CFD Results Update

Damiano 06/06/24

CERN

GENERAL APPROACH!

1. GEOMETRY

Recreate the geometries using a 3D model

2. MESHING

Mesh the fluid volume and ensure a good trade-off between mesh quality and computational cost

3. STEADY-STATE

Run a steady-state simulation to understand the equilibrium behavior of the systems

4. TRANSIENT

Focus on transient conditions to visualize the transport and the motion of the fluids

5. ITERATING

Iterate with increasing complexity to obtain a result as close as possible to reality

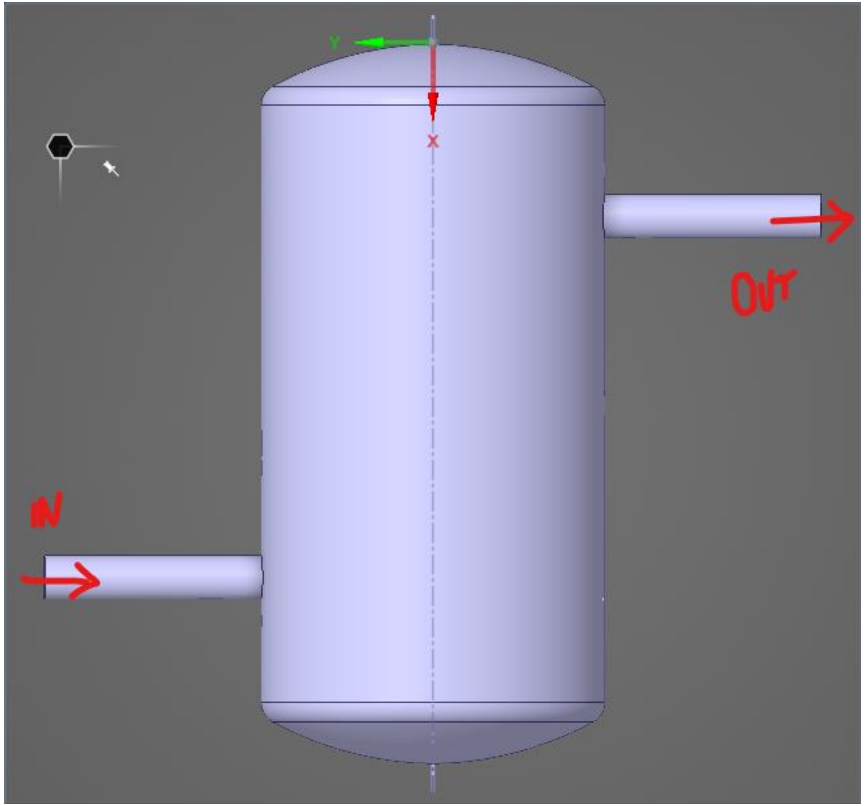
Geometry Definition

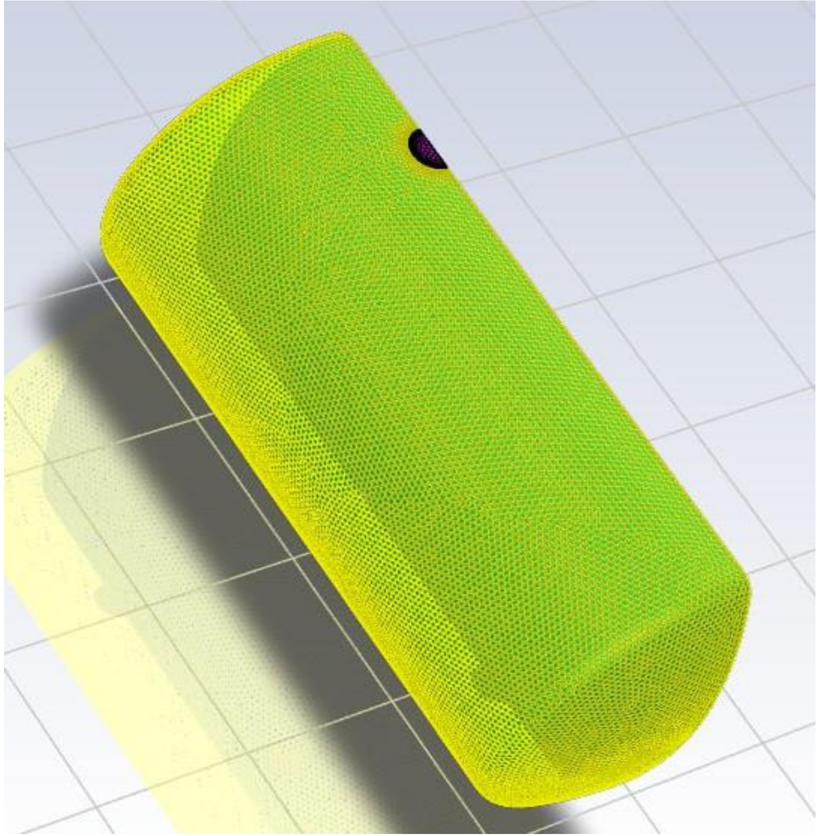
The Tank has the following geometry:

Height = 2.5 m
Inner Diameter = 1.2 m
Inlet Diameter = 0.15 m
Outlet Diameter = 0.15
Inlet Length = 0.75 m
Outlet length = 0.75 m

To reduce the computational complexity the tank has been cut using an XY plane at z = 0.6 m. The geometry is in fact symmetric with respect to vector cutting the inlet pipe in half.

Inlet and outlet pipes have a ratio L/D = 5 to guarantee a completely developed flux





- Number of Cells > 8e+5
- Min/Avg Orthogonal Quality = 0.4/0.98

Meshing

a. Surface mesh:
   a. Min. cell size = 1.5e-4 m (to have 20 cells along the radial direction of the pipes)
   b. Max. cell size = 1.5e-2 m (100 times the minimum cell size)

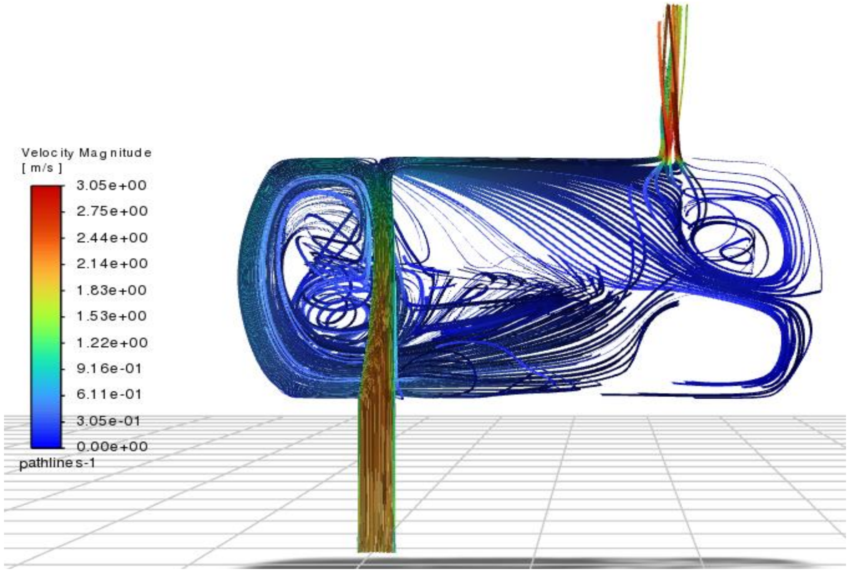
c. Boundaries:
   i. **Inlet: set velocity (2 m/s)**
   ii. Outlet: set pressure
   iii. Symmetry: cutting the fluid domain along the plane XY
   iv. Walls: no-slip condition

d. Boundary layer:
   i. Uniform
   ii. 10 layers with a 1.2 growth rate
   iii. First height 1.5e-3 m to have y+ = 5 within the high gradient region

e. Volume mesh
   i. Polyhedral, growth rate 1.1 and 15e-3 max length

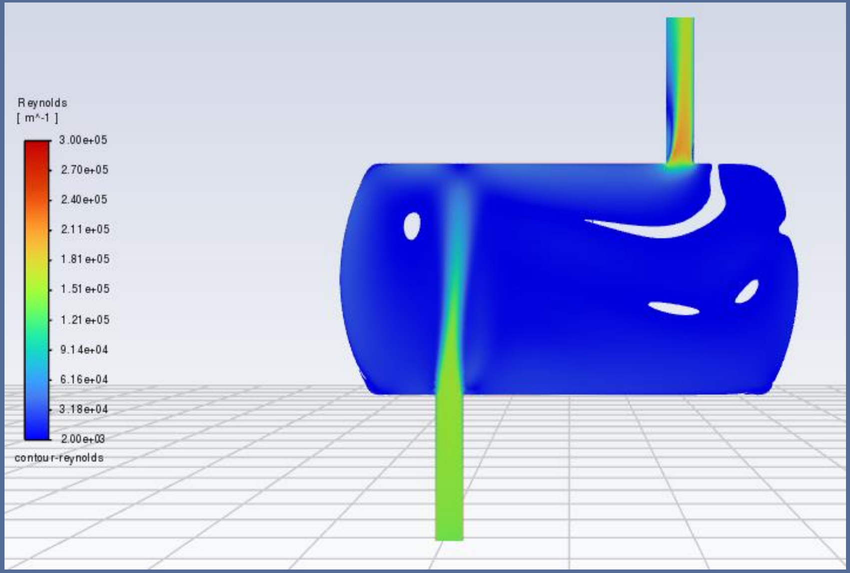
AIR ONLY STEADY STATE

- Inlet Velocity = 2 m/s
- Pressure = 1 atm



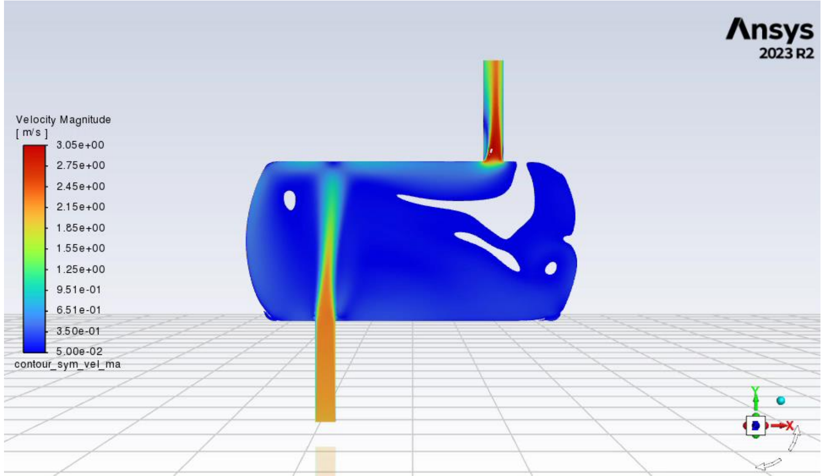
In Steady-State conditions, it is observable that the turbulence is high enough to guarantee a Re>2000 in almost the entire volume**

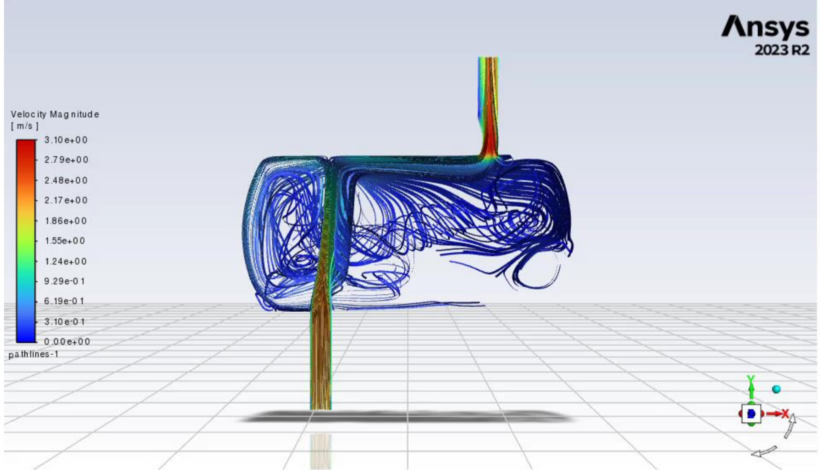
- Inlet Velocity = 2 m/s
- Pressure = 5 atm



In this second picture, we can visualize the turbulence. The empty zones are the one not respecting the constrain (where Re <2000)

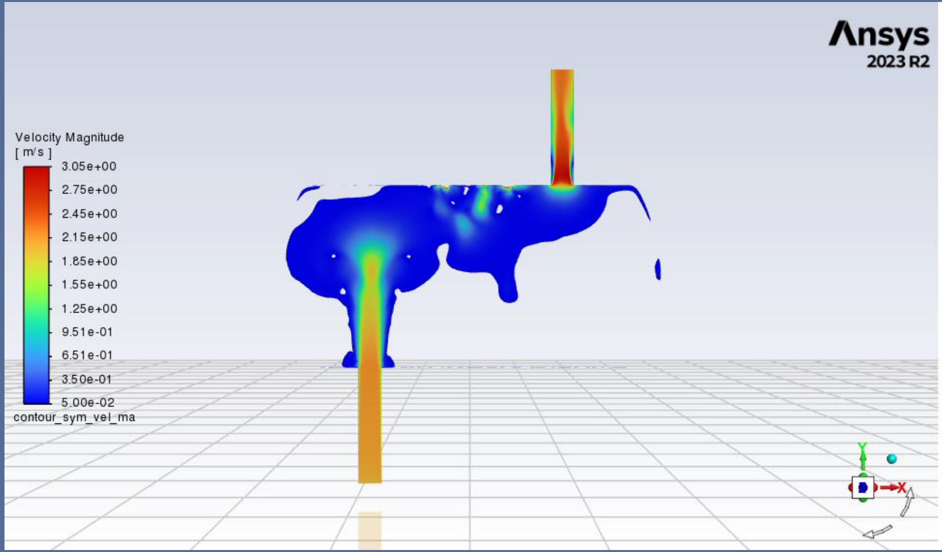
AIR ONLY TRANSIENTS





The left-side videos represent transient conditions after the convergence of a ss solution.

The video on the right side is more interesting as it reports the filling process from the starting time. It is possible to see that acceptable turbulence is reached after 8 seconds

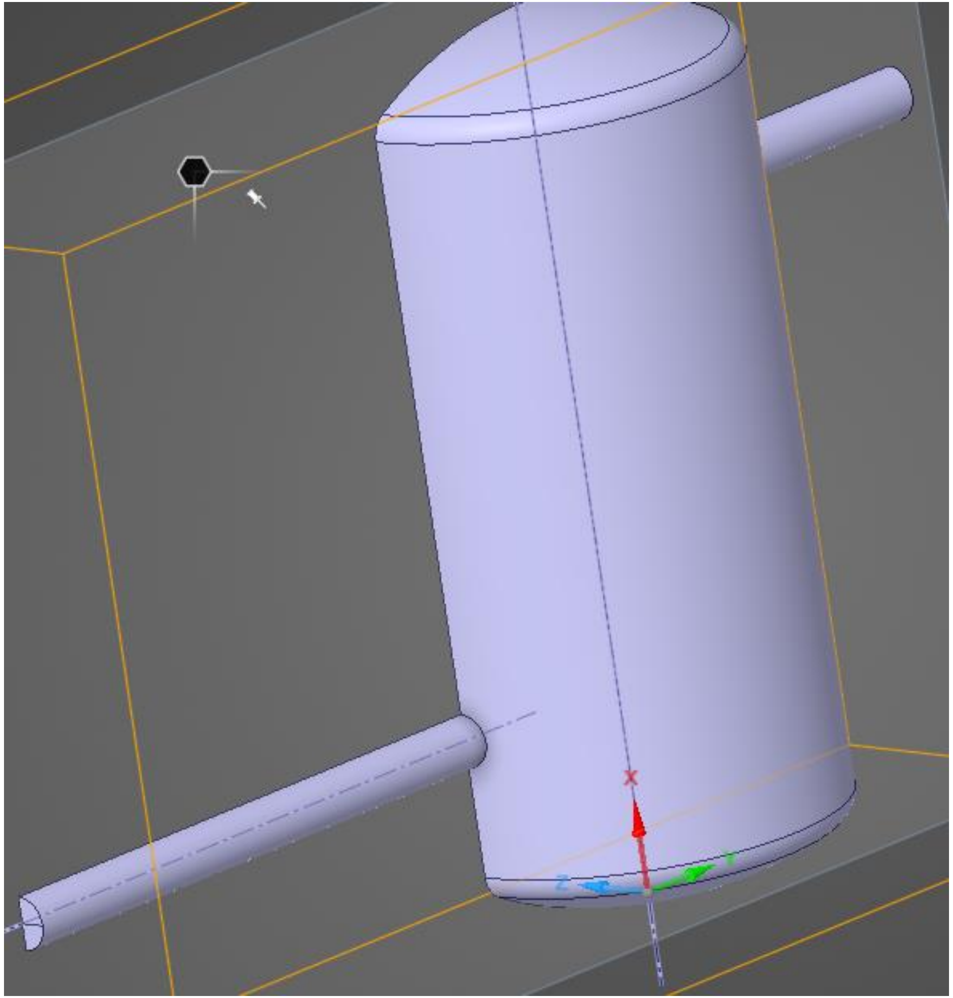


CO2-C4F10
Equimolar Mixture

Using a mixture introduces a series of complexities.
- The momentum transport equation is doubled (N components => N-1 sets of RANS)
- The energy equation is activated

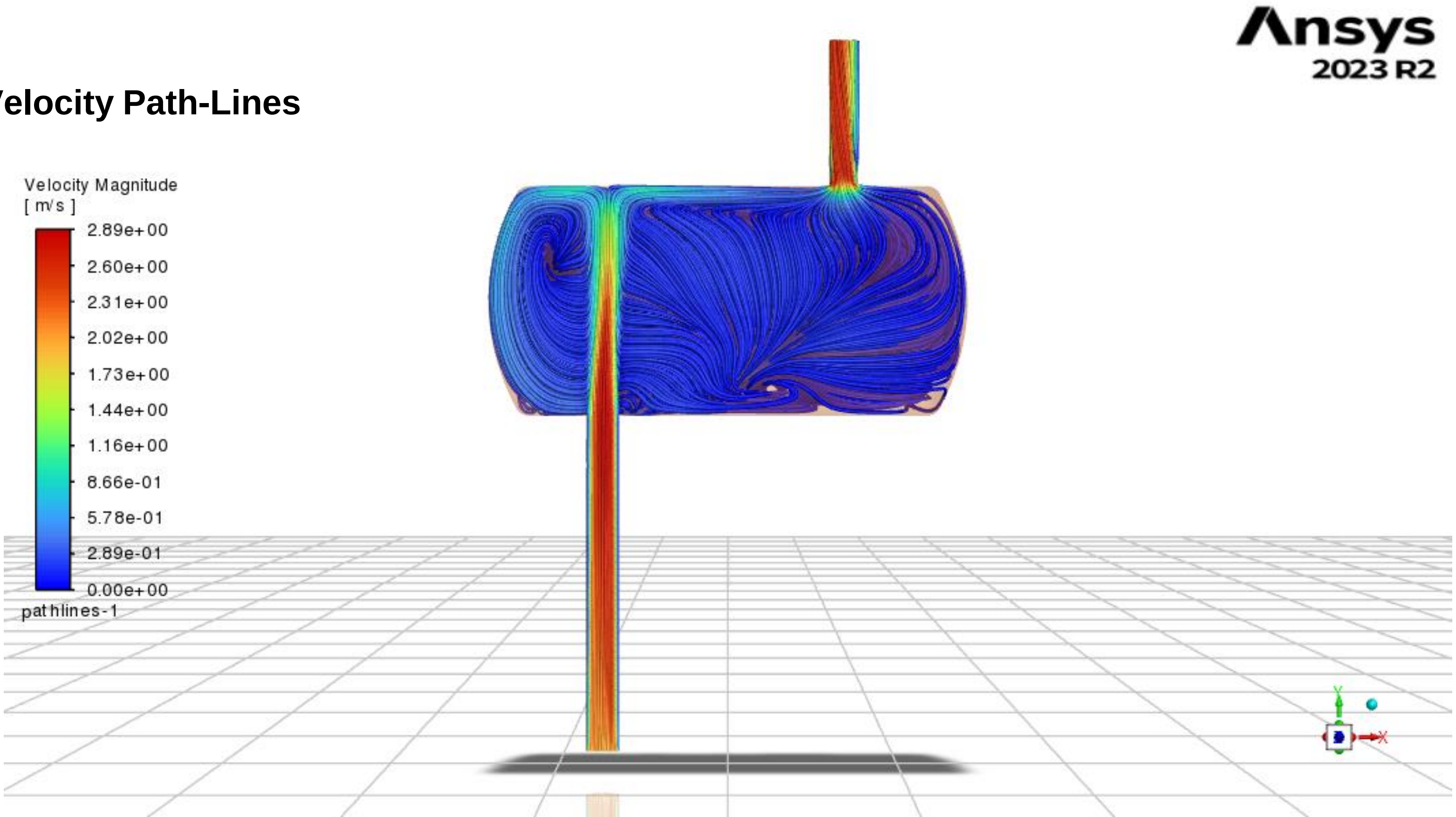
Moreover, a new inlet must be added as the two fluids are simulated to enter almost separately (non-premixed flow). This puts us in a worst-case scenario. As a fact, the two components are non-completely segregated when exiting the detector, thus a certain degree of mixing is already achieved.

To have a certain mixture before entering the tank the inlet pipe length has been doublet (1.5 m)



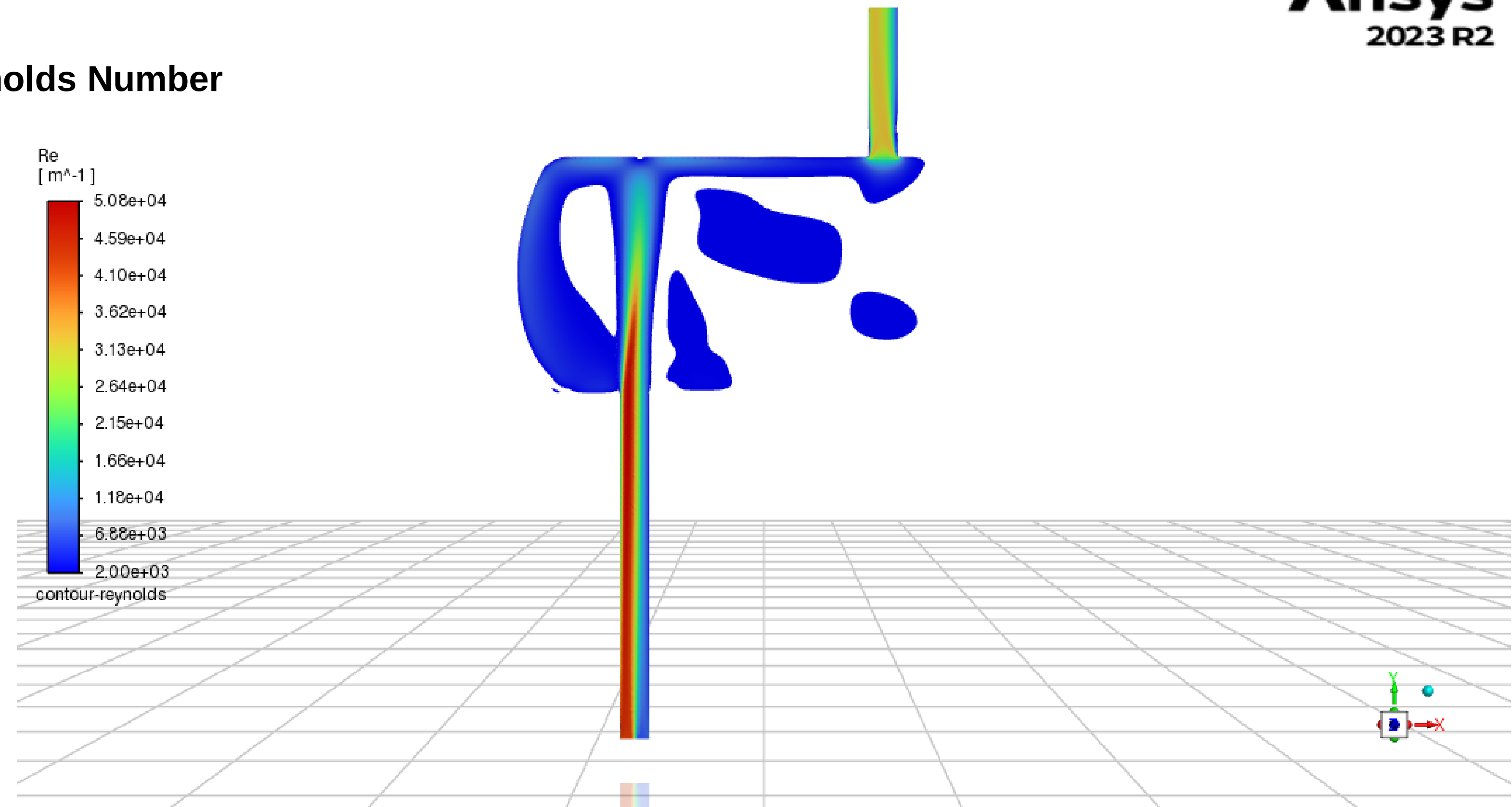
MIXTURE: STEADY-STATE

Fluid Velocity Path-Lines



MIXTURE: STEADY-STATE

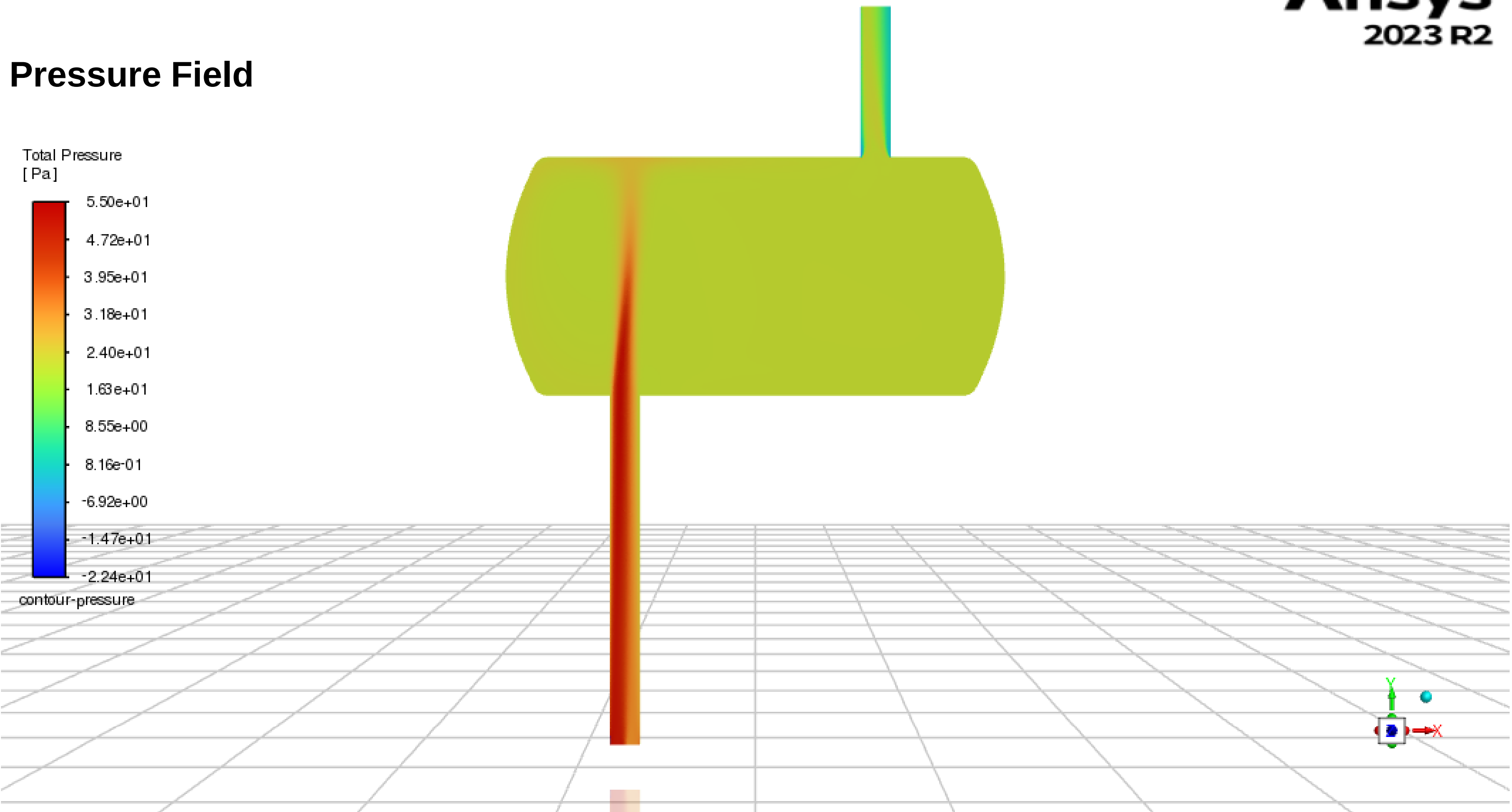
Reynolds Number



MIXTURE: STEADY-STATE

Ansys
2023 R2

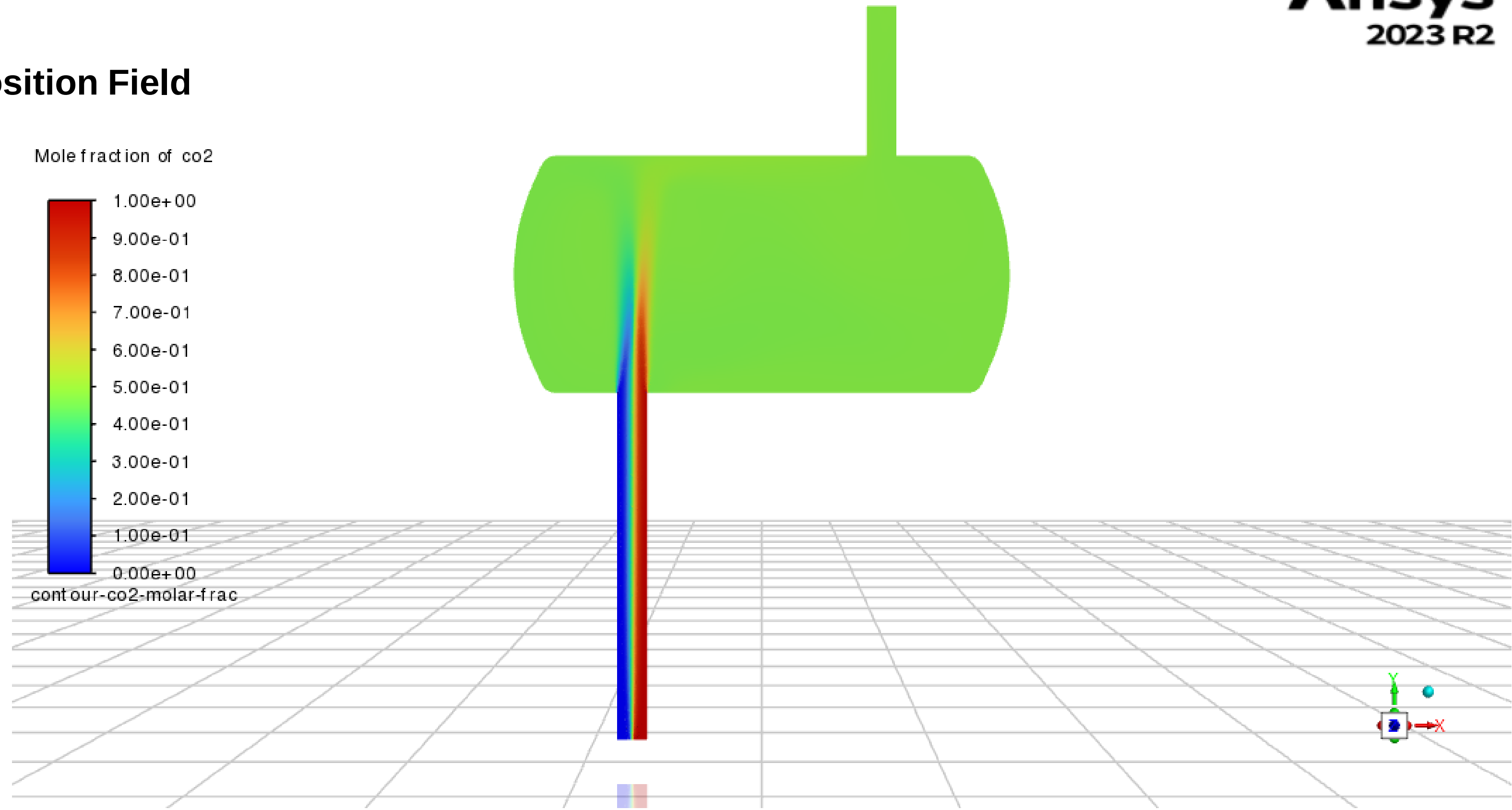
Total Pressure Field



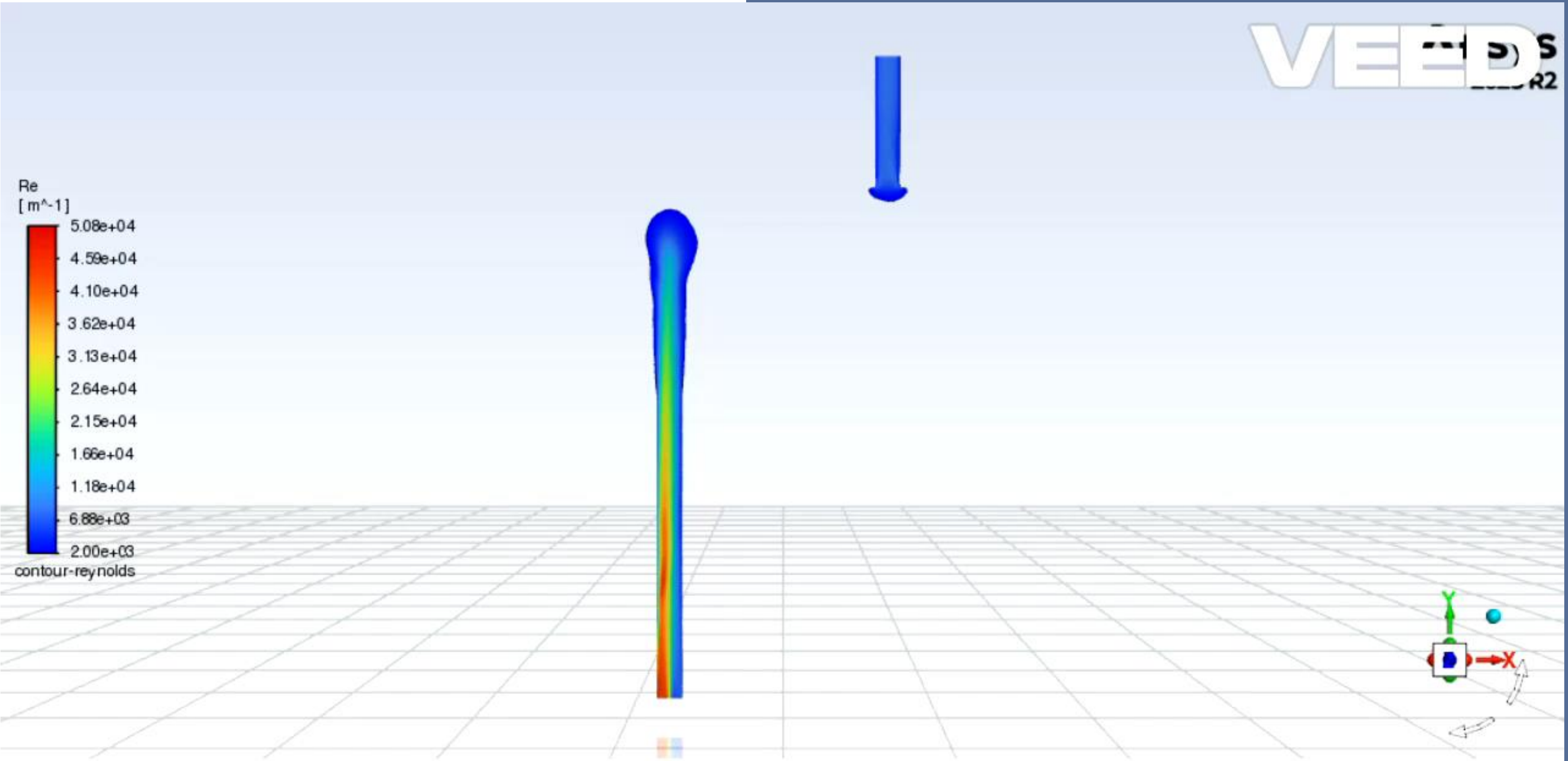
MIXTURE: STEADY-STATE

Ansys
2023 R2

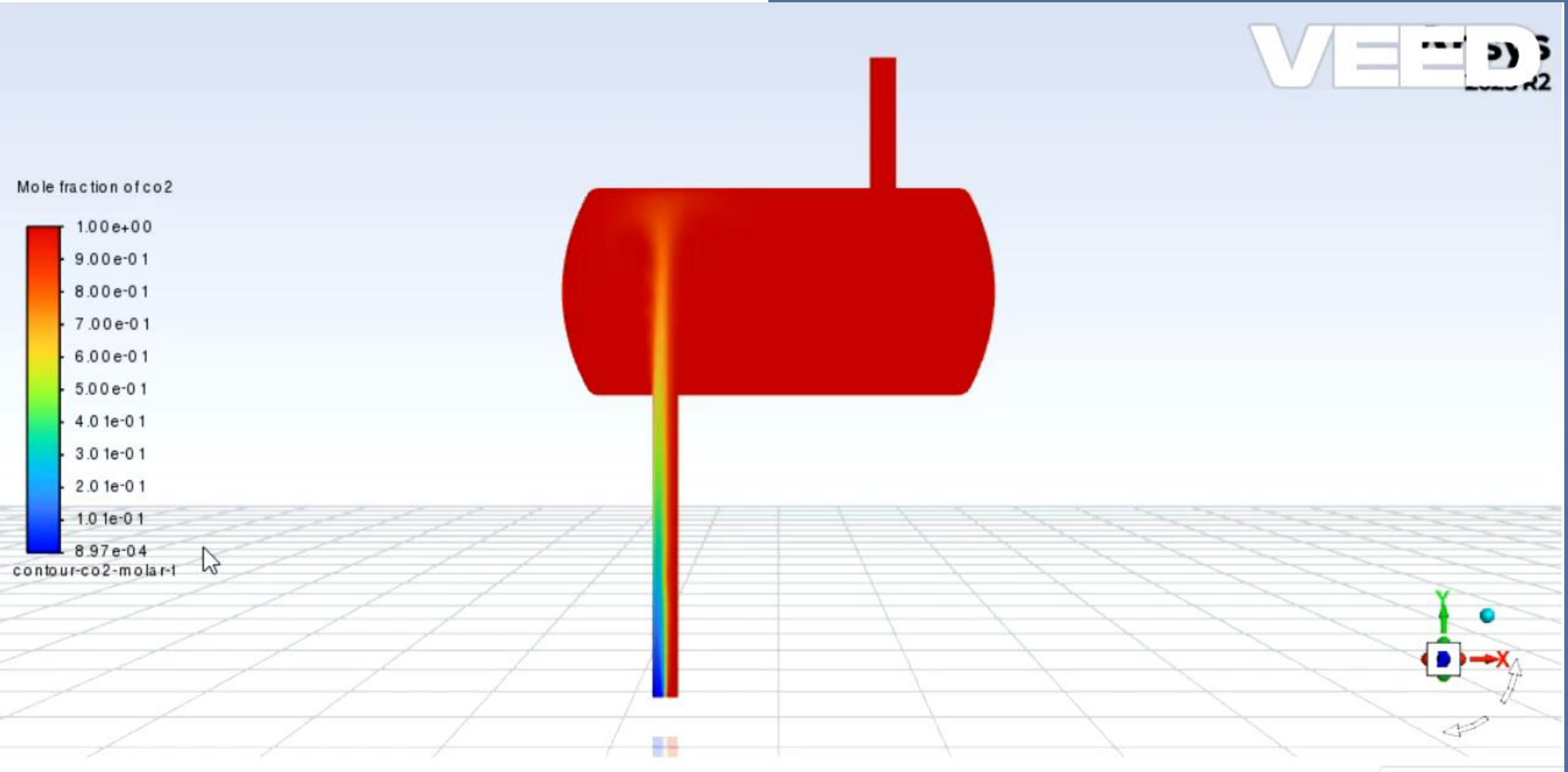
Composition Field



MIXTURE TRANSIENTS



MIXTURE TRANSIENTS



$$v = 2 \left[\frac{m}{s}\right]$$

$$S = \frac{\pi D^2}{4} * \left(\frac{1}{4}\right) = 4.41e-3\ [m^2]$$

$$V = 2 * v * S = 1.76e-2 \left[\frac{Nm^3}{s}\right]$$

Converting from [m^3/s] to [m^3/h]

$$V = 63 \left[\frac{Nm^3}{h}\right]^{**}$$

Conclusions

- The flowrate (V) to ensure a homogeneous composition in around 60 seconds is just above 60 Nm3/h
- The steady-state is reached

Next Steps

- Find the minimum flowrate at which a homogeneous steady-state is reached
- Increase the flowrate to guarantee homogeneity in a fixed time

** The flowrate is calculated for a mixture 50%CO2 - 50%C4F10