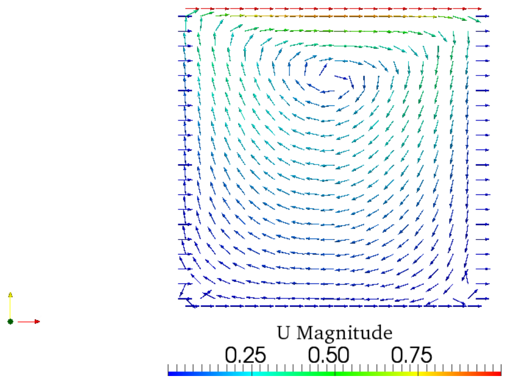
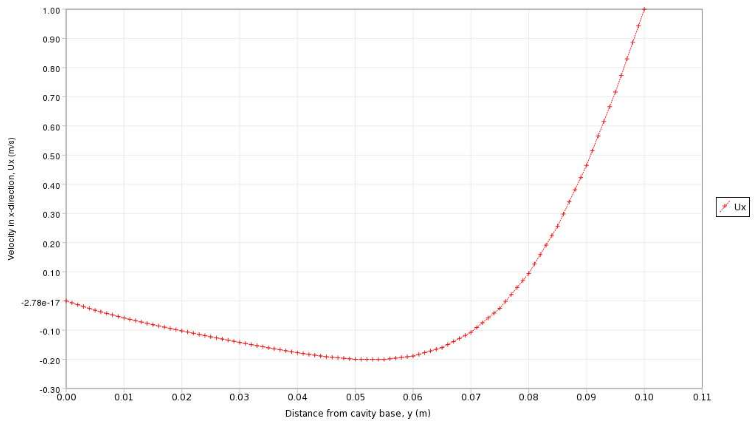
icoFoam: cavity



- In ParaFoam, **All Mesh Parts** included.
- Make a **slice** of the part - Z-axis normal.
- Surface representing streamlines colored by the velocity.
- Streamlines - **Scale factor:** 0.005

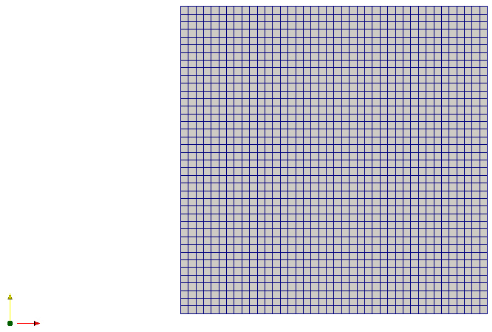


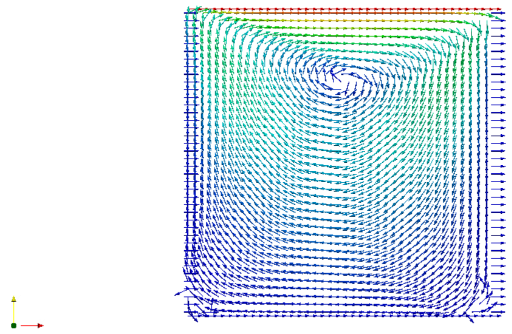
- Extract velocity components U_x, U_y, U_z:

  `foamCalc components U`

- All **patches** are deselected, otherwise incorrect interpolation.
- **Filter - Plot over Line**.
- Plot showing U_x as a function of the distance from the cavity base.

icoFoam: cavityFine

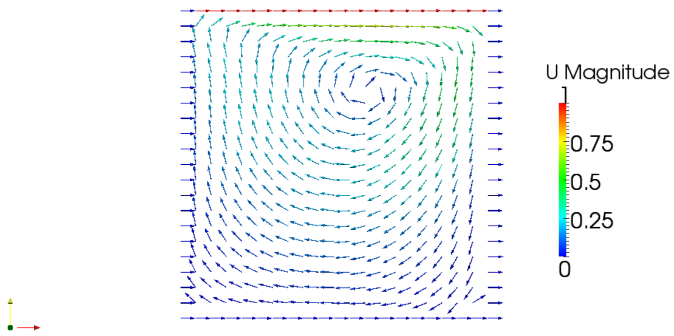


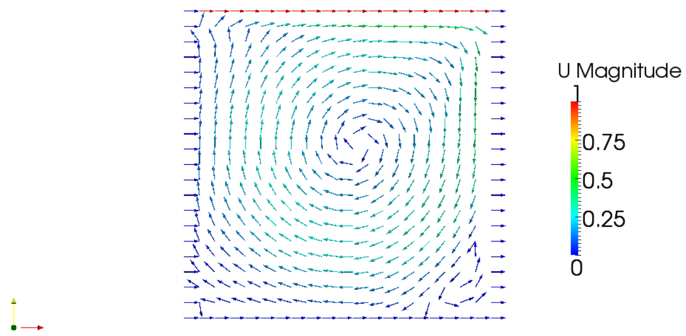


- All Mesh Parts included in ParaFoam.
- Surface representation : **Surface with edges**, showing the mesh.

- Make a **slice** of the part - Z normal.
- Visualizing streamlines colored by the velocity magnitude.

icoFoam: cavityHighRe

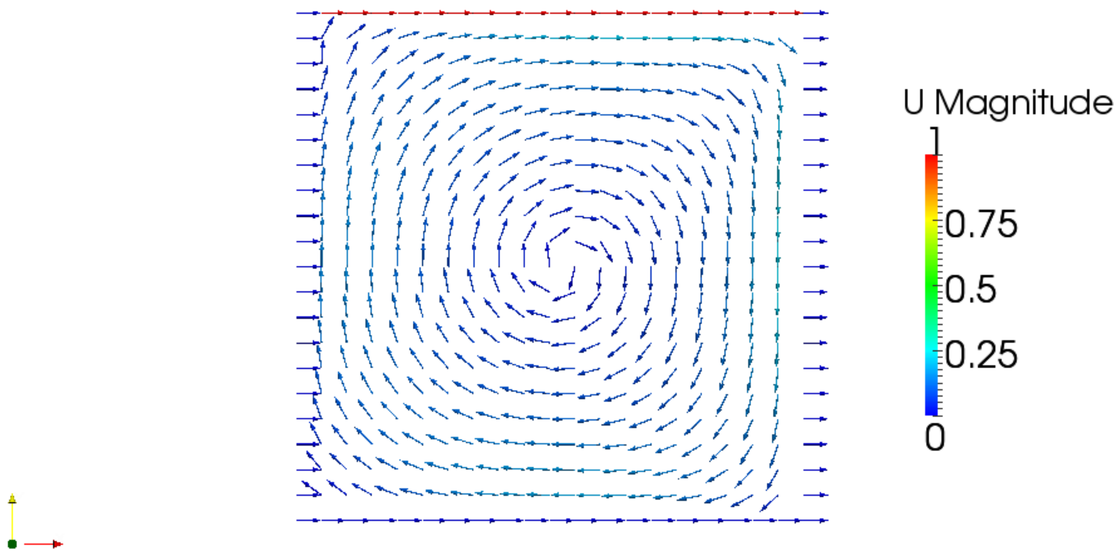




- Make a **slice** of the part - Z-axis normal.
- Using **Stream Tracer** to view streamlines, colored by the velocity magnitude.
- The kinematic viscosity, $\nu = 0.001$.

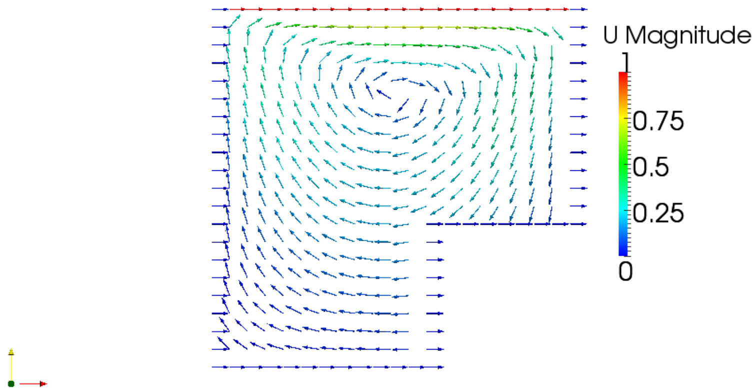
- Visualizing streamlines colored by the velocity magnitude.
- The kinematic viscosity, $\nu = 0.0001$, yielding a higher Re number.

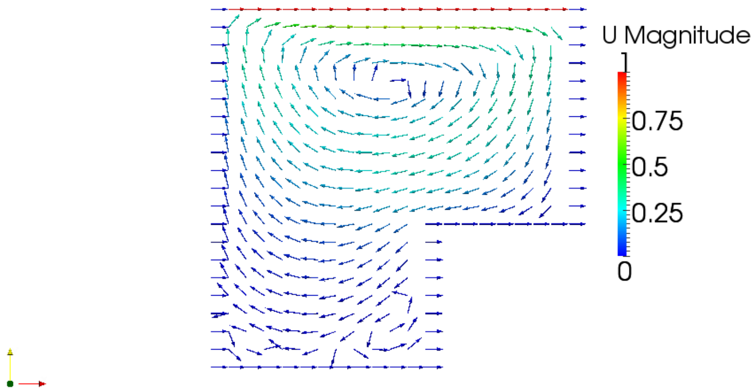
pisoFoam: cavity



- Make a **slice** of the part - Z normal.
- Visualizing streamlines colored by the velocity magnitude.
- Using the $k\epsilon$-turbulence model to model the turbulence.

icoFoam: cavityClipped

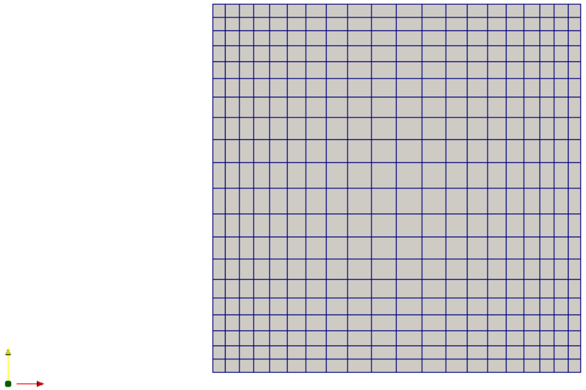


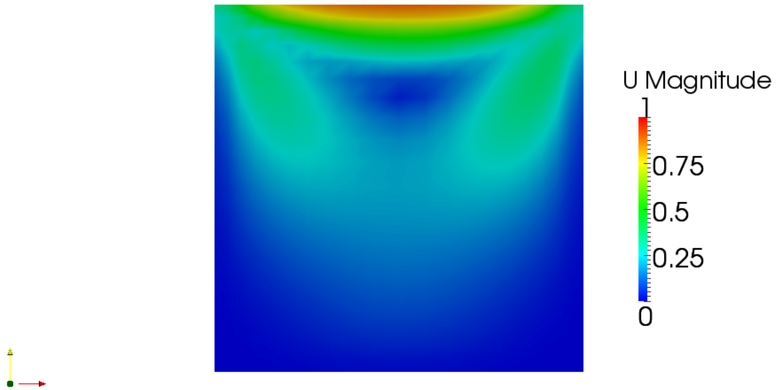


- Comparison of the solution at two different time steps.
- Showing the solution at time $0.5\,seconds$.
- Visualizing streamlines colored by the velocity magnitude.

- Showing the solution at time $0.6\,seconds$. Note the vortex structure a the lower part of the figure.
- Visualizing streamlines colored by the velocity magnitude.

icoFoam: cavityGrade

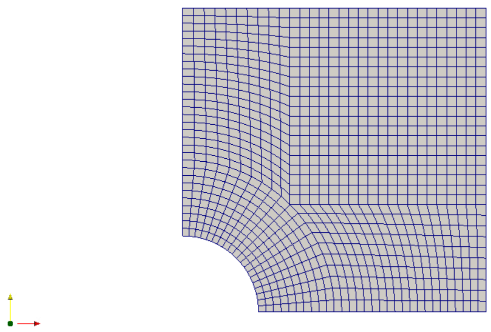


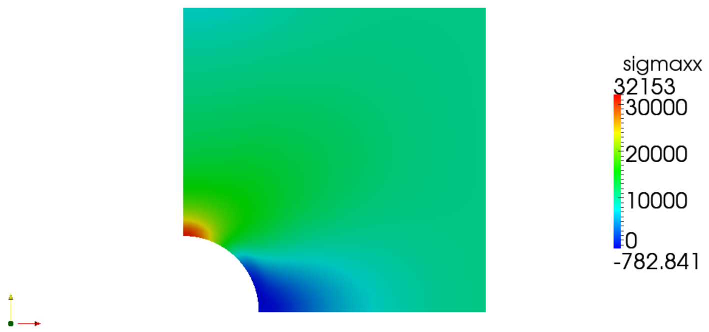


- Figure showing the mesh of the case, here a graded mesh is used.
- All Mesh Parts are included. In the figure, both surface and edges are displayed.

- Figure showing the velocity field (velocity magnitude).
- Visualization: Surface colored by the velocity at the last time step.

solidDisplacementFoam: plateHole





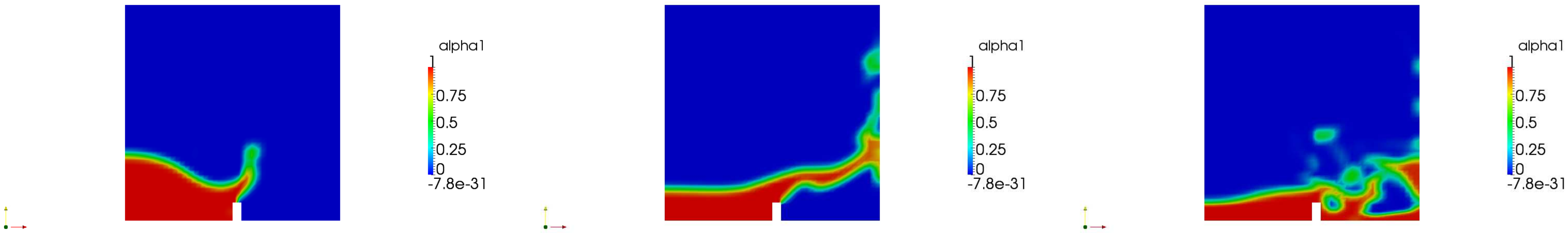
- Figure showing the mesh resolution of the case.
- **All Mesh Parts** are included. **Surface and edges** are displayed in the figure.

- The figure shows the stress component σ_{xx}.
- The stress components are obtained by using

  `foamCalc components sigma`

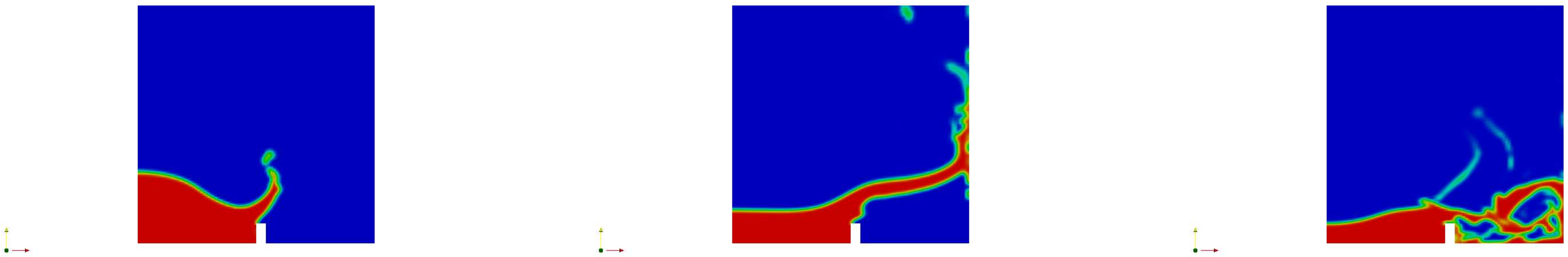
  in the command line.
- Visualization: Surface colored by σ_{xx}.

interFoam: damBreak



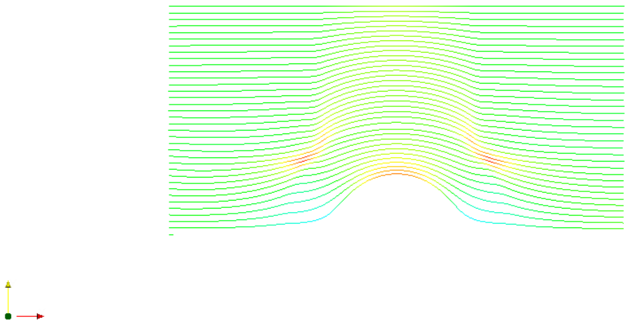
- Figures showing the phase fraction, α_1, at different steps in the simulation.
- Figures created using **Surface Representation** in the pull-down list.
- Color scheme changed to **Blue to Red** in **Edit Color Map**.

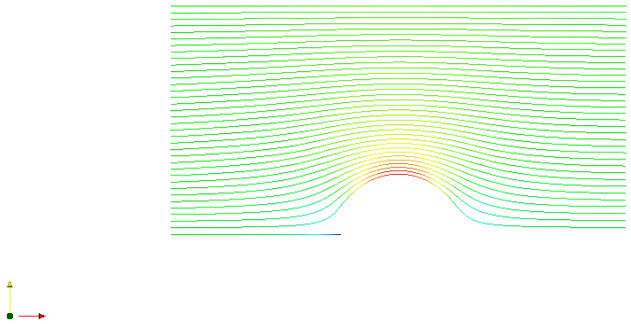
interFoam: damBreakFine



- Figures showing the phase fraction, α_1, at different steps in the simulation.
- Figures created using **Surface Representation** in the pull-down list.
- Note that the figures from **damBreakFine** visualizes the same phenomenon at the same simulation steps as in **damBreak**, but with a higher resolution of the mesh.

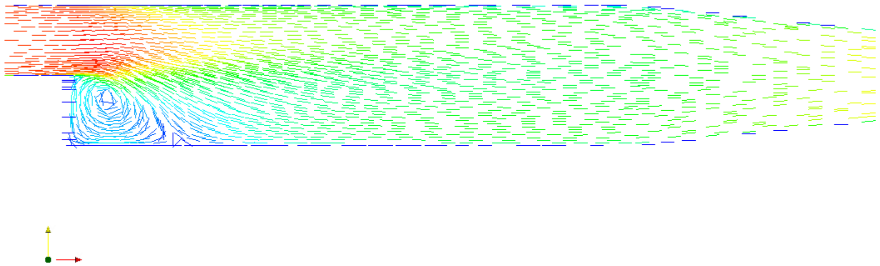
potentialFoam: cylinder

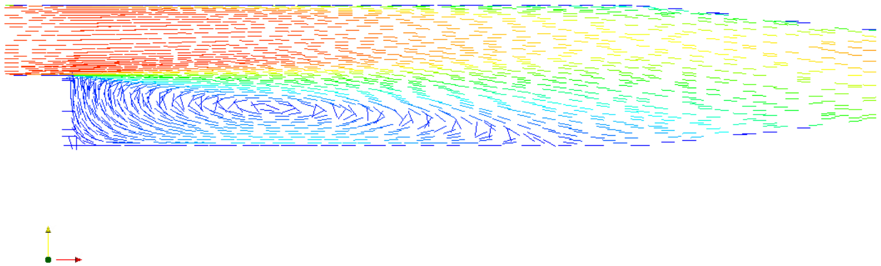




- Figures showing velocity streamlines for *nNonOrthogonalCorrectors* = *0* and *nNonOrthogonalCorrectors* = *3* respectively.
- A plane is created using **Slice** in ParaFoam, on which **Stream Tracer** is used to visualize streamlines.
- Streamlines are colored by the velocity magnitude.

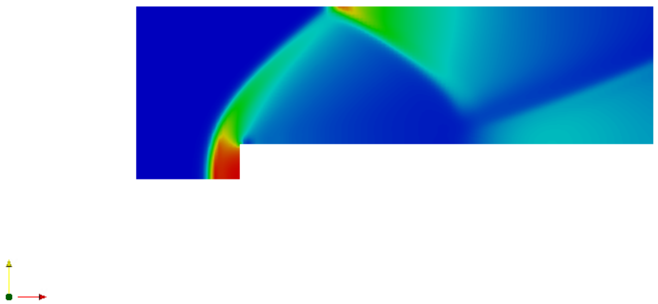
simpleFoam: pitzDaily

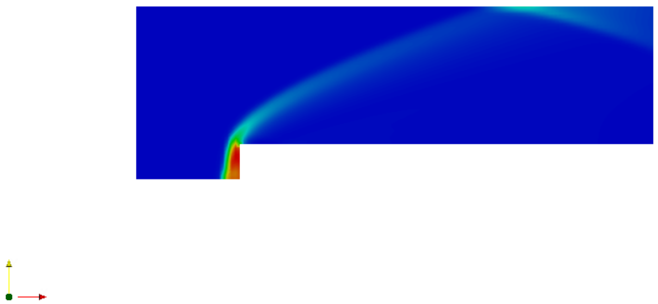




- Figures showing velocity vectors for two different simulation steps.
- A plane is created using **Slice** in ParaFoam, on which **Glyph** is used to visualize velocity vectors.
- The velocity vectors are colored by the velocity magnitude, which is chosen from the pull-down list.

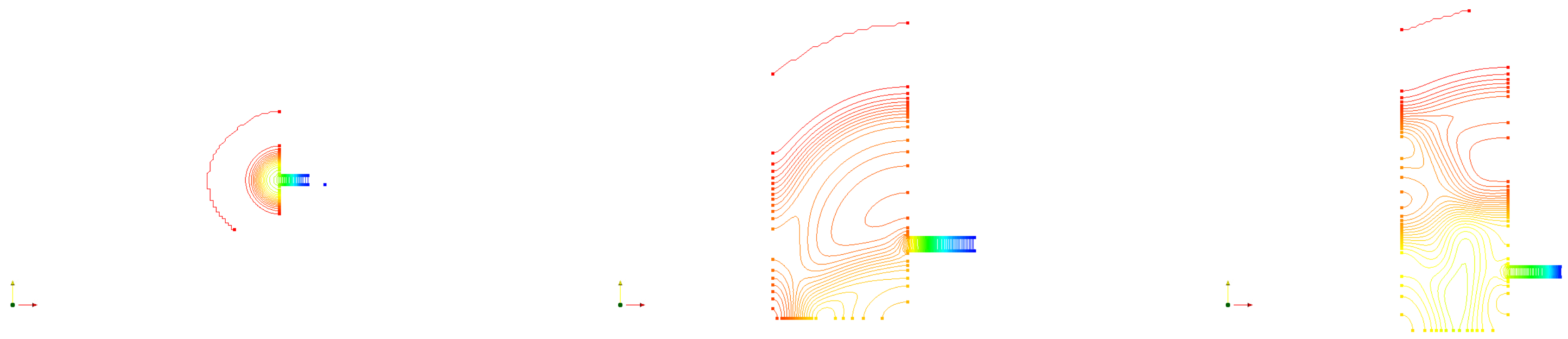
sonicFoam: forwardStep





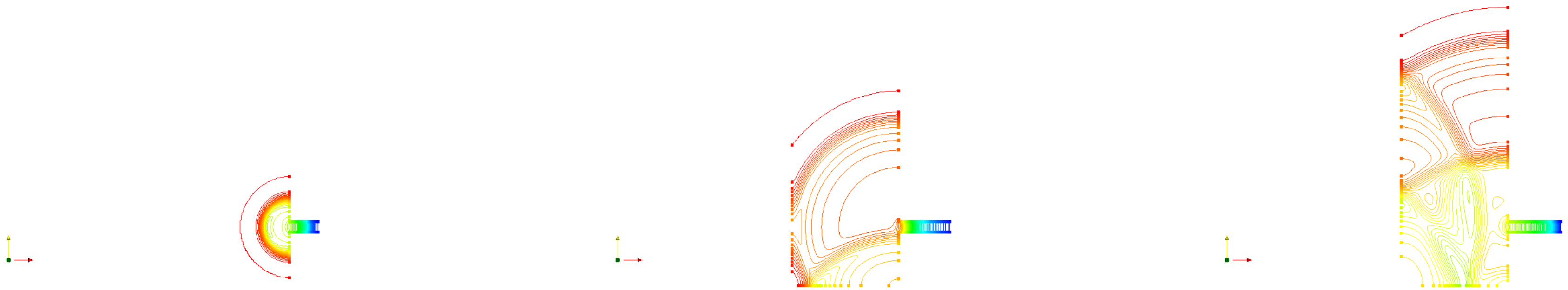
- Figures showing pressure distribution for two different inlet velocities, $v = 3\,\frac{m}{s}$ and $v = 6\,\frac{m}{s}$.
- Using **Slice** in ParaFoam a plane is created with Z-plane as normal vector, on which **Glyph** is used to visualize velocity vectors.
- The velocity vectors are colored by the velocity magnitude, which is chosen from the pull-down list.

sonicLiquidFoam: decompressionTank



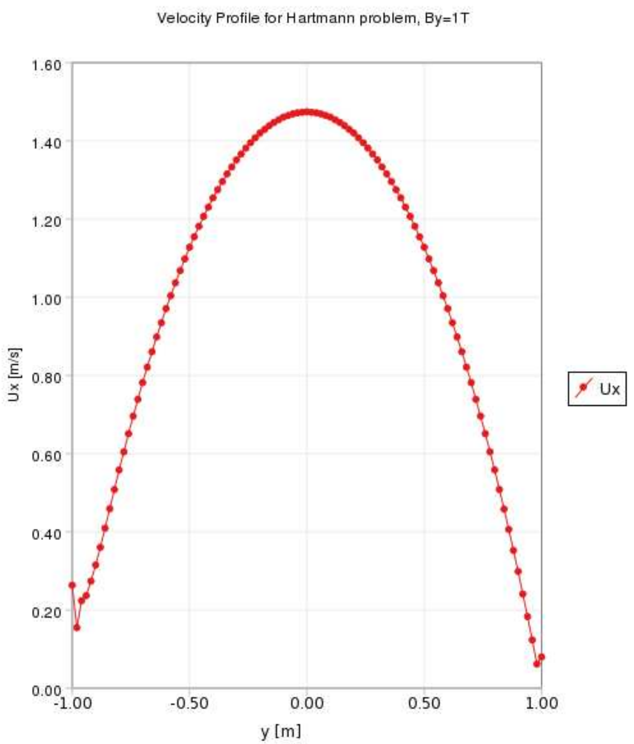
- Figures showing pressure contours at three different simulation times, $50\,\mu s$, $100\,\mu s$ and $150\,\mu s$.
- Using **Slice** in ParaFoam a plane is created with Z-plane as normal vector. On the plane **Contour** is used to visualize the pressure.
- The contours are colored by the pressure, and the **Value Range** is set from 0 to $10^7\,Pa$.

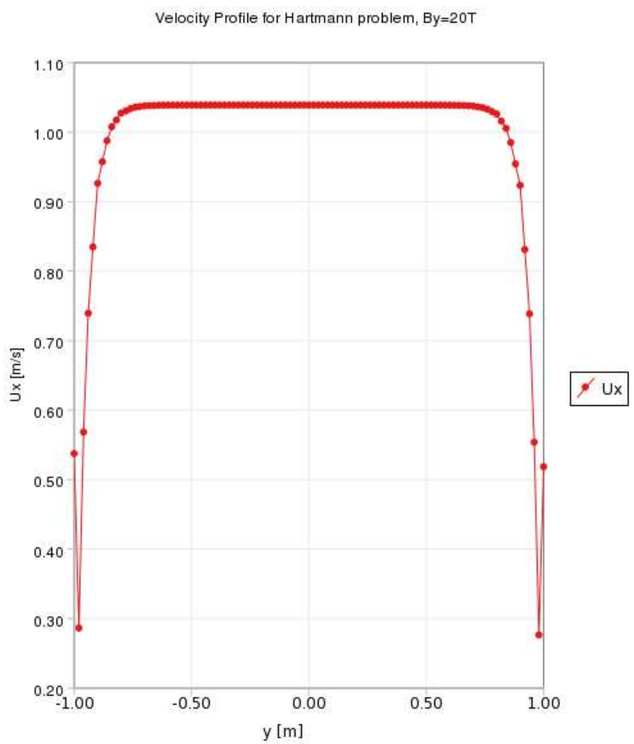
sonicLiquidFoam: decompressionTankFine



- Figures showing pressure contours at three different simulation times, $50\,\mu s$, $100\,\mu s$ and $150\,\mu s$.
- The pressure contours are obtained using the same method as for **decompressionTank**.
- Note that the resolution of the mesh is higher for **decompressionTankFine** than for **decompressionTank** and therefore the figures are somewhat different.

mhdFoam: hartmann





- Plots showing the velocity profile, U_x versus distance in y-direction.
- Plots are obtained by using **Plot over Line** in ParaFoam and choosing U_x as the **Line Series** plotted against **Data Array** *Points (1)*, the distance in y-direction.

Modified tutorial

This is a modification of the **pisoFoam: cavity** tutorial. In the original tutorial the $k\epsilon$-turbulence model is used; here it is decided to change to the $k\omega$-turbulence model to see if there are any differences in the results.

- The value of ω that is to be used in the set-up of the case has to be calculated. This is done by using the following relationship:

  $$\omega = \frac{\sqrt{k}}{C^{1/4} \cdot l} = 14.86904 \frac{1}{s}$$

  where $C \approx 0.09$, $l \approx 0.07 \cdot L$ (L is the characteristic length, $0.1\,m$ in this case) and $k = 0.00325$ according to the original tutorial.

- The calculated value of ω is inserted in a file **omega** located in **0/**. Also, the correct dimensions of the variable ω is inserted in the **dimensions** field. **omegaWallFunction** is used for both the **movingWall** and **fixedWalls** boundaries.

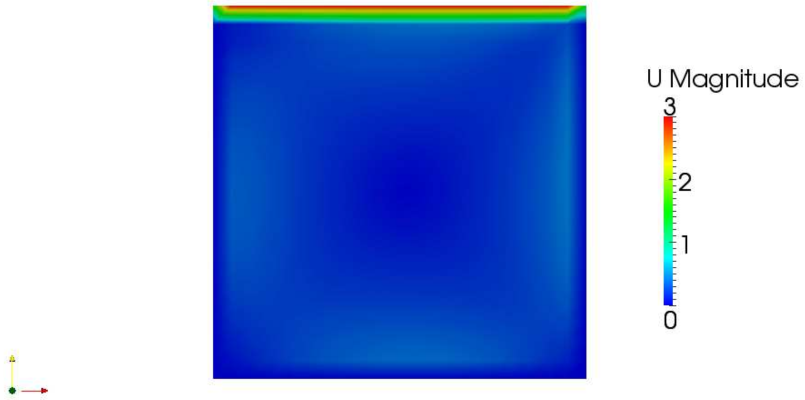
- In **constant/RASProperties**, the $k\omega$-turbulence model has to be chosen.

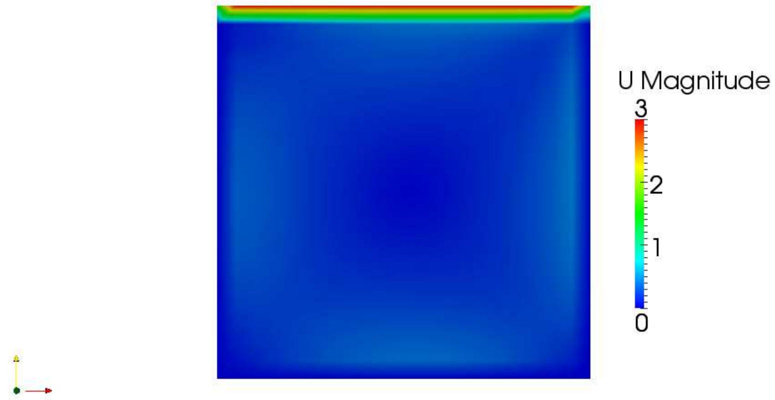
Modified tutorial

- In **system/fvSchemes**, keywords suitable for ω has to be inserted in **divSchemes** and **laplacianSchemes** at the places where keywords for ϵ were put in the original tutorial.
- In **system/fvSolution**, the convergence criteria for ω has to be defined. In this case, the **tolerance** is set to 10^{-5}.
- In **system/controlDict**, the **startTime** is set to **0** and the **endTime** is set to **20** with a **deltaT** of **0.002**.
- The mesh is furthermore created using **blockMesh** and the case is run with the **pisoFoam** solver.

Modified tutorial - Results

The leftmost figure is for the case run with the $k\omega$-turbulence model while the rightmost figure is for the case where the $k\epsilon$-turbulence model is used.





- The velocity field for the two cases are very similar; it is not possible to detect any differences just by comparing the figures.