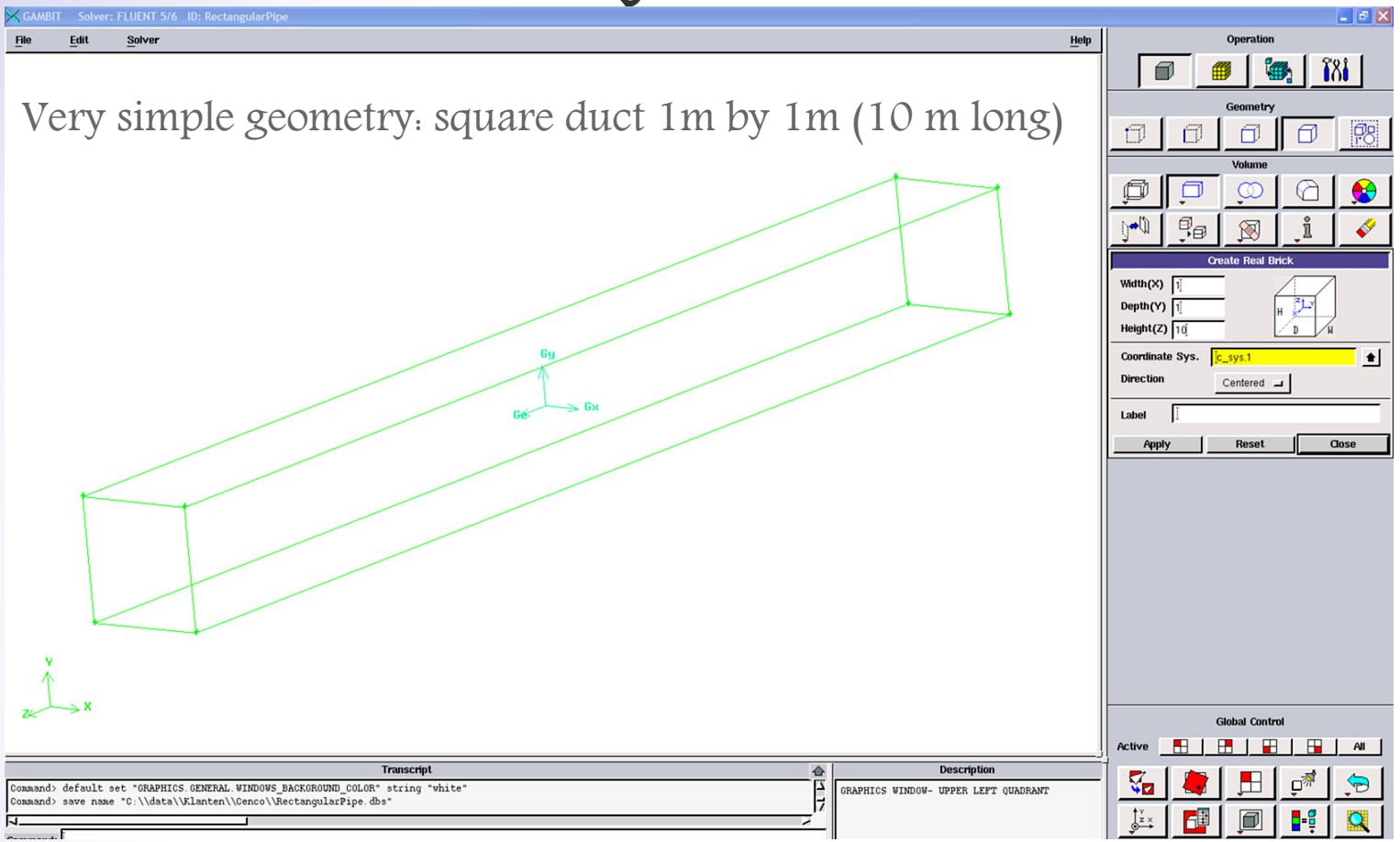
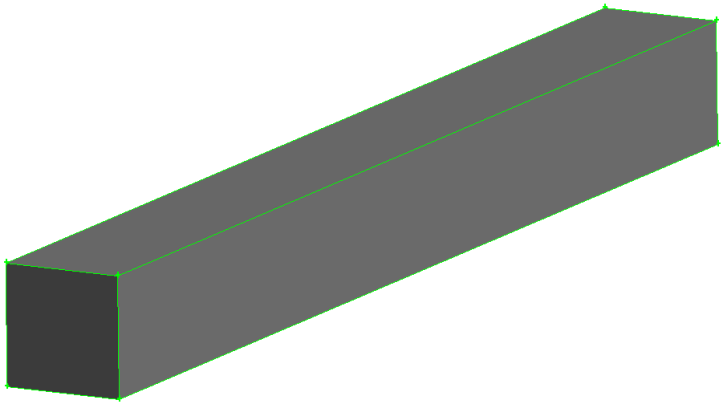
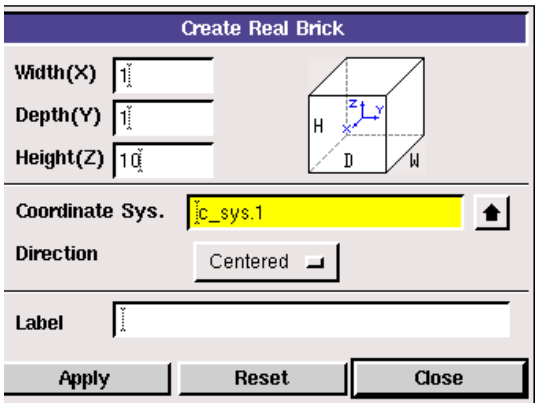
Aeolus
CFD Consultancy
Flow Simulations

Few words about meshing...

Geometry Overview



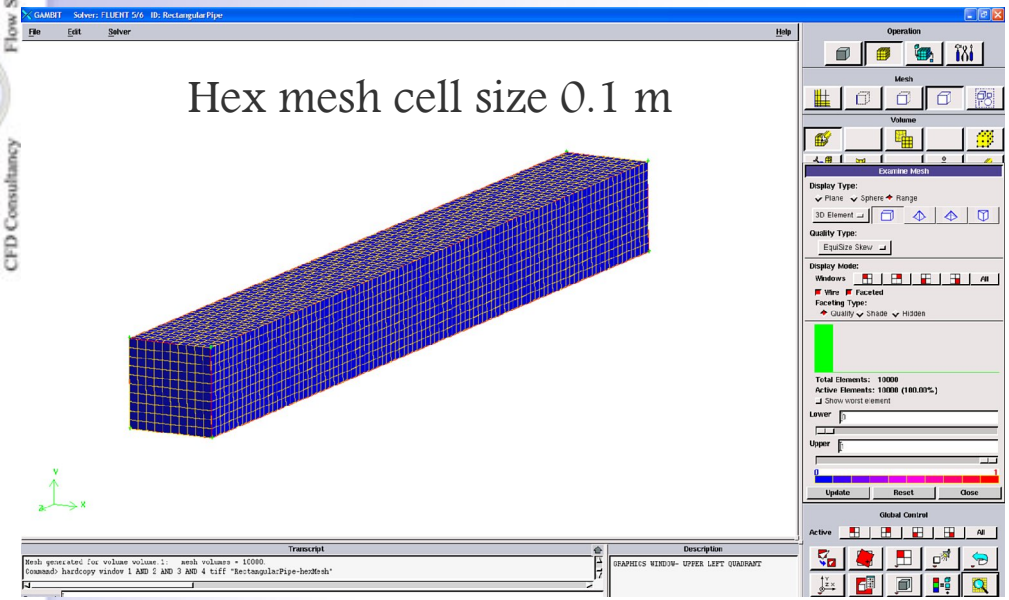




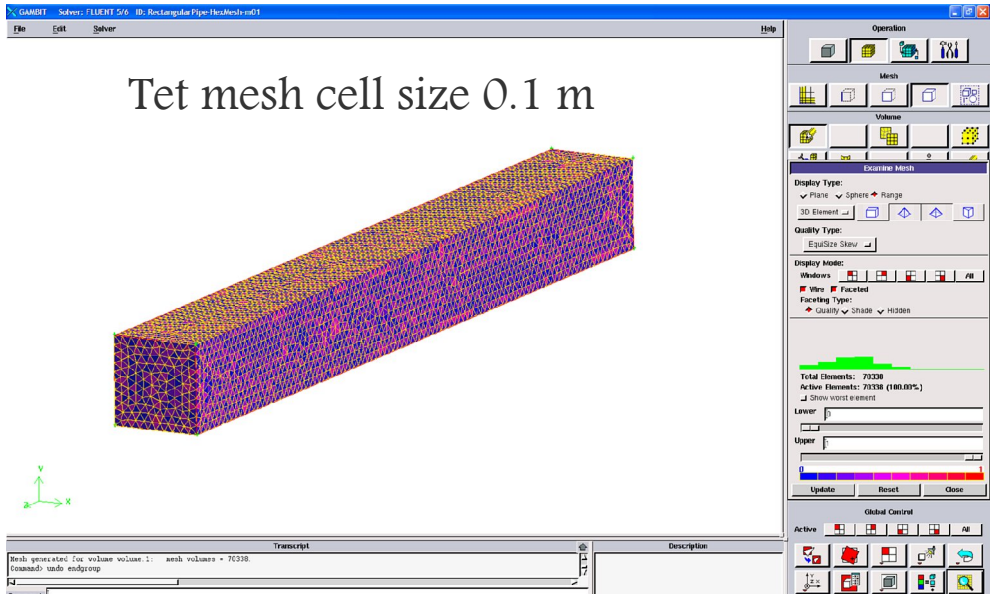
MeshingTypes,

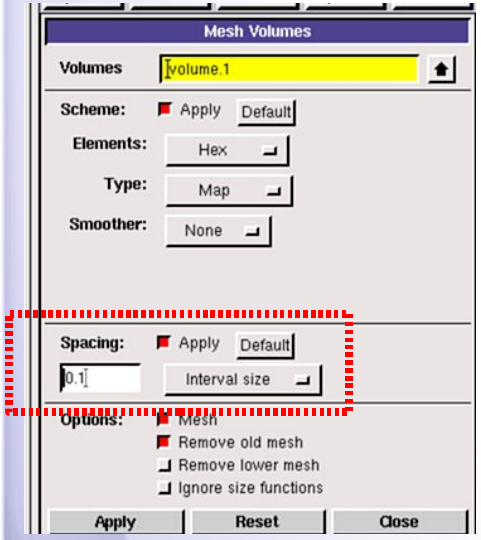
Mesh Overview

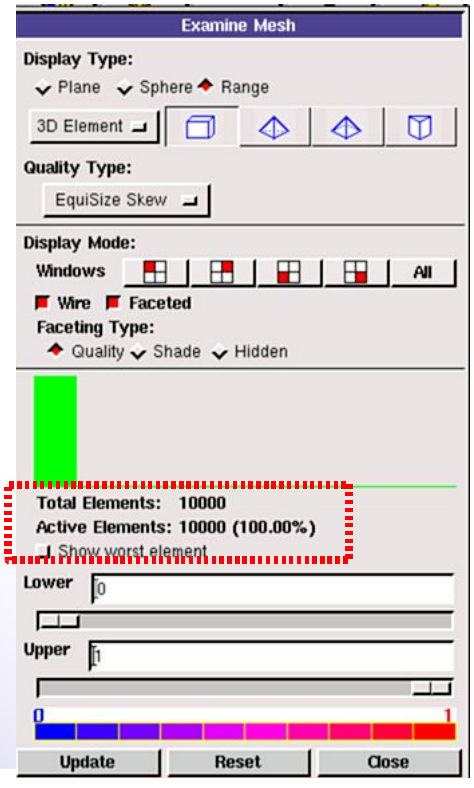
Hex mesh cell size 0.1 m



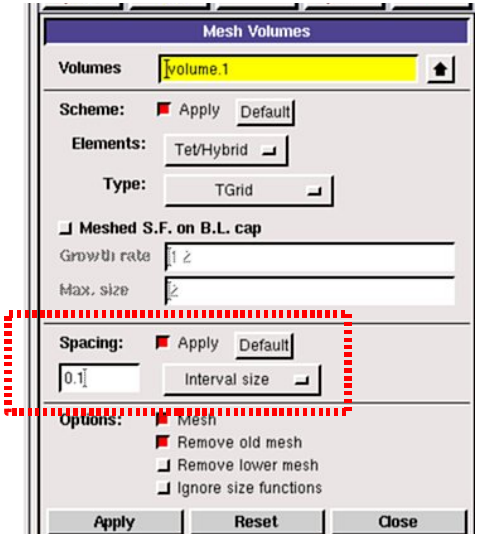
Tet mesh cell size 0.1 m

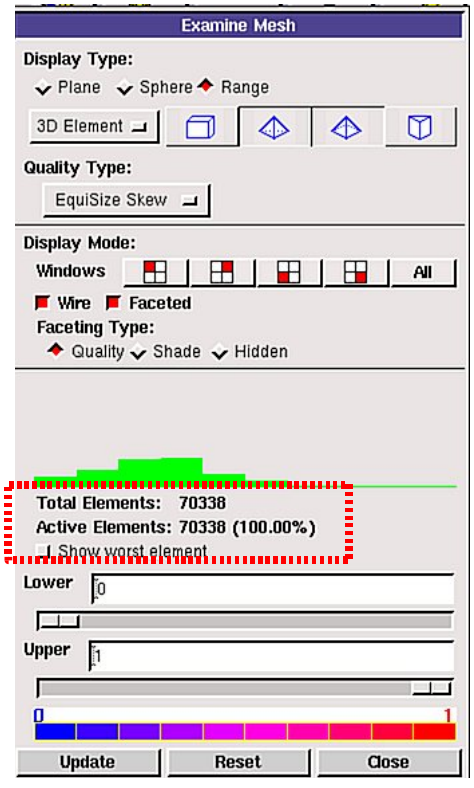






10,000 hex cells

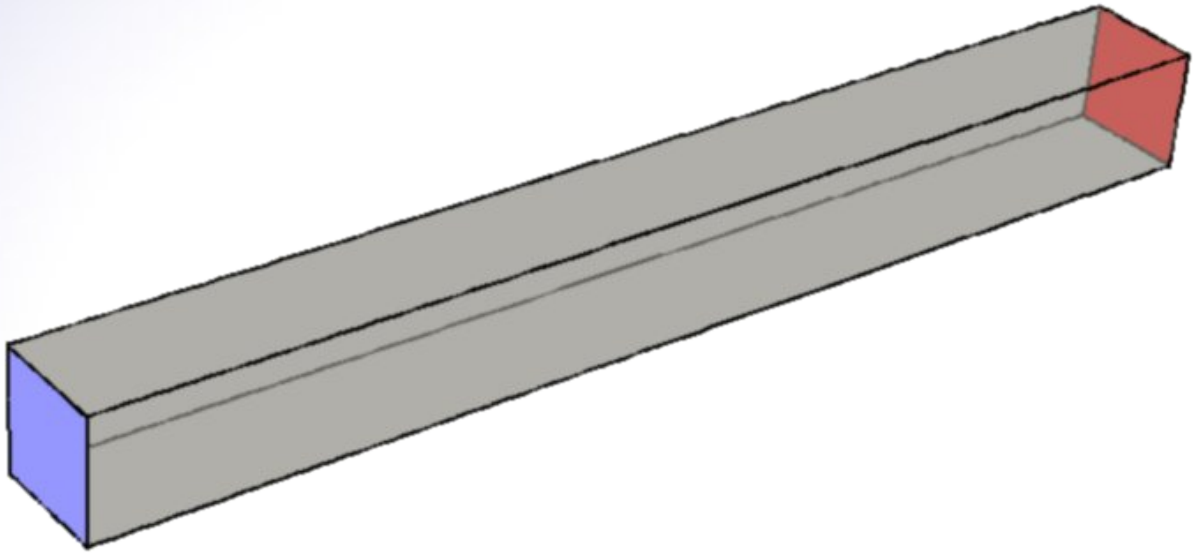




70,000 tet cells

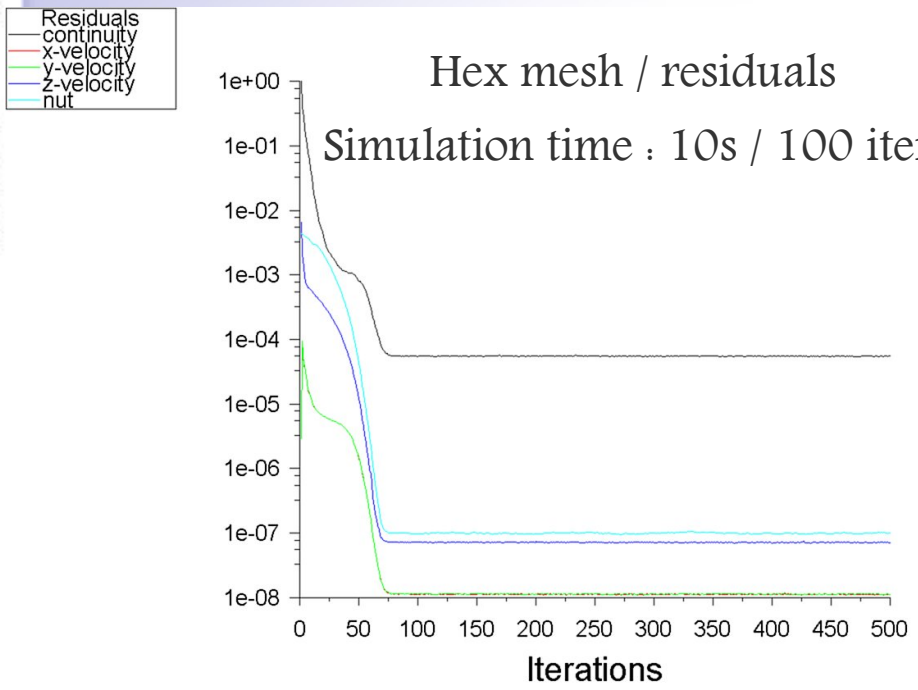
MeshingTypes,

Simulation Settings



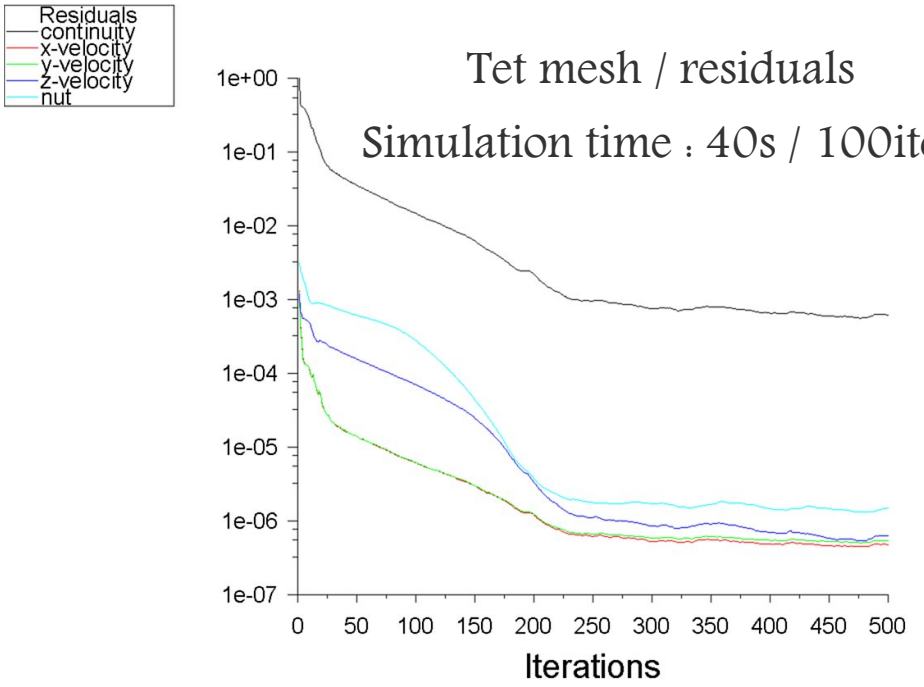
- Velocity inlet : 5 m/s
- Pressure outlet : 1 atm.
- Steady state simulation
- One equation turbulent model
- First order accuracy results

Convergence Behavior



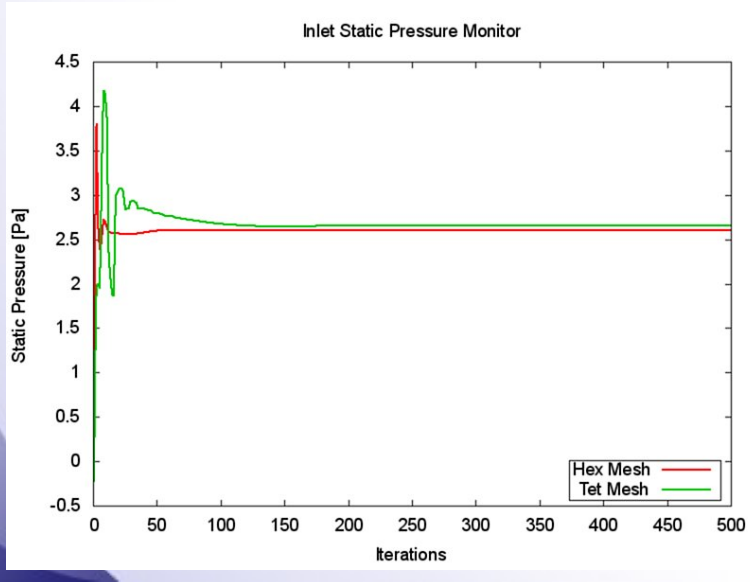
Hex mesh / residuals

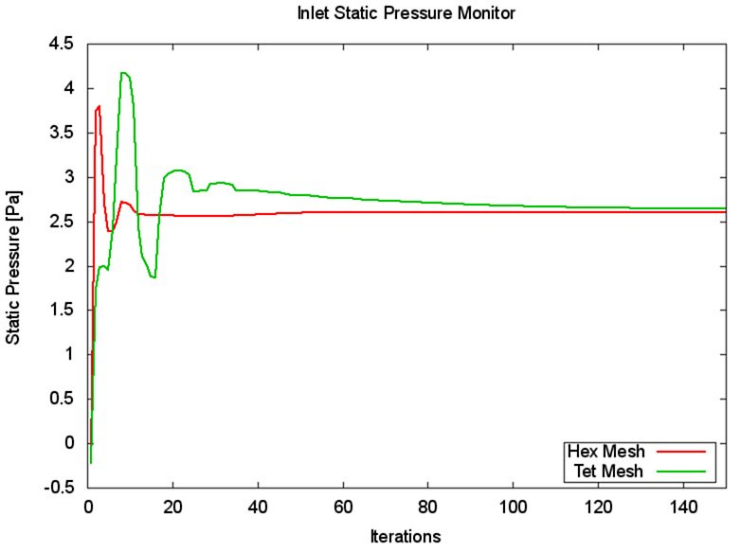
Simulation time : 10s / 100 iter.



Tet mesh / residuals

Simulation time : 40s / 100iter

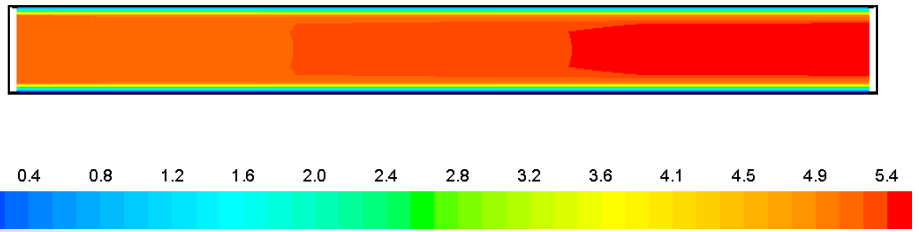




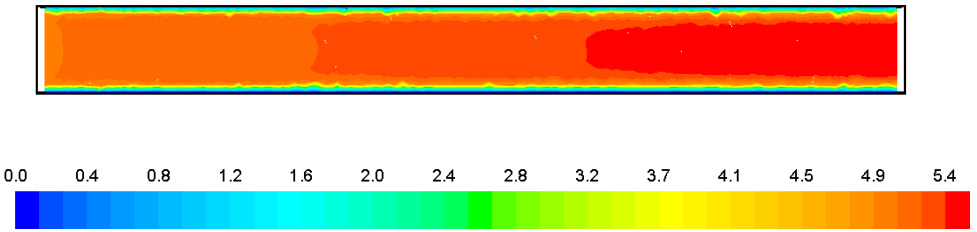
MeshingTypes,

Results

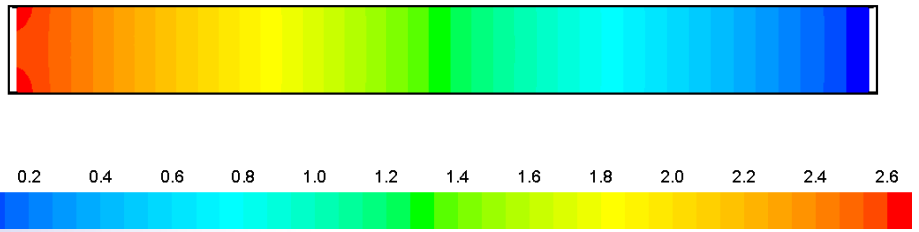
Hex mesh / velocity magnitude [m/s]



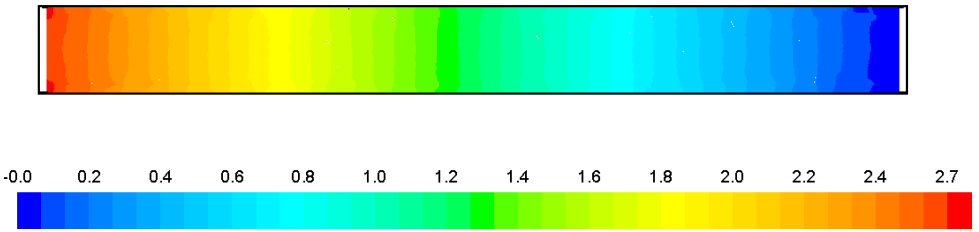
Tet mesh / velocity magnitude [m/s]



Hex mesh / static pressure [Pa]



Tet mesh / static pressure [Pa]



Hex mesh

inlet static pressure average = 2.601 [Pa]

Tet mesh

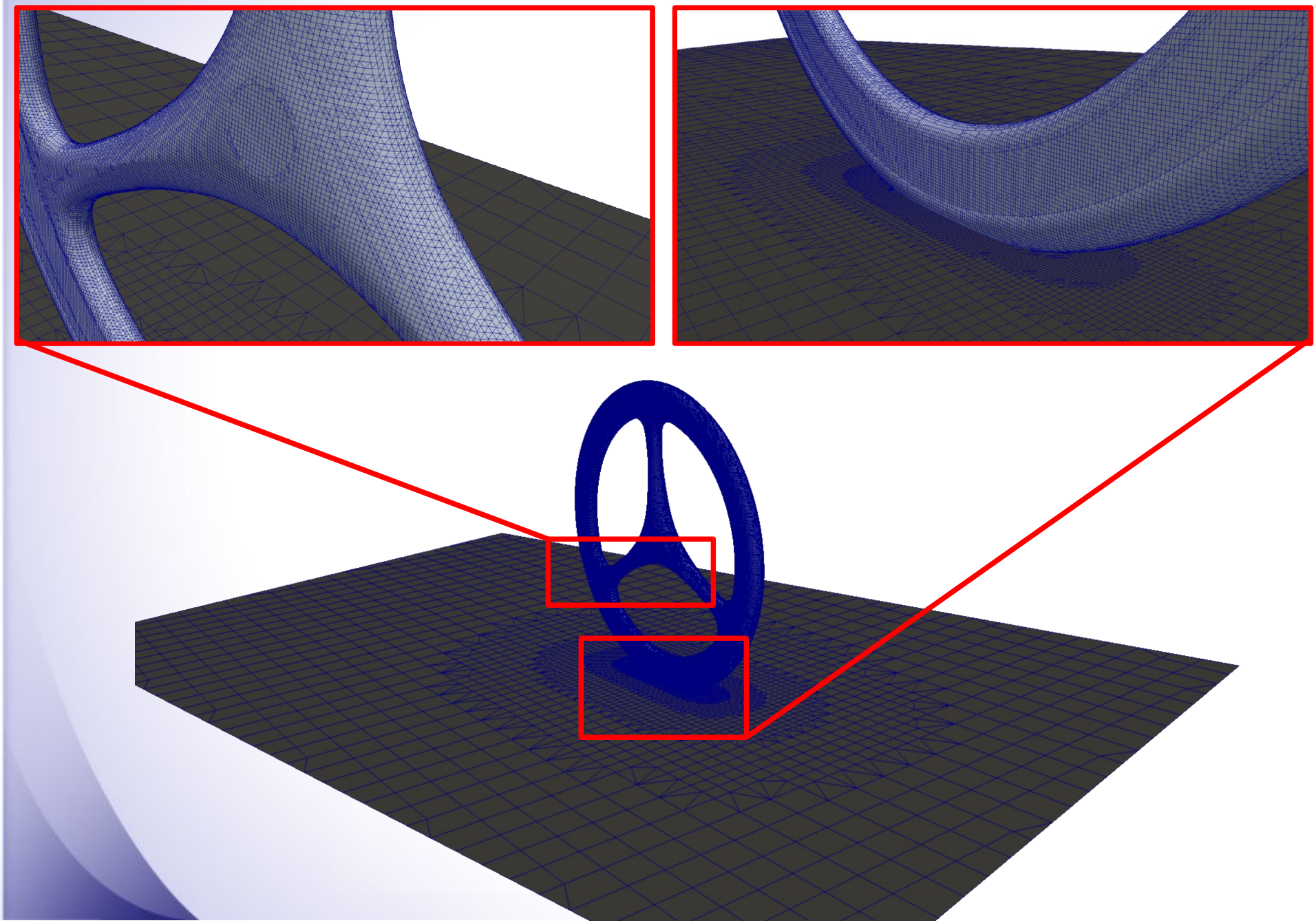
inlet static pressure average = 2.659[Pa]

Aeolus CFD Consultancy Flow Simulations

Conclusions:

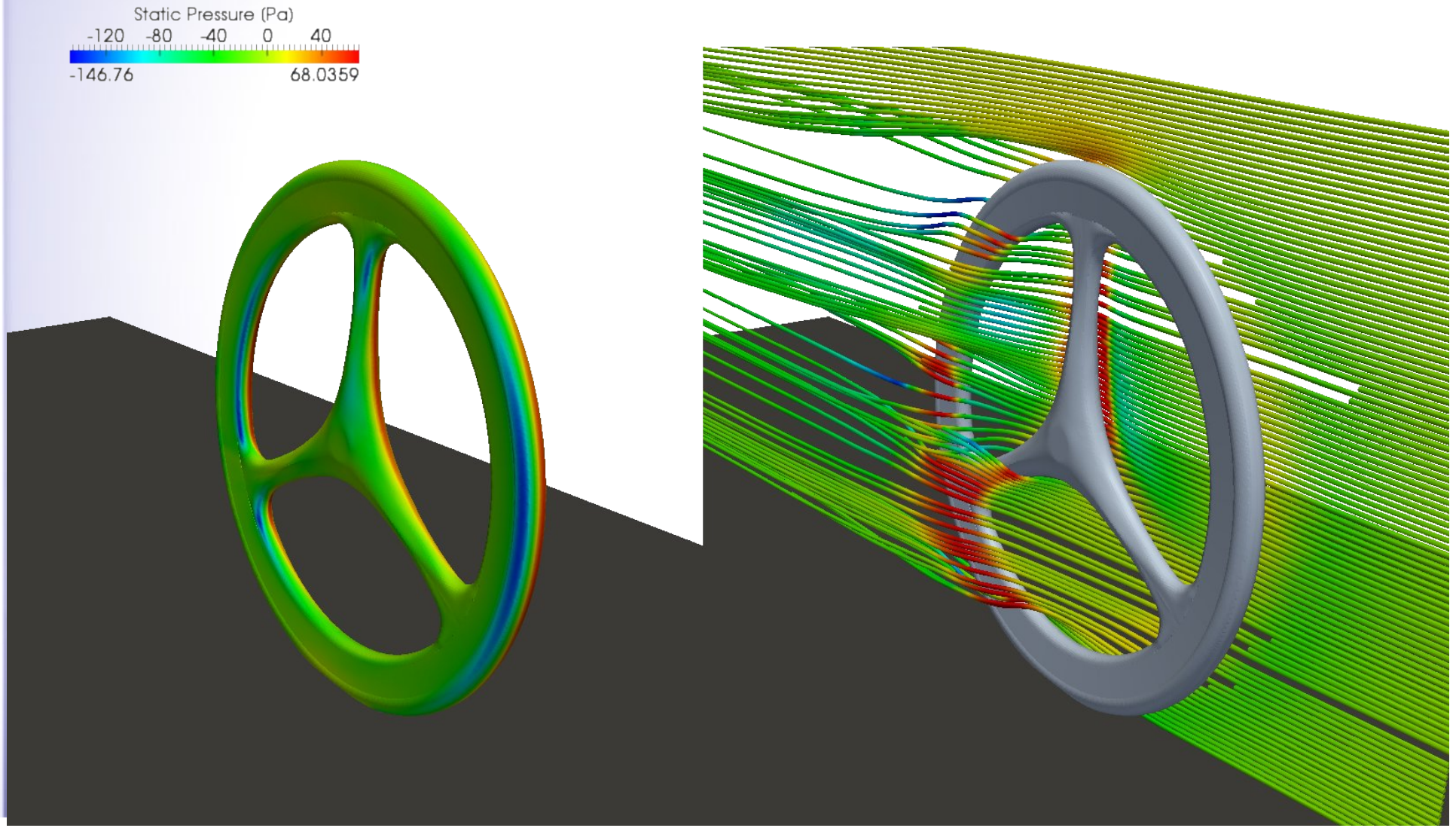
- For the same volume, and with the same grid spacing, the total cell count varies between 1 and 7 depending on the cell shape.
- For simple geometry, the use of hexahedral cells helps to:
  - Reduce the total cell count,
  - Reduce the simulation time (4 times faster),
  - Speed up the convergence (less iterations to reach convergence).
- For complex geometry, one can take advantage of the use of tetrahedral cells... or …
- One can take advantage of the cutcell method (available in Tgrid from ANSYS, CCM+ from CD-Adapco, OpenFoam ...and more) which fills the entire domain with cartesian hexahedral cells (thus reduces the total cell count) and cuts these hex cells closed to boundaries (to accurately capture the surface shape).

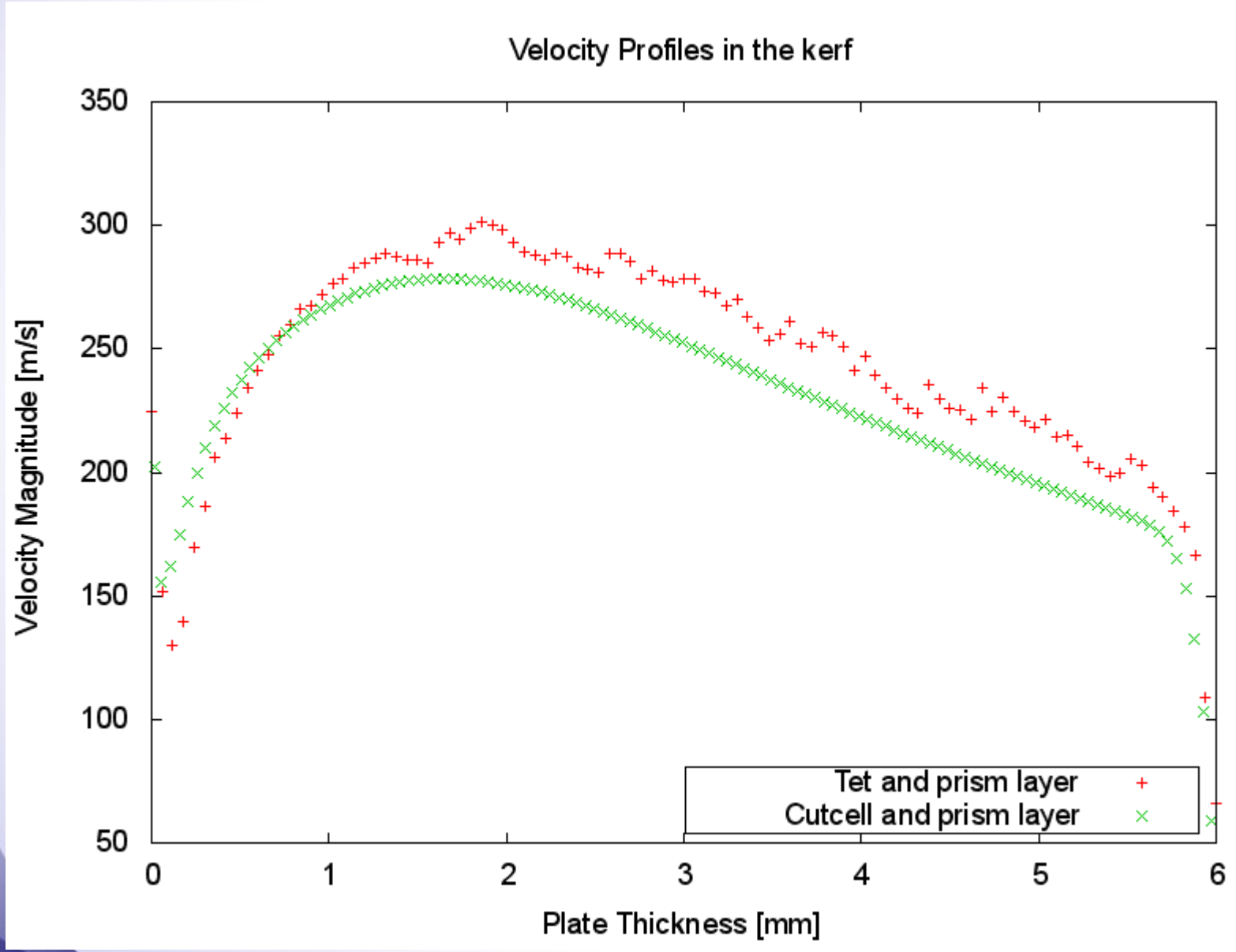
CutCell method:



CutCell results:

Contours of Static pressure on the wheel and velocity pathlines



CutCell mesh: results

Mesh comparison: supersonic jet flow on a plate