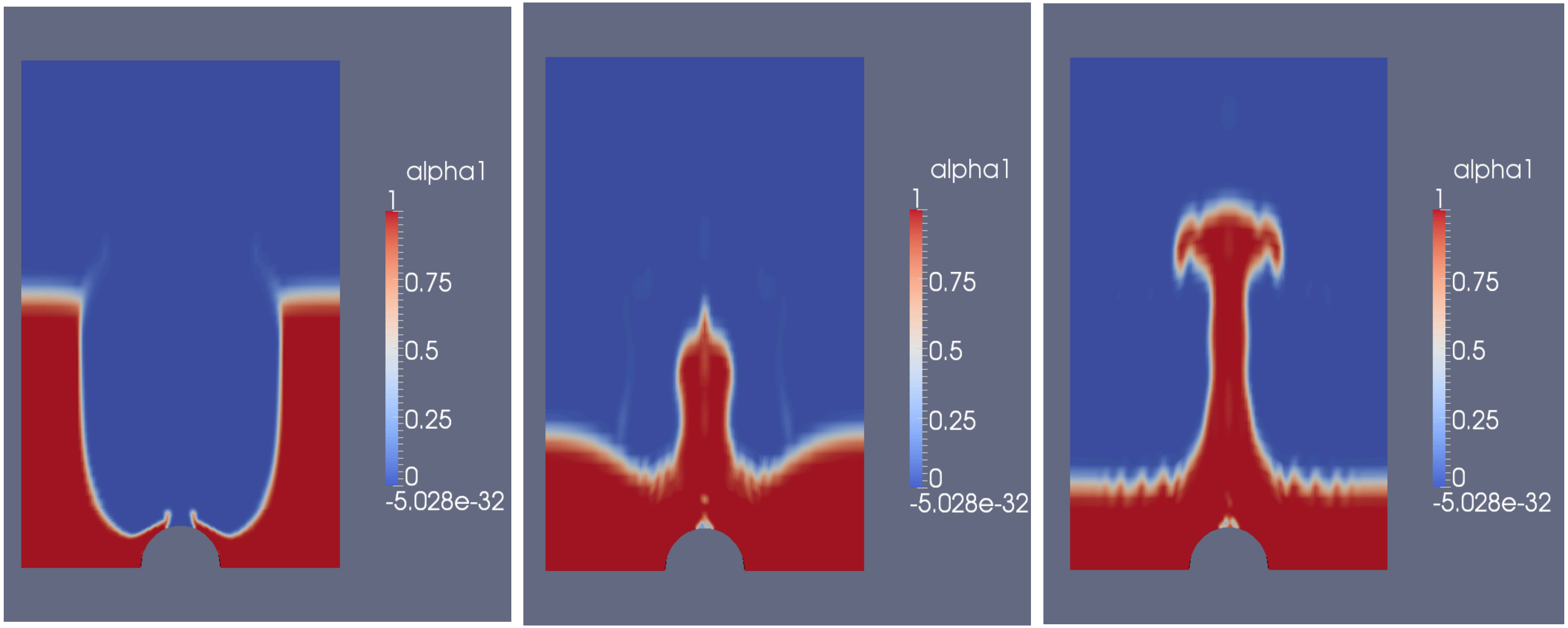
Case: damBreakModified

Description:

The geometry of the dam has been modified and changed to semi cylinder. Besides; the initial condition of phase has been changed. By using setFields similar phase is implemented on the other side of the dam.

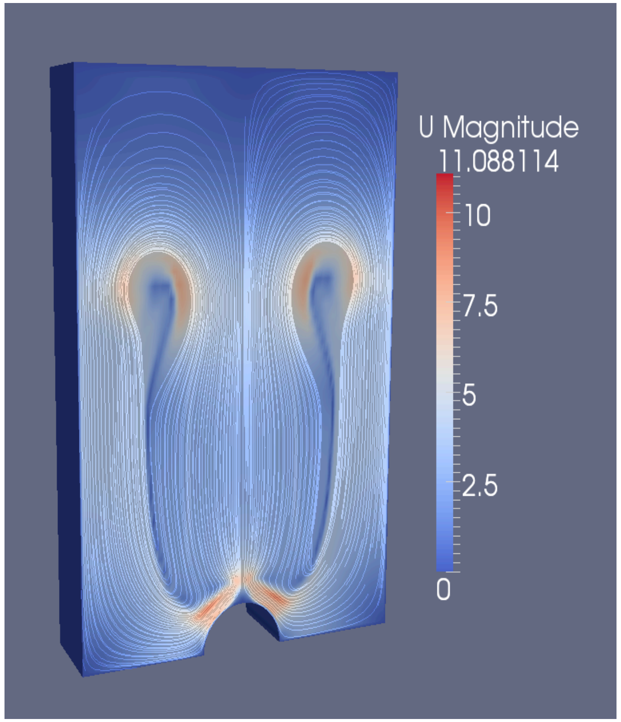




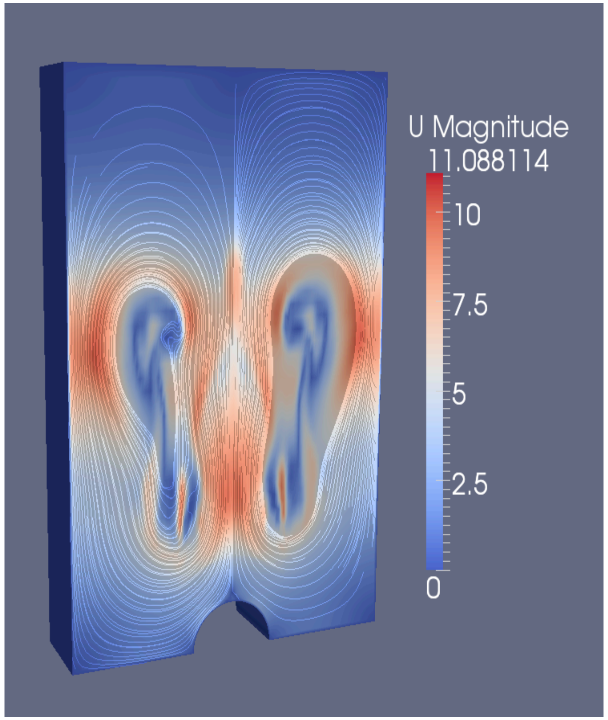


t1 **t2** **t3**

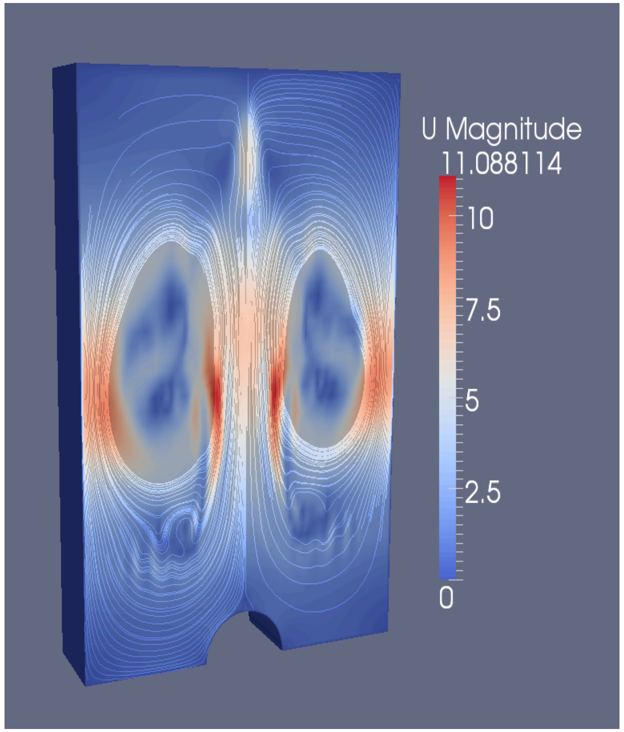
Phase distribution after breaking the dam during various time step. (t3> t2> t1)



t1



t2



t3

Velocity Streamline after breaking the dam during various time step. (t3> t2> t1)

SetField Modification:

```
defaultFieldValues
(
   volScalarFieldValue alpha1 0
regions
(
   boxToCell
   {
      box (-2 0 -.4) (-1.25 4 0.4);
      fieldValues
      (
         volScalarFieldValue alpha1 1
      );
   }
   boxToCell
   {
    box (1.25 0 -.4) (2 4 .4);
      fieldValues
      (
         volScalarFieldValue alpha1 1
      );
   }
);
```