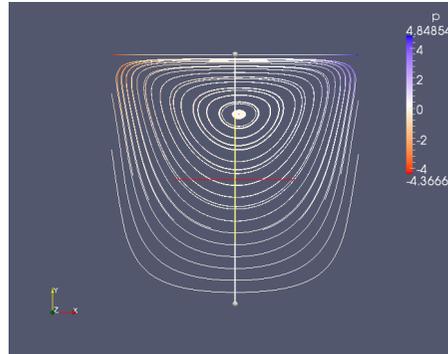


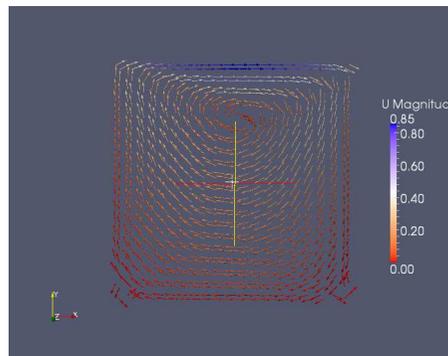
Tutorial and My Case presentation

The following slides will present the Tutorials in OpenFOAM.
In the end my case will be presented as a new Tutorial.

Case: cavity

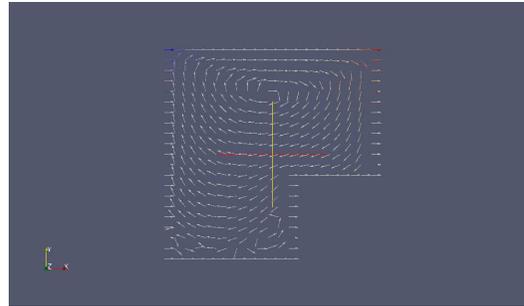


The StreamTracer which is visualized in the figure above is achieved in paraFoam by first making a plane and then using the Filter StreamTracer.



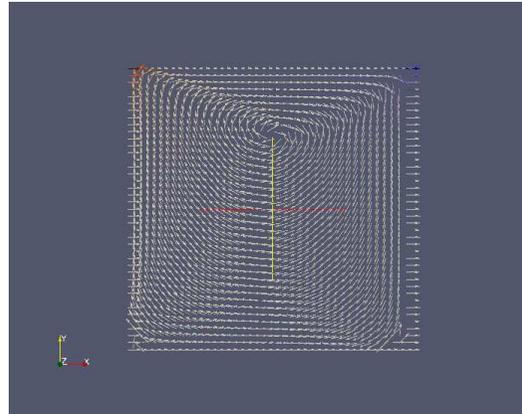
The velocity vectors shown in this figure is obtained by using the Glyph Filter. The color in the figure visualizes the velocity of the flow.

Case: cavityClipped

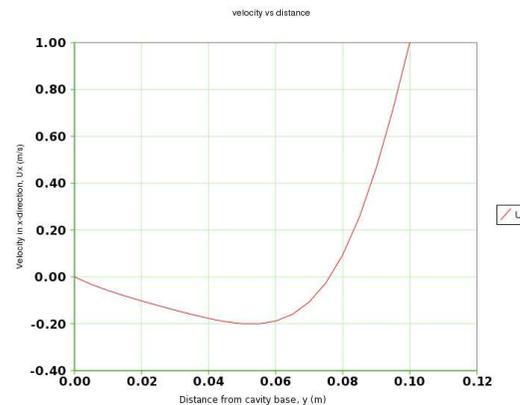


This figure shows the velocity vectors of the flow for the cavityClipped case. In the right lower edge a vertex is visualized.

Case: cavityFine

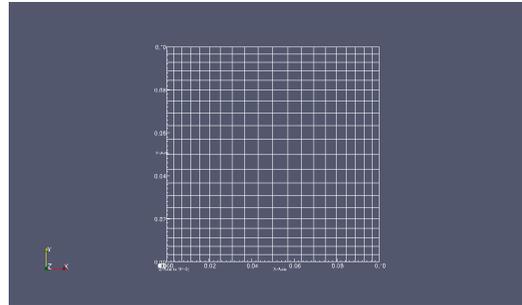


This figure visualizes the velocity vectors for the cavityFine case.

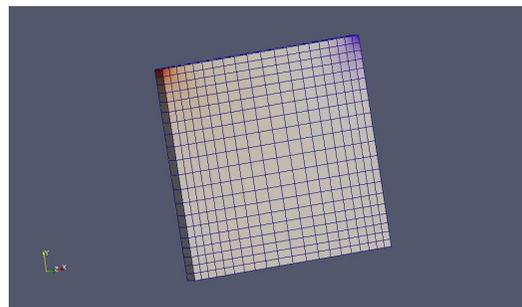


The plot is obtained by using the Filter PlotOverLine. Here the velocity in the x-direction is plotted versus the distance from the cavity base.

Case: cavityGrade

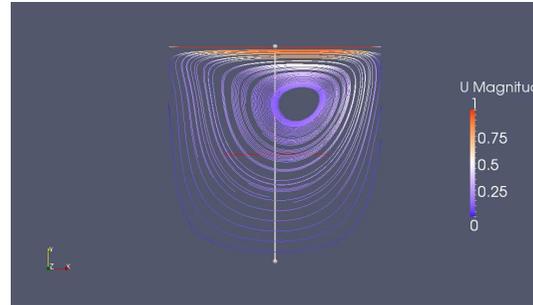


To grade the mesh some parameters in the blockMeshDict file has to be changed, this file is found under the constant/polyMesh/ directory. The last bracket at each block should be changed in order to achieve a graded mesh.



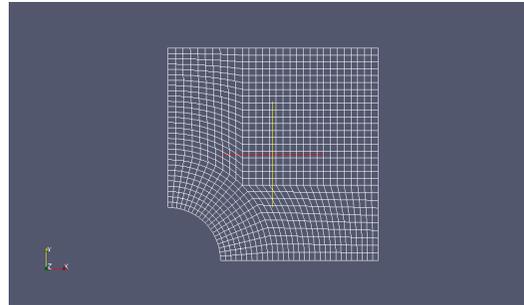
In this figure the wireframe is seen for the case cavityGrade.

Case: cavityHighRe

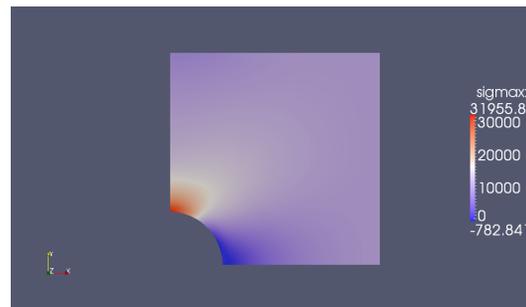


In this figure the streamlines for higher Reynolds number is shown together with the velocity magnitude for the CavityHiRe case.

Case: plateHole

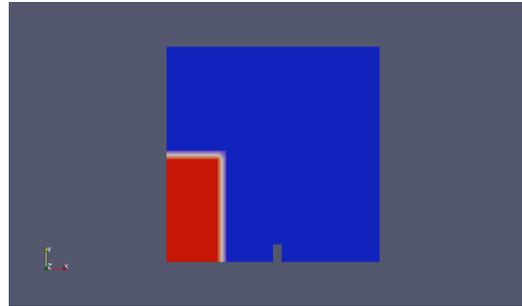


The figure above shows the wireframe for the plateHole tutorial.

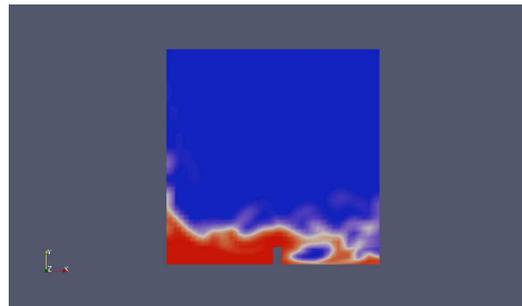


The stress in the x-direction is viewed above. The stress in the x-direction must be imported in paraFoam to be able to visualize it.

Case: damBreak

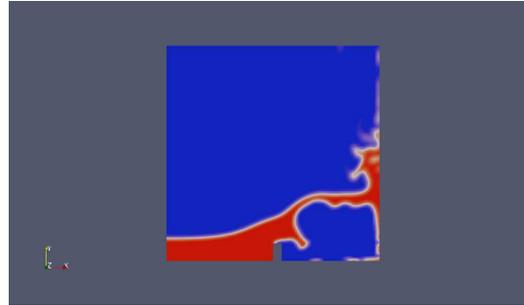


This figure shows the layout for the plateHole tutorial before the membrane is removed.



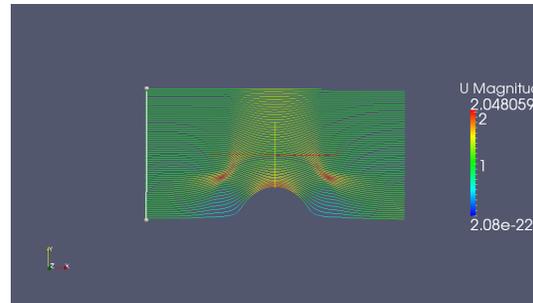
In this figure the membrane is removed and the water flows out from the tank.

Case: damBreakFine

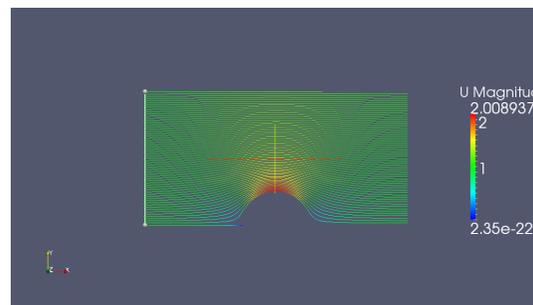


In the tutorial damBreakFine the number of cells is increased. This is performed by changing in the blockMeshDict file.

Case: cylinder

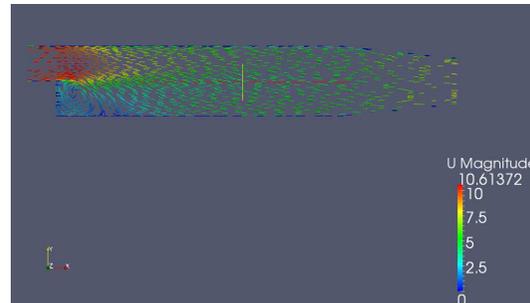


This figure visualizes the streamlines around the cylinder when there is no non-orthogonal correction.

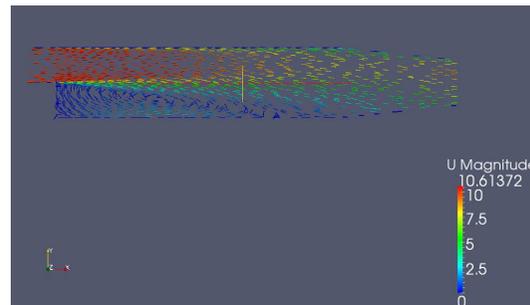


Now the geometry is visualized with some non-orthogonal correction. This is performed by setting the `nNonOrthogonalCorrectors` to 3, this file is placed in the `fvSolution` map.

Case: pitzDaily

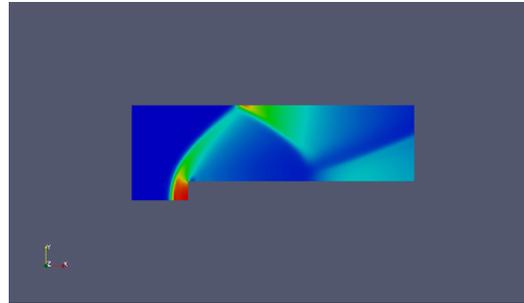


This figure views the velocity magnitude through the 2D geometry at 50 iterations.



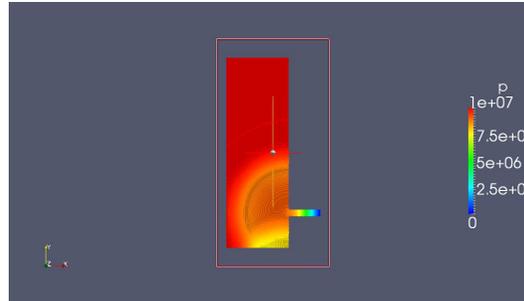
This figure shows the velocity at 1000 iterations.

Case: forwardStep



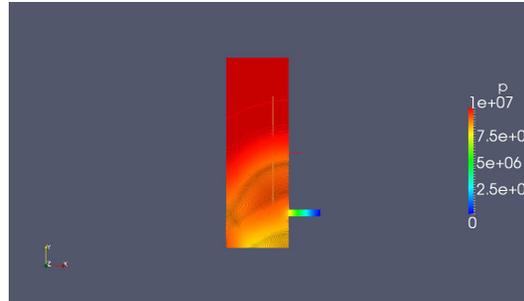
In this figure the shock frons is seen for the forwardStep case.

Case: decompressionTank



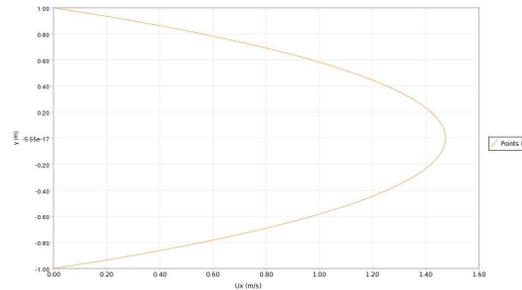
In this figure the pressure waves propagation is visualized. There is a low pressure at the pipe which increases further out in the tank.

Case: decompressionTankFine

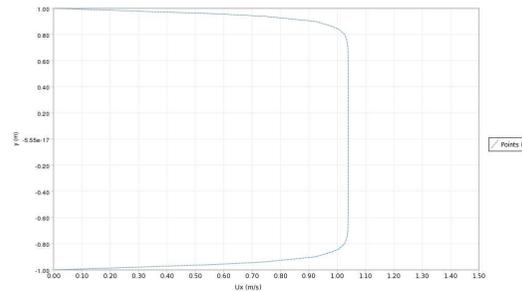


The only difference between this figure and the figure at the previous slide is that this geometry has a higher resolution.

Case: hartmann



These two graphs shows the velocity profile in the hartmann problem for $B_y=1\text{T}$ and $B_y=20\text{T}$.



My Case

The reference to this case is the cavityClipped case. The changes that is going to be made for this case are that:

- the box is going to be extended in the x-direction.
- the number of cells is going to be duplicated.
- the moving lid, at the top, is going to be stationary.
- and there will be an airflow with a velocity of $(1\ 0\ 0)$ m/s coming in from the left wall.

My Case

To change the geometry of the box the control/polyMesh/blockMeshDict file has to be modified. In this case the box is going to be extended 0.04 m in the x-direction. To make this geometrical change the defined vertices in the file have to be changed in the x-direction for the points that is going to be moved.

In this case there is only 4 vertices that needs to be changed in order to achieve the results. Below the vertices are seen for the new geometry. The vertices at row 25, 28, 34 and 37 are changed in the x-direction from a value of 1 to a value of 1.4.

```
19  vertices
20  (
21  (0 0 0)
22  (0.6 0 0)
23  (0 0.4 0)
24  (0.6 0.4 0)
25  (1.4 0.4 0)
26  (0 1 0)
27  (0.6 1 0)
28  (1.4 1 0)
29
30  (0 0 0.1)
31  (0.6 0 0.1)
32  (0 0.4 0.1)
33  (0.6 0.4 0.1)
34  (1.4 0.4 0.1)
35  (0 1 0.1)
36  (0.6 1 0.1)
37  (1.4 1 0.1)
38
39  );
```

My Case

The next step is to improve the resolution of the geometry. This is performed by increasing the number of cells in the control/polyMesh/blockMeshDict file. In this case the number of cells is going to be doubled in the x- and y-direction. To increase the resolution the second bracket in the 'blocks' has to be multiplied by two. The changes is seen below.

```
blocks
(
hex (0 1 3 2 8 9 11 10) (24 16 1) simpleGrading (1 1 1)
hex (2 3 6 5 10 11 14 13) (24 24 1) simpleGrading (1 1 1)
hex (3 4 7 6 11 12 15 14) (32 24 1) simpleGrading (1 1 1)
);
```

When the resolution of the mesh is increased it is importenet to recalculate the Courant number. This is performed to achieve a more accurate and numerical sstability when running icoFoam.

To achieve a Courant number lower than 1 the timestep, which is defined in the system/controlDict file, needs to be set to 0,0033.

My Case

The third and fourth steps in this case can be performed at the same time. The third step is to change the moving lid, at the top, and make it stationary. The fourth step is to add an airflow with a velocity of $(1\ 0\ 0)$ m/s coming in from the left wall. To make these changes these three files have to be modified:

- the control/polyMesh/blockMeshDict file
- the 0/U file
- and the 0/p file

My Case

In the control/polyMesh/blockMeshDict file the patches have to be changed. The moving lid can be deleted and the right and left side of the box must be added. These two walls are going to be patches since the air is flowing through them. The new code is seen below.

```
patches
patch left
(0 8 10 2)
(2 10 13 5)
patch right
(7 15 12 4)
```

My Case

The velocity of the flow of each type of the geometry is defined in the 0/U file. The two patches that were added in the blockMeshDict file must also be defined in the 0/U file. The velocity at the left wall is constant and it is set to (1 0 0), to indicate that there is a velocity of 1 m/s in the x-direction.

There is no defined velocity at the right wall and it is therefore set to zeroGradient. Below the velocity definition for the two new boundaryFields are shown.

```
boundaryField
left
type    fixedValue;
value   uniform (1 0 0);
right
type    zeroGradient;
```

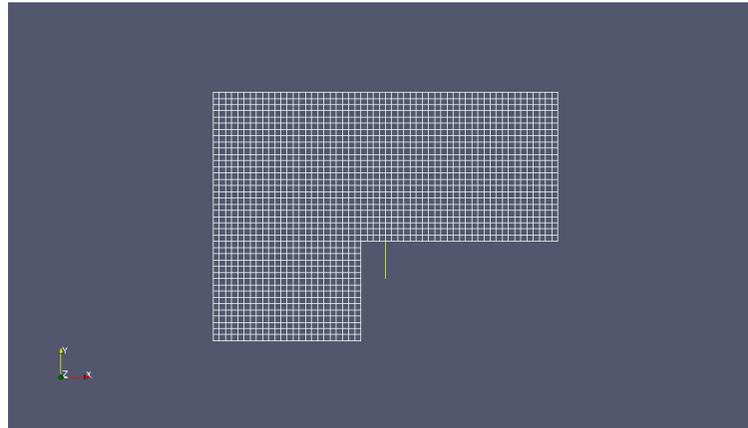
My Case

The pressure of each type of the geometry is defined in the 0/p file. The two patches that were added in the blockMeshDict file must also be defined in this file. The pressure at the left wall is set to zeroGradient. The pressure at the right wall is set to 0. Below the pressure definition for the two new boundaryFields are shown.

```
boundaryField
left
type zeroGradient;
right
type fixedValue;
value uniform 0;
```

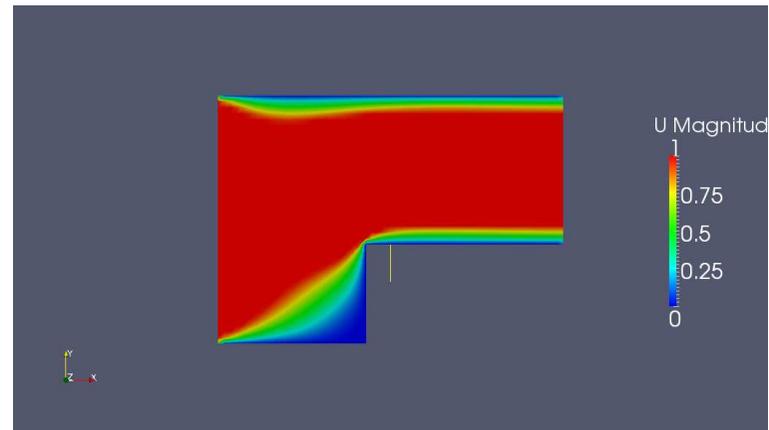
My Case-Results

In the figure below the wireframe for the new geometry is shown. The number of cells have been doubled and the geometry is also extended in the x-direction.



Case: cavity

In the figure below the velocity magnitude inside the box is visualized.



Case: cavity

In the figures below the velocity vectors and streamlines inside the box are visualized.

