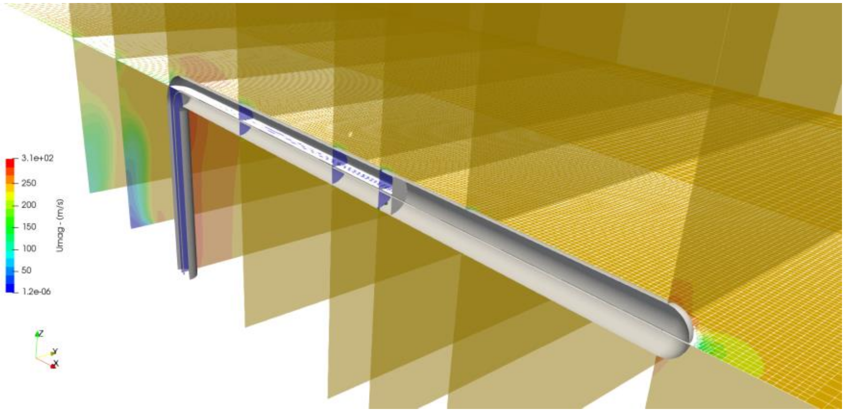
SIMULATION OF A PITOT TUBE WITH OPENFOAM®

Paulin FERRO
CFD Engineer at G-MET Technologies

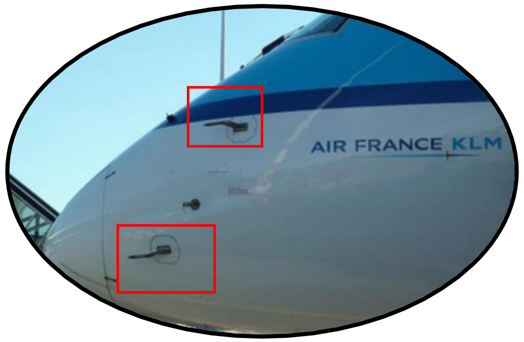


Open-source environnement
Geometry : Salome®
Meshing : blockMesh and snappyHexMesh
OpenFOAM® Solver : simpleFoam and sonicFoam
Post-processing : Paraview® and Python

❑ GENERAL CONSIDERATIONS AND PITOT TUBE 3D GEOMETRY

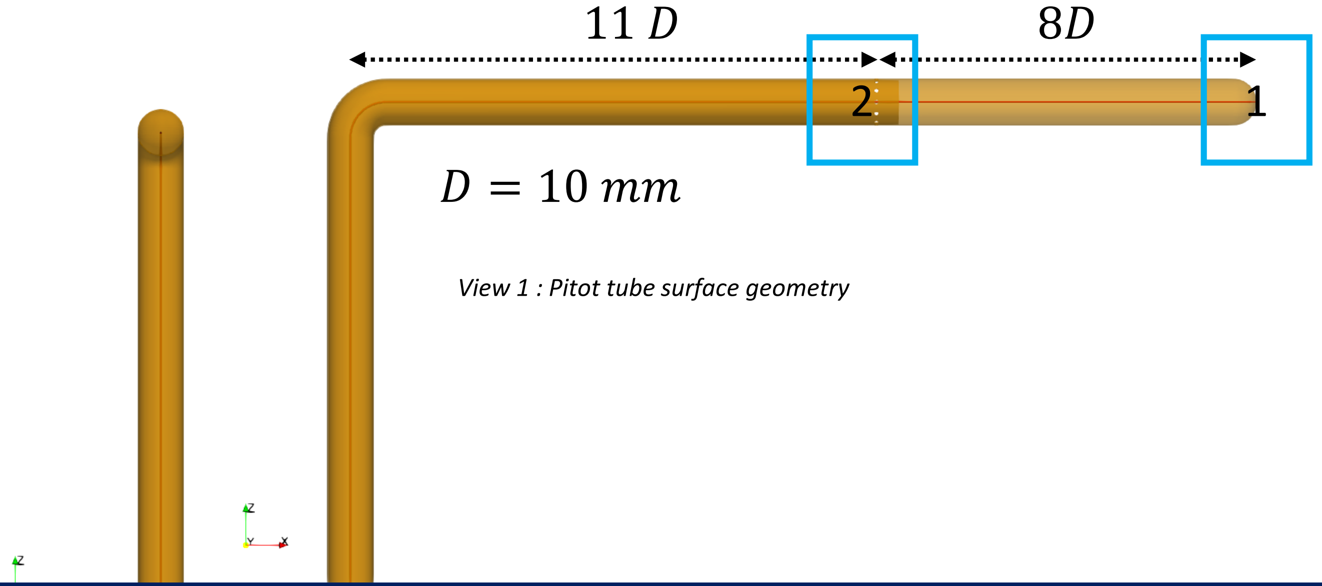
Aim of the study : Compare theoretical and CFD pressure difference ΔP_{THE} vs ΔP_{CFD}.
5 configurations : 0.1 ; 1 ; 67 ; 100 and 240 m/s
Free geometry, fluid properties and physics

Pitot tube was invented by Henri PITOT in 1732 (French engineer)
Improved design by Prandtl and Darcy in the mid-19th century
Flow device used to measure fluid flow velocity (widely used for ship, aircraft and other applications)

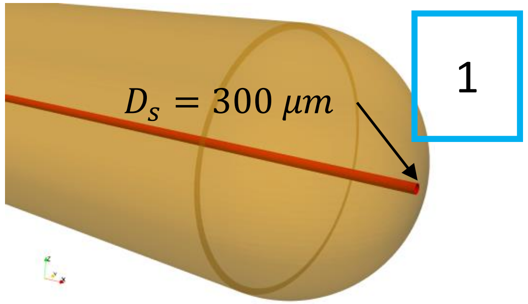


Example of Aircraft Pitot tubes for speed measurement
Source : Toulouse University

- The surface geometry of the device is generated using **SALOME®**
- The static pressure port is located 8D behind the pitot noze (D being the Pitot's tube diameter)
- The total pressure port consists of a tube of diameter 300 μm
- The static port consists of an internal cavity linked to outside with 12 holes of diameter 500 μm

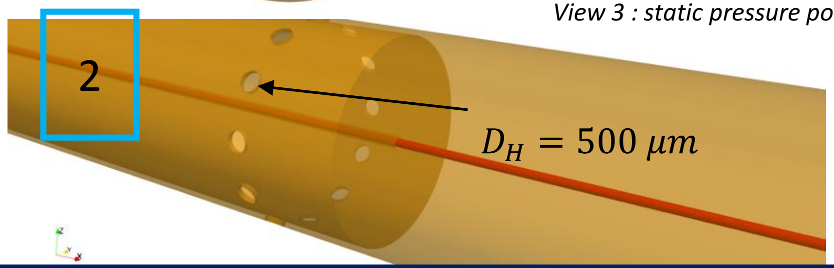


View 1 : Pitot tube surface geometry



View 2 : Total pressure port

View 3 : static pressure port, 12 holes



PHYSICS AND NUMERICAL SETTINGS

- At low **Mach number (Ma < 0.3)** CFD results can be compared to either $p_t - p_s = \frac{1}{2}\rho v^2$ or $\frac{p_t}{p_s} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$

- Using **air** as working fluid leads to a **Mach number** of approximatively 0.64 for V = 240 m/s. (*Assuming ρ = 1 kg/m³ and P = 1 bar: c = 376.6 m/s*). Thus a **compressible solver** of OpenFOAM® should be used to take into account compressibility effects. In this configuration the CFD pressure ratio has to be compared to $\frac{p_t}{p_s} = \left(1 + \frac{\gamma-1}{2}M^2\right)^{\frac{\gamma}{\gamma-1}}$

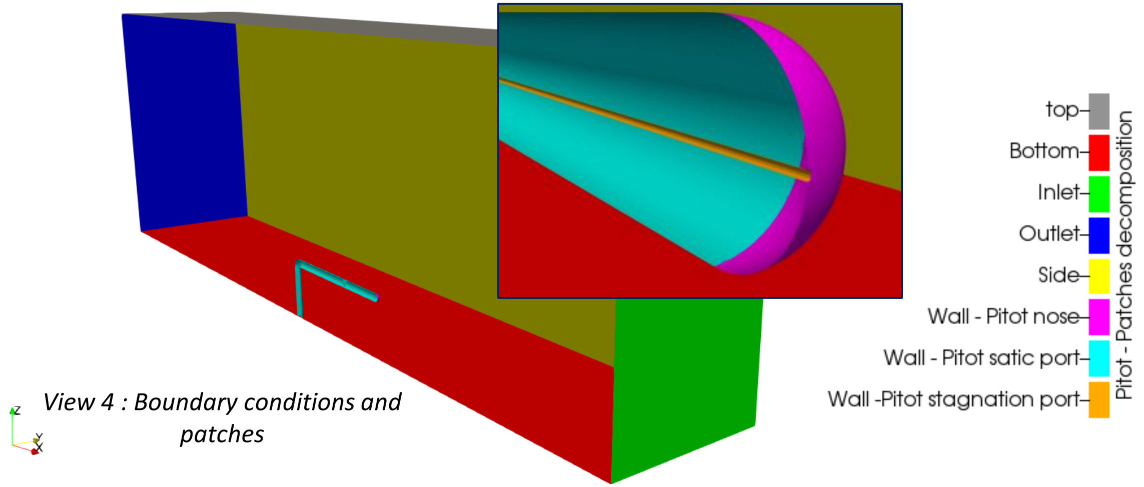
- The following table resume the main **physical characteristics** for each case :

Free stream velocity – m/s	Reynolds number (L = 0.2 m)	Turbulence	Mach number *ρ = 1, c = 376.6 m/s*	Compressibilty effect
0.1	1280	OFF (Laminar)	$2.6\ 10^{-4}$	OFF - simpleFoam
1	12820	On (k-omega **SST**)	$2.6\ 10^{-3}$	OFF - simpleFoam
67	$8.59\ 10^{5}$	On (k-omega **SST**)	0.18	OFF - simpleFoam
100	$1.28\ 10^{6}$	On (k-omega **SST**)	0.27	OFF - simpleFoam
240	$3.08\ 10^{6}$	On (k-omega **SST**)	0.64	ON – sonicFoam OFF – simpleFoam

Both simpleFoam and sonicFoam will be run for this speed

- Numerical schemes are second order for **div(phi,U)** (Gauss linearUpwindV grad(U) with limiter on grad(U) : cellLimited Gauss linear 1) and first order for other quantities such as turbulence (Gauss upwind)

❑ BOUNDARY CONDITIONS AND 3D MESHING WITH SNAPPYHEXMESH (SHM)

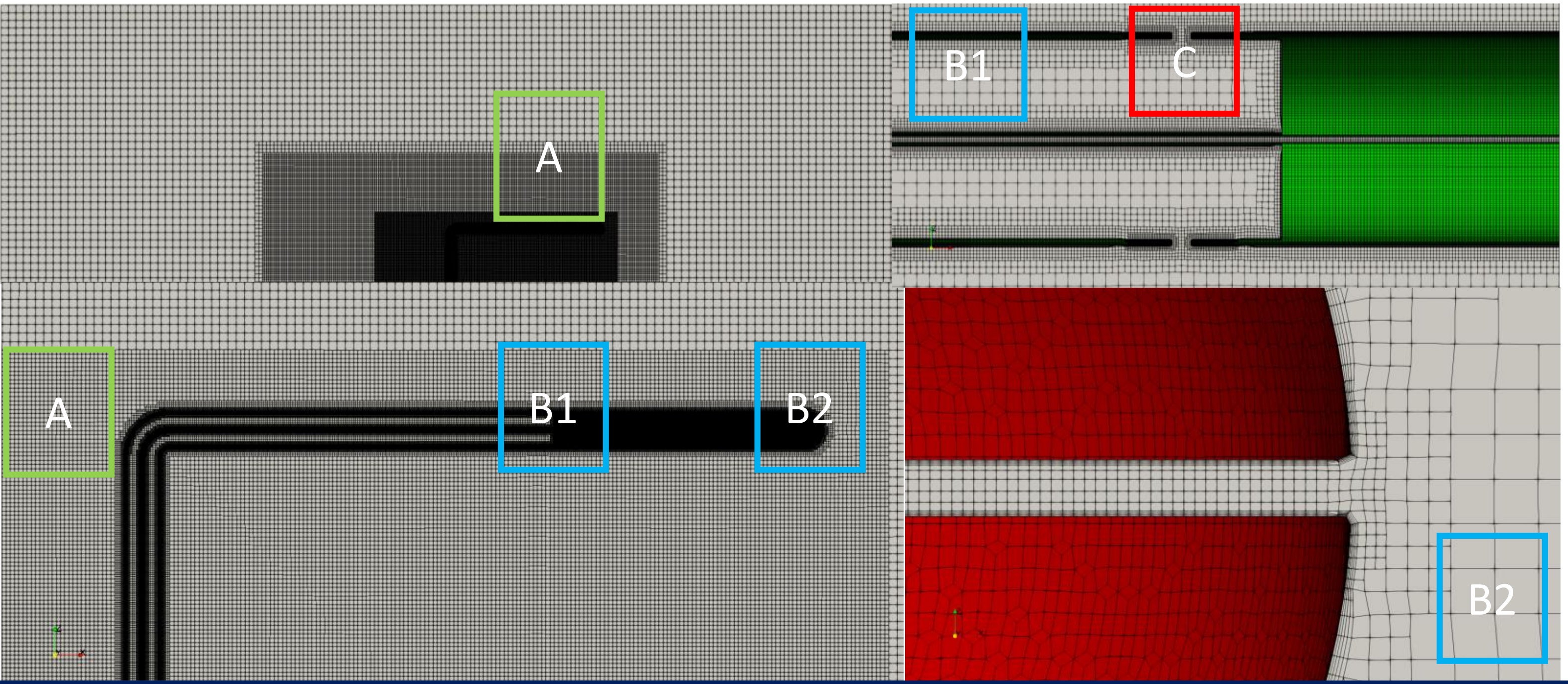


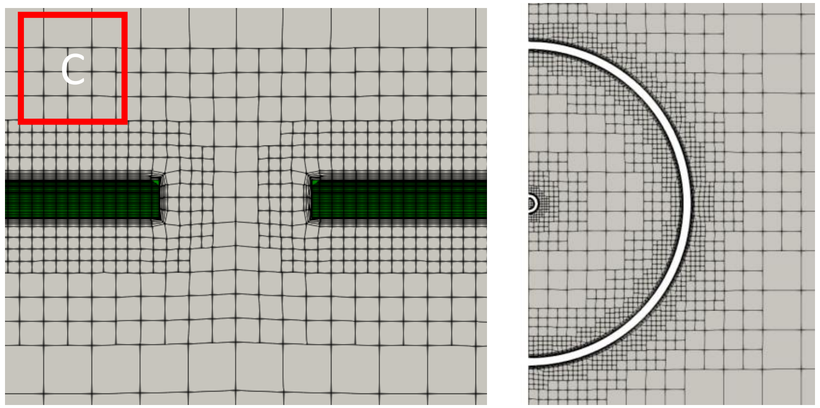
View 4 : Boundary conditions and patches

- At inlet : velocity *fixedValue* ; *zeroGradient* condition for pressure. turbulence quantities are assessed using following expressions:

$$I_t = 0.01 \qquad k = 0.5(UI_t)^2 \qquad \omega = \frac{k^{\frac{3}{2}}}{l_t} \qquad l_t = 20\ cm$$

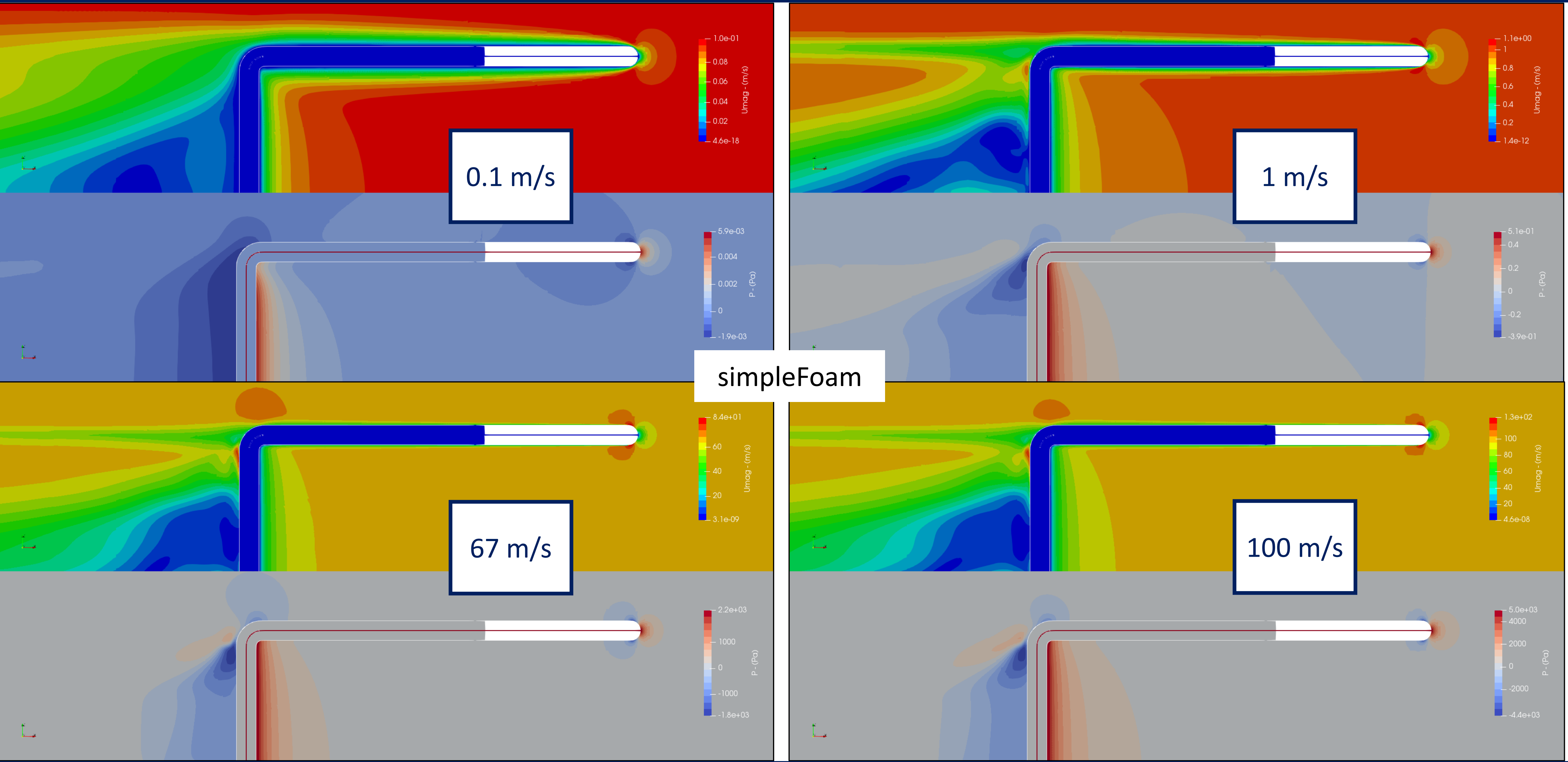
- Side, bottom and top patches are assumed to behave as "*slip wall*"
- Outlet : *fixedValue* for pressure and *zeroGradient* for other variables





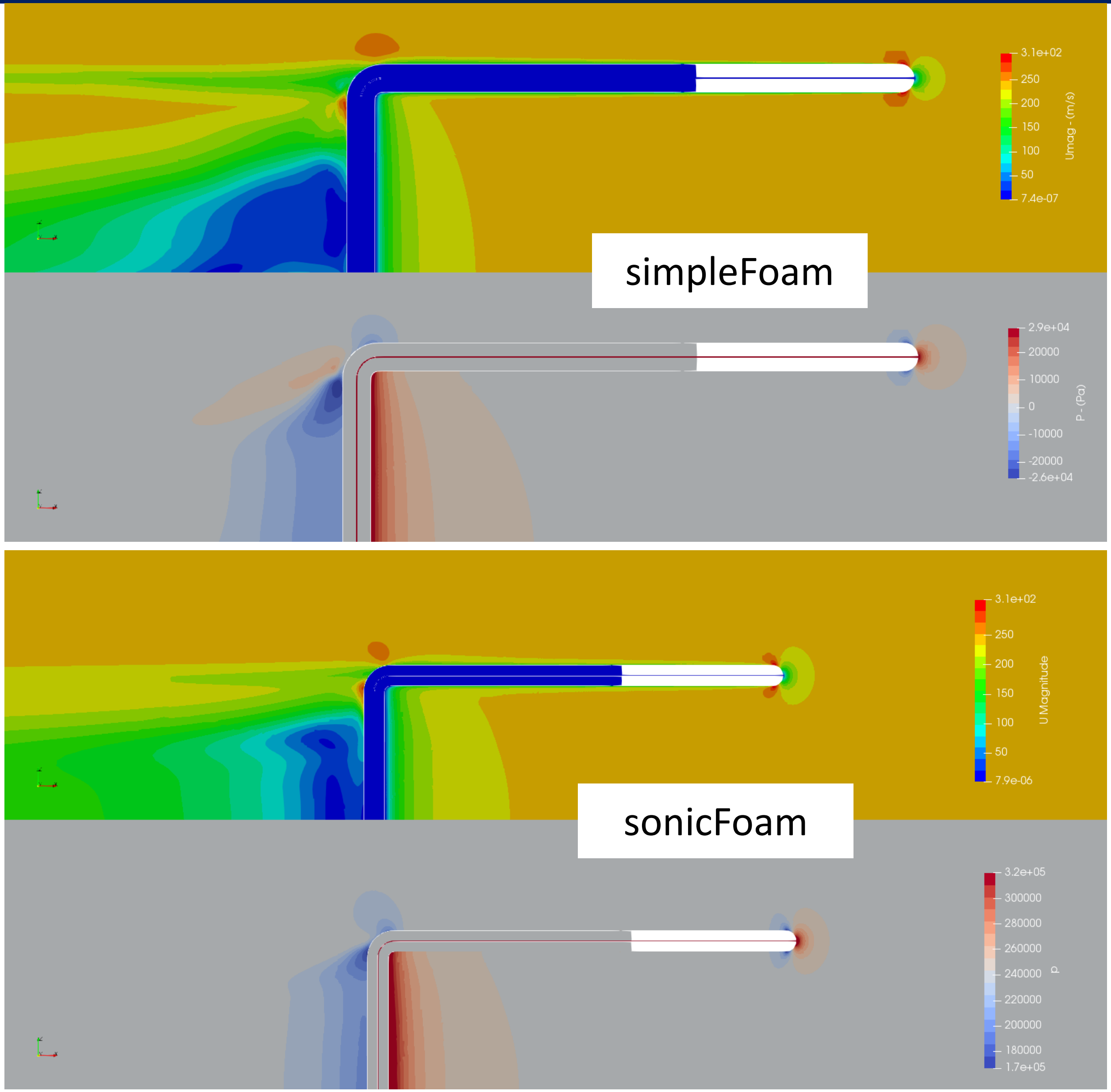
- The surrounding box and background mesh are generated with **blockMesh** utility
- The mesh consists of approximately **6.3 M** hexahedral and prism layer cells and was generated with **snappyHexMesh** (semi-automated mesher).
- Insertion of **5 boundary layer elements** (cover layer of at least **99 %**)
- **Small holes** are challenging with SHM and handled locally using the "**thin gap**" option of SHM.

RESULTS – PRESSURE AND VELOCITY FIELDS

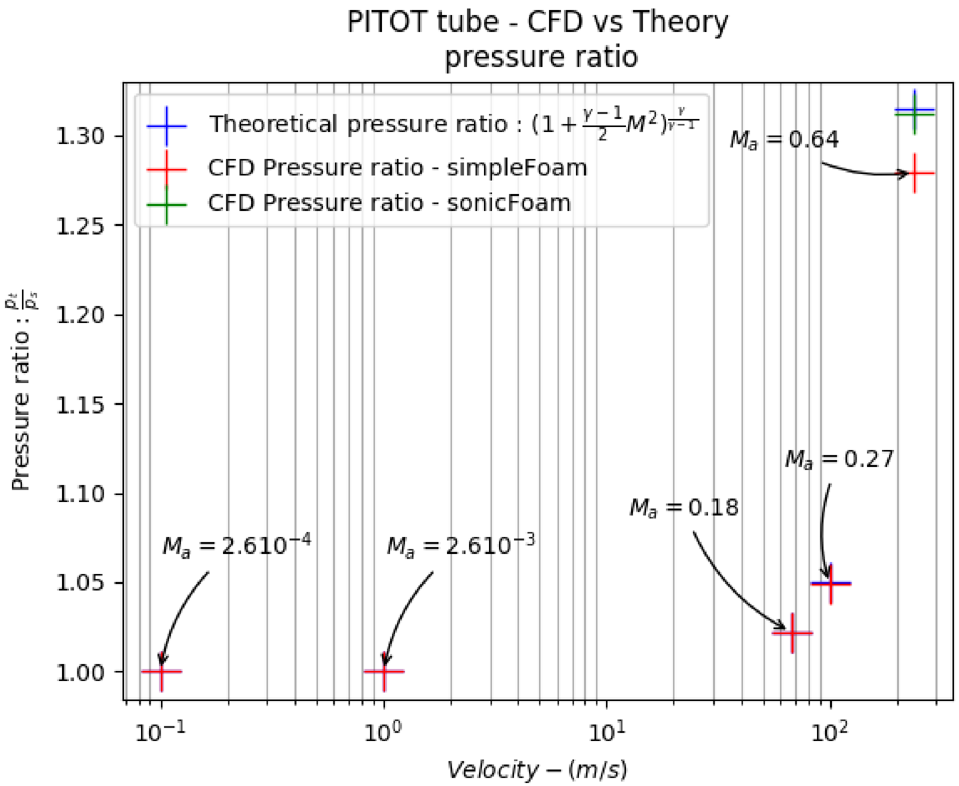


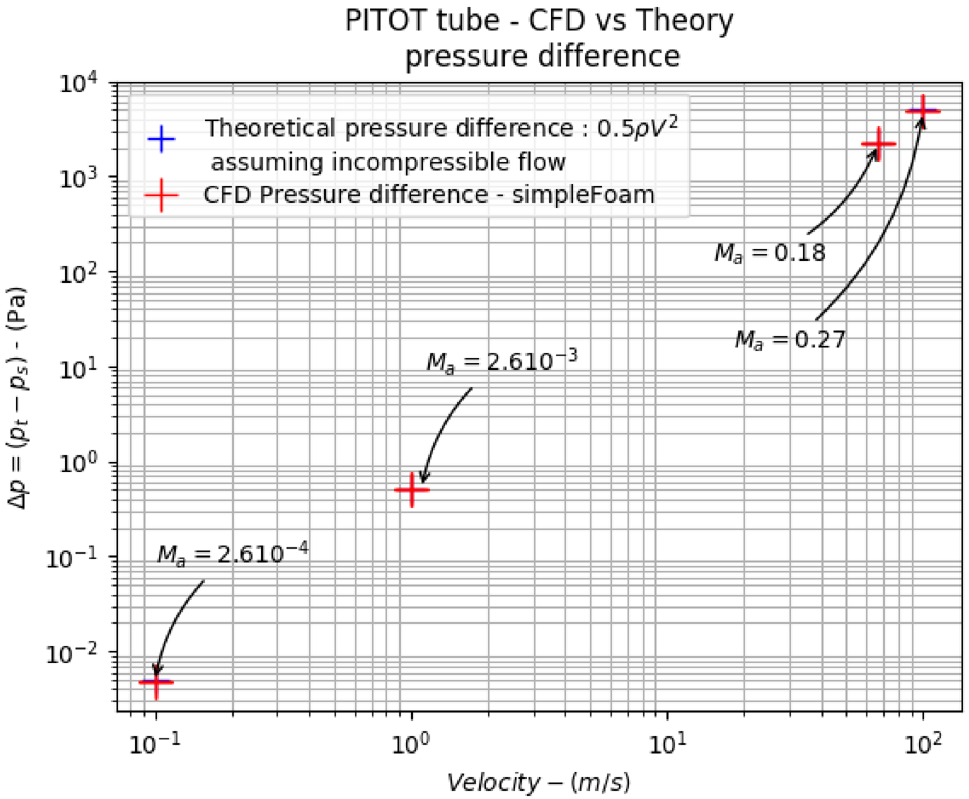
❑ RESULTS – PRESSURE AND VELOCITY FIELD

240 m/s



❑ RESULTS – CFD vs THEORY - COMPARISON





- Pressure difference is evaluated at the bottom of static and total pressure ports and compared to theoretical values.
- Results obtained with incompressible solver *simpleFoam* are in agreement with analytical pressure ratio if Ma < 0.3.
- At Ma 0.67 (V = 240 m/s) compressibility has to be taken into account. As expected results obtained with simpleFoam don't match theoretical pressure ratio
- Error with theory is significantly reduced using compressible solver sonicFoam.

The tutorial case is available at www.cfd-training.com !