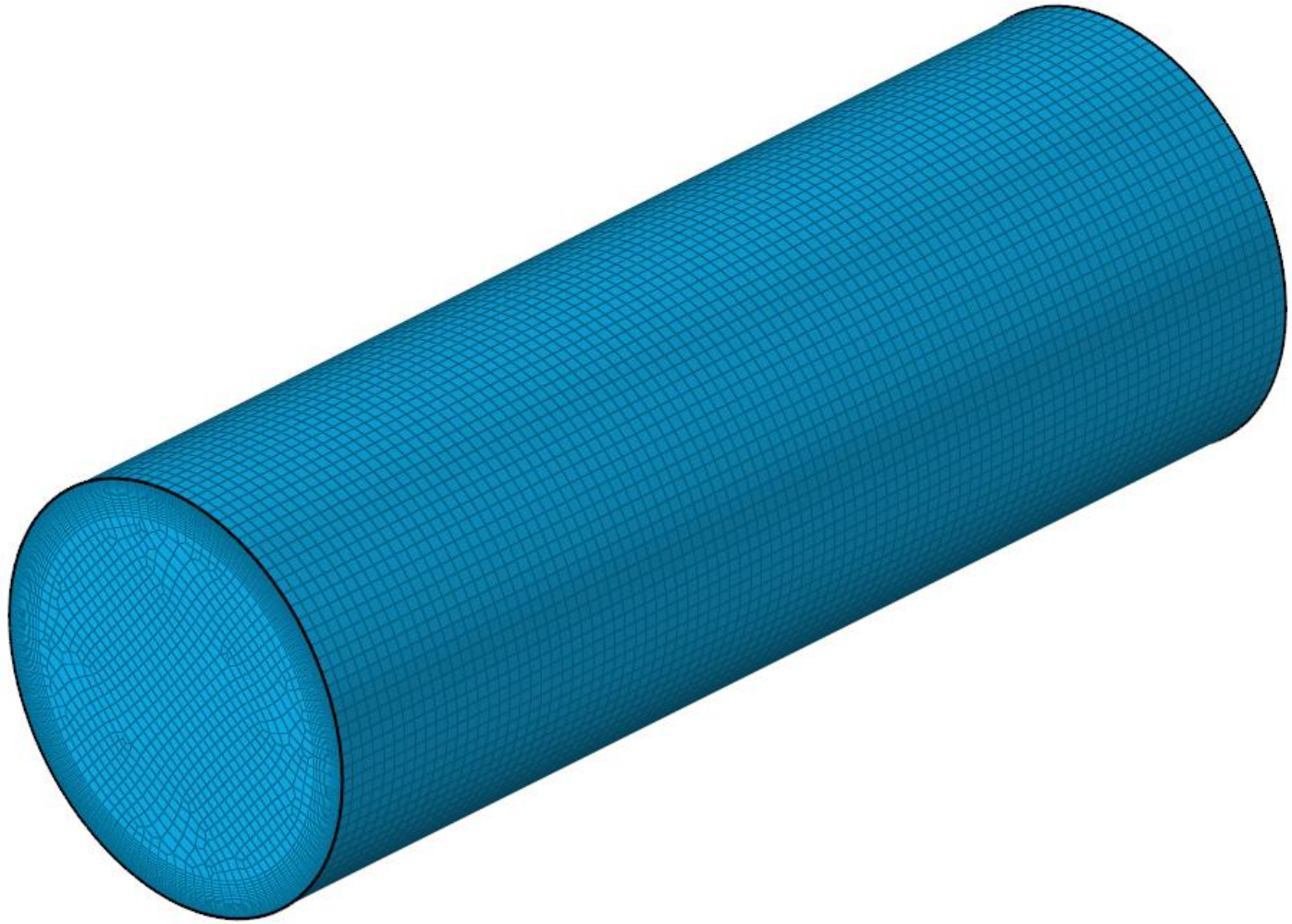
icemm
SIMULATION ENGINEERING

CFD Tutorial 10

LES Simulation & Artificial Turbulence

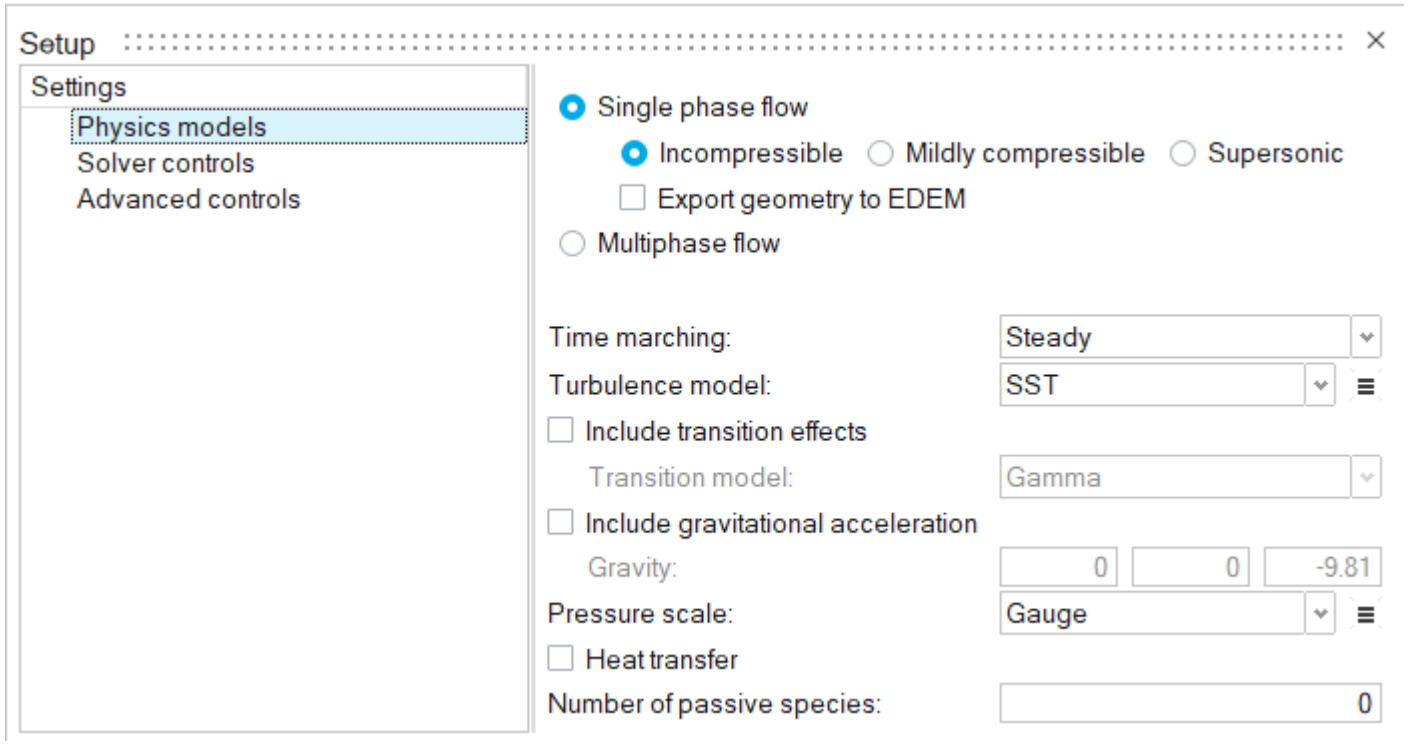
CERTIFICACIÓN DEL SISTEMA DE CALIDAD
DNV·GL
ISO 9001
PYME INNOVADORA

- In this example, a LES simulation of the flow in a simple pipe will be performed.
- The LES model allows to see the oscillations of the flow on large scales, which is useful when the flow is unsteady.
- It is also intended to check how turbulence conditions at the inlet can drastically influence the unsteady flow structure that is generated.



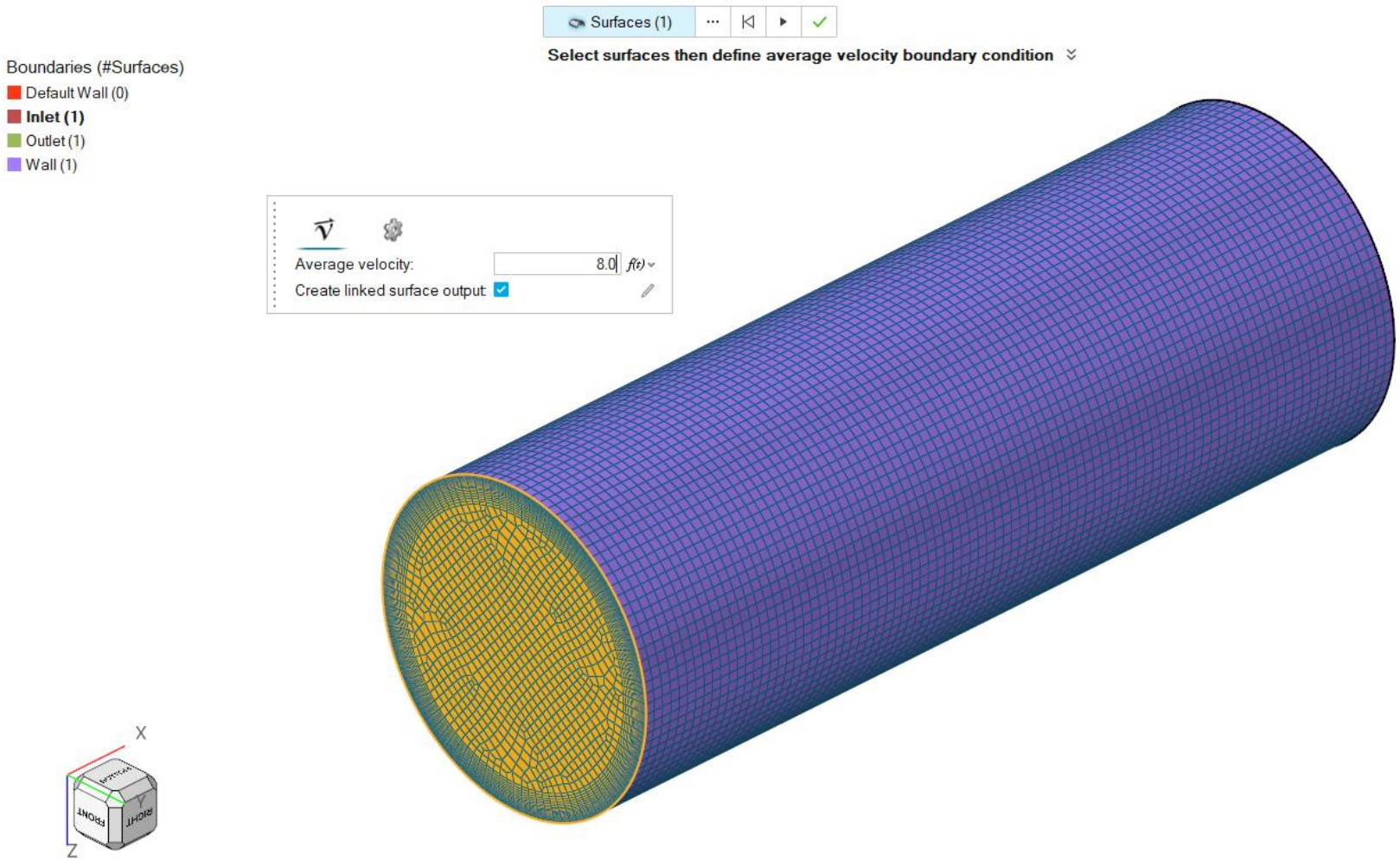
Steady-state simulation: Physics

- The geometry of the already meshed model is imported as a Fluent file (.cas) directly into Hyperworks CFD.
- In Flow>>Physics the simulation type is set.
- First a stationary case with SST turbulence model is simulated.
- This will be used as the initial condition of the transient simulation to improve and speed up the convergence.



Boundary conditions

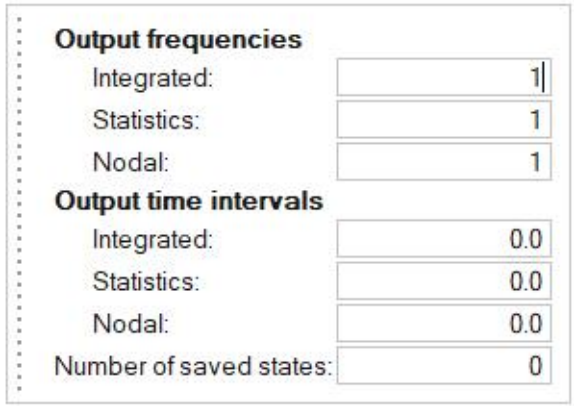
- As boundary conditions, a mean velocity (Profiled Inlet) of 8 m/s is imposed.
- A pressure of 0 Pa is selected at the outlet.
- The wall is of type No-Slip Wall with Wall Resolved for the boundary layer.

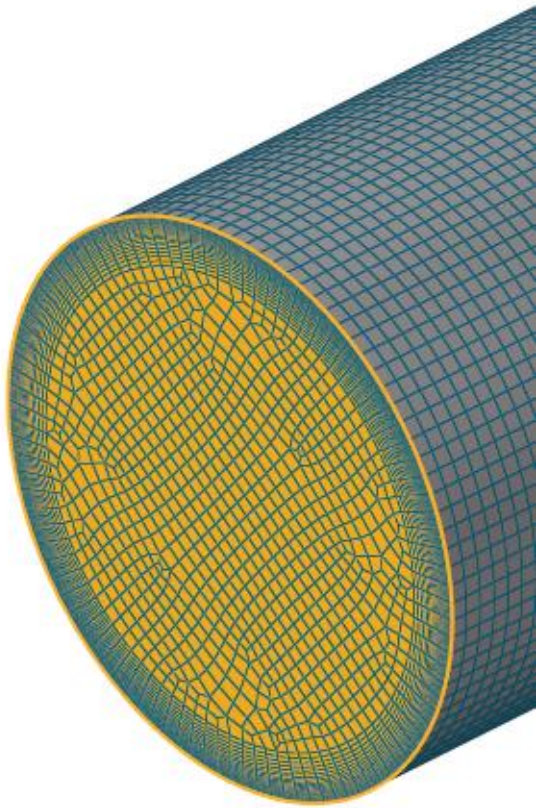


Surface output

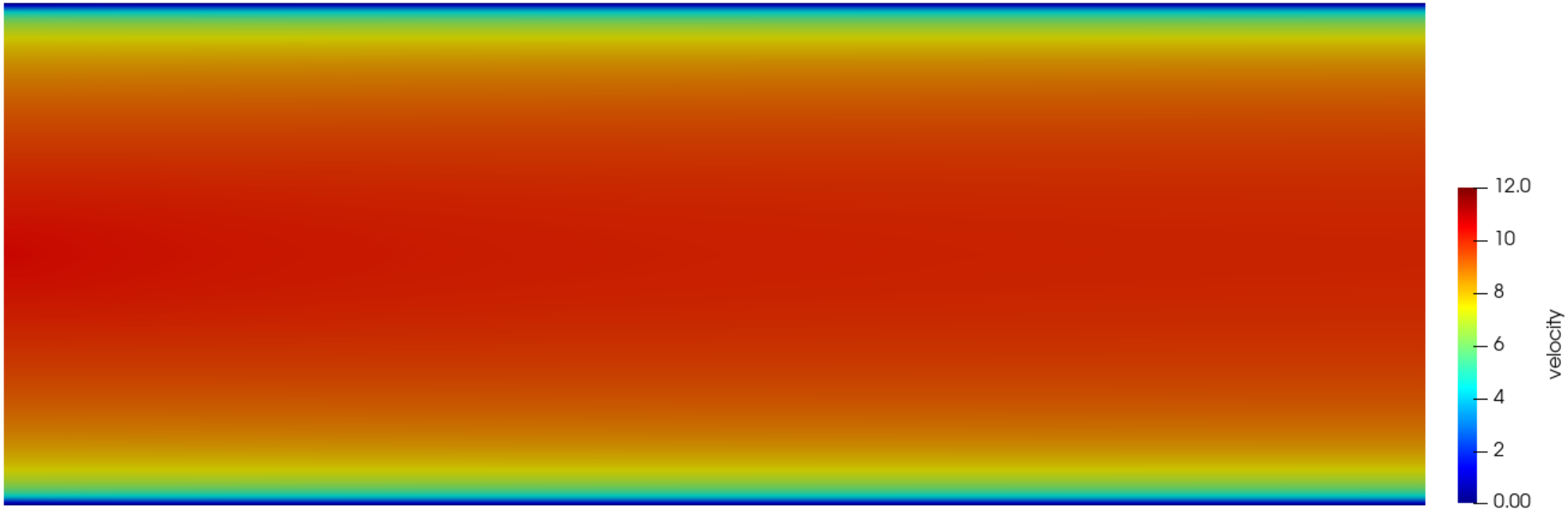


- A Surface Monitor with a nodal frequency of 1 is placed in the inlet.
- The simulation is launched from Solution>>Run.



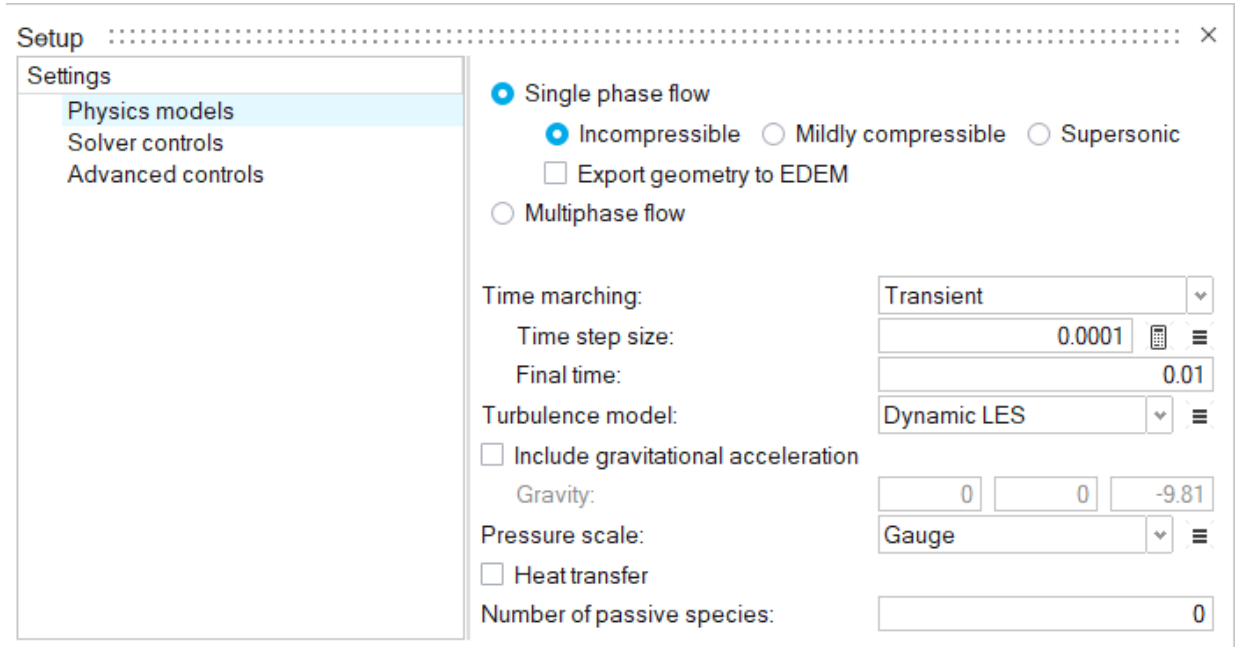


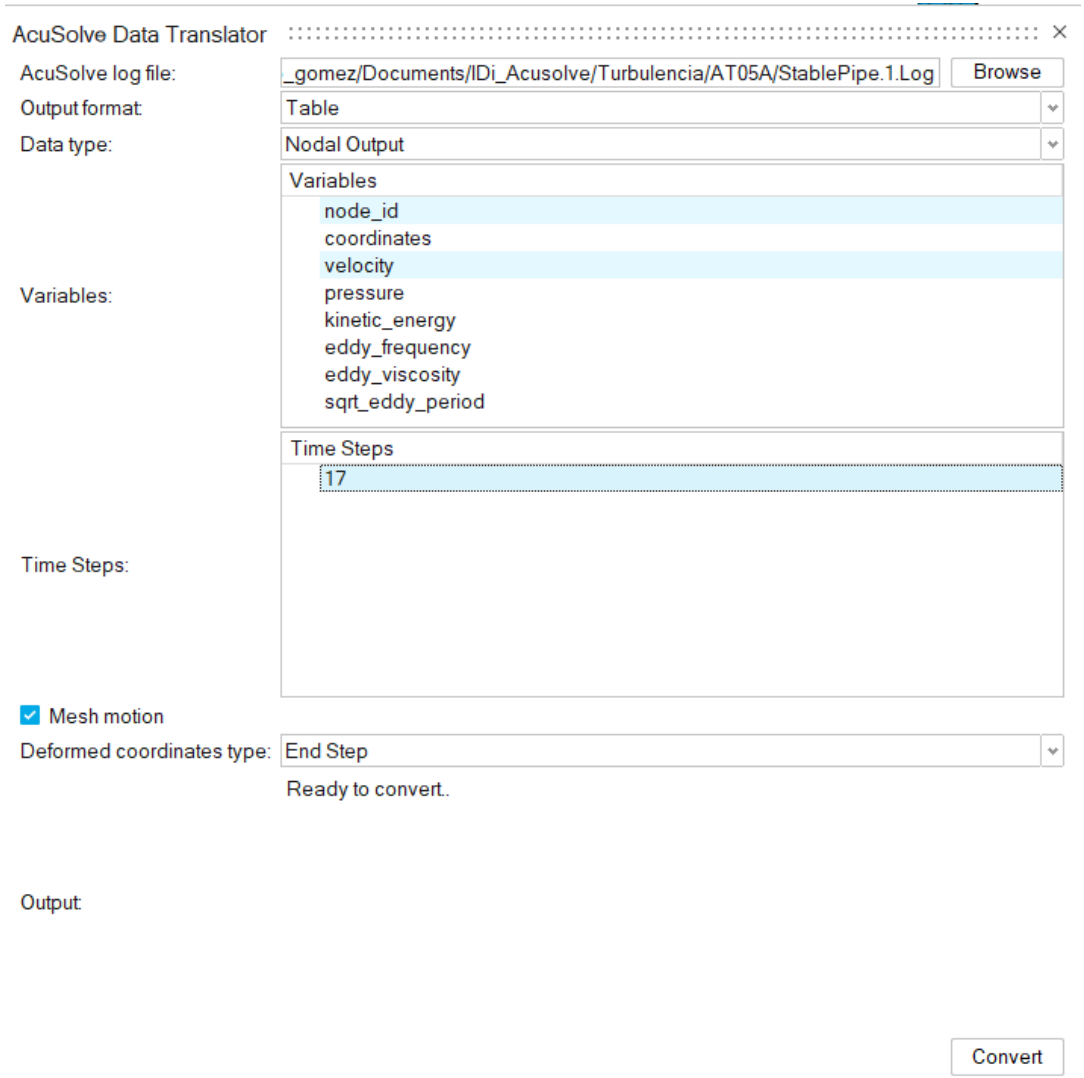
- The velocity results are the ones expected for the flow in the pipe.
- They will serve as an initial condition in the LES simulation to accelerate its convergence.



Transient case: Export steady-state solution

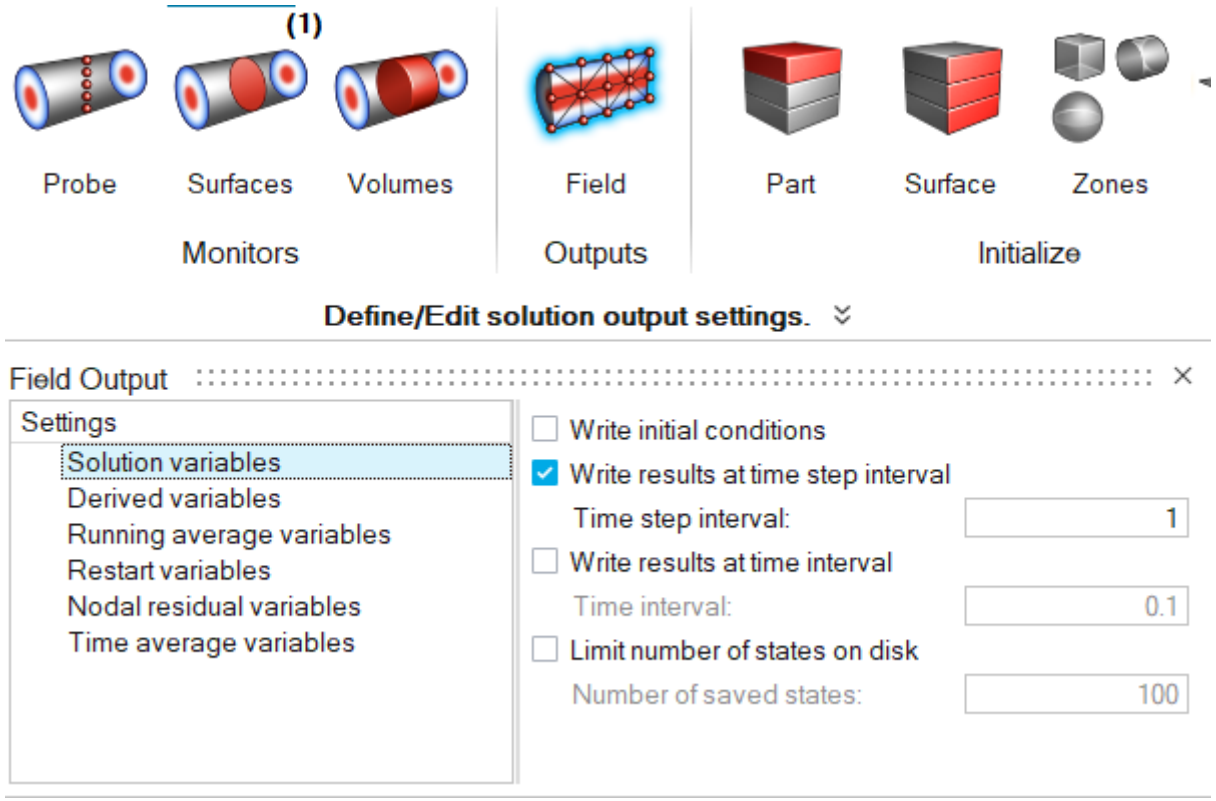
- The velocity and pressure solution is exported with the process already used in other occasions, from Solution>>Convert to Table format and properly changing the names of the files before exporting the next one.
- The transient simulation is done with the Dynamic LES model with the time step of the image to obtain good results.
- LES methods require a fine mesh size and a low time step to show the unsteady flow movements.





Transient case: Field Outputs

- To see the evolution in the transient analysis, from the Field Outputs menu, the Write Time Step Interval must be changed to 1, both in Solution Variables and Derived Variables.



Transient case: Set initial conditions

- After the changes in the Flow>>Physics section, the model is exported from Solution>>>Run>>>Export.
- In the .inp file, the lines of the image are added replacing the default initial condition.
- The .out files are those exported before from the stationary solution.

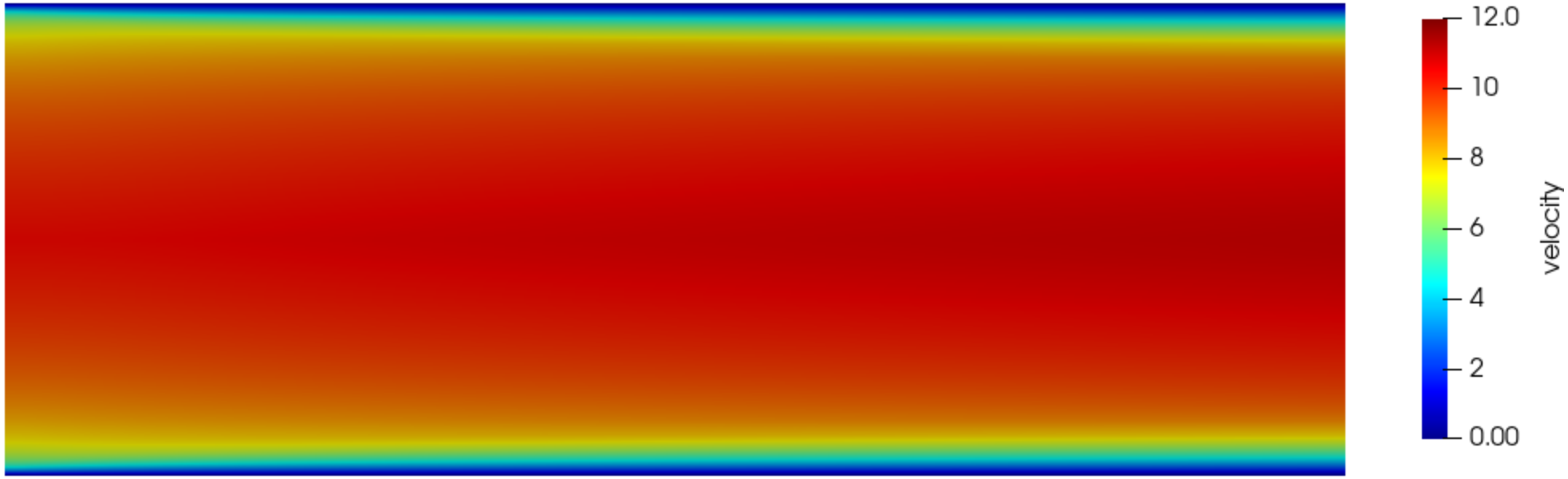
```
# +--------------------------------------------------------------------+
# | Nodal Initial Condition                                            |
# +--------------------------------------------------------------------+

NODAL_INITIAL_CONDITION( "pressure" ) {
    selection_type                  = all
    type                            = nodal_values
    variable                        = pressure
    default_value                   = 0
    nodal_values                    = Read("StablePipe_step17_Pressure.out")
    precedence                      = 0
    satisfy_boundary_condition      = off
}

# +--------------------------------------------------------------------+
# | Nodal Initial Condition                                            |
# +--------------------------------------------------------------------+

NODAL_INITIAL_CONDITION( "velocity" ) {
    selection_type                  = all
    type                            = nodal_values
    variable                        = velocity
    default_values                  = {1,0,0}
    nodal_values                    = Read("StablePipe_step17_Velocity.out")
    precedence                      = 0
    satisfy_boundary_condition      = off
}
```

- The results throughout the entire transient simulation are stable and similar to the picture shown, which also closely resembles the stationary case.
- This is because nothing initiates turbulence in the flow during the simulation, which in many situations does not reflect reality well.



Synthetic/Artificial Turbulence

- To give turbulence values, a Synthetic or Artificial Turbulence card can be included in the inp file and the inlet can be modified as shown in the image to indicate the turbulence values.
- In this case, high values are taken to get a good view of the phenomenon, similar to those set by an Ansys Fluent example of this type.
- Appropriate values of the turbulence parameters (K-omega or K-epsilon) can be estimated with online tools such as::
  - https://www.cfd-online.com/Tools/turbulence.php
  - https://2022.help.altair.com/2022.1/hwcfdsolvers/acusolve/topics/acusolve/training_manual/inlet_turb_params_r.htm

```
# +-------------------------------------------------------------------------+
# | Turbulence model options                                                |
# +-------------------------------------------------------------------------+

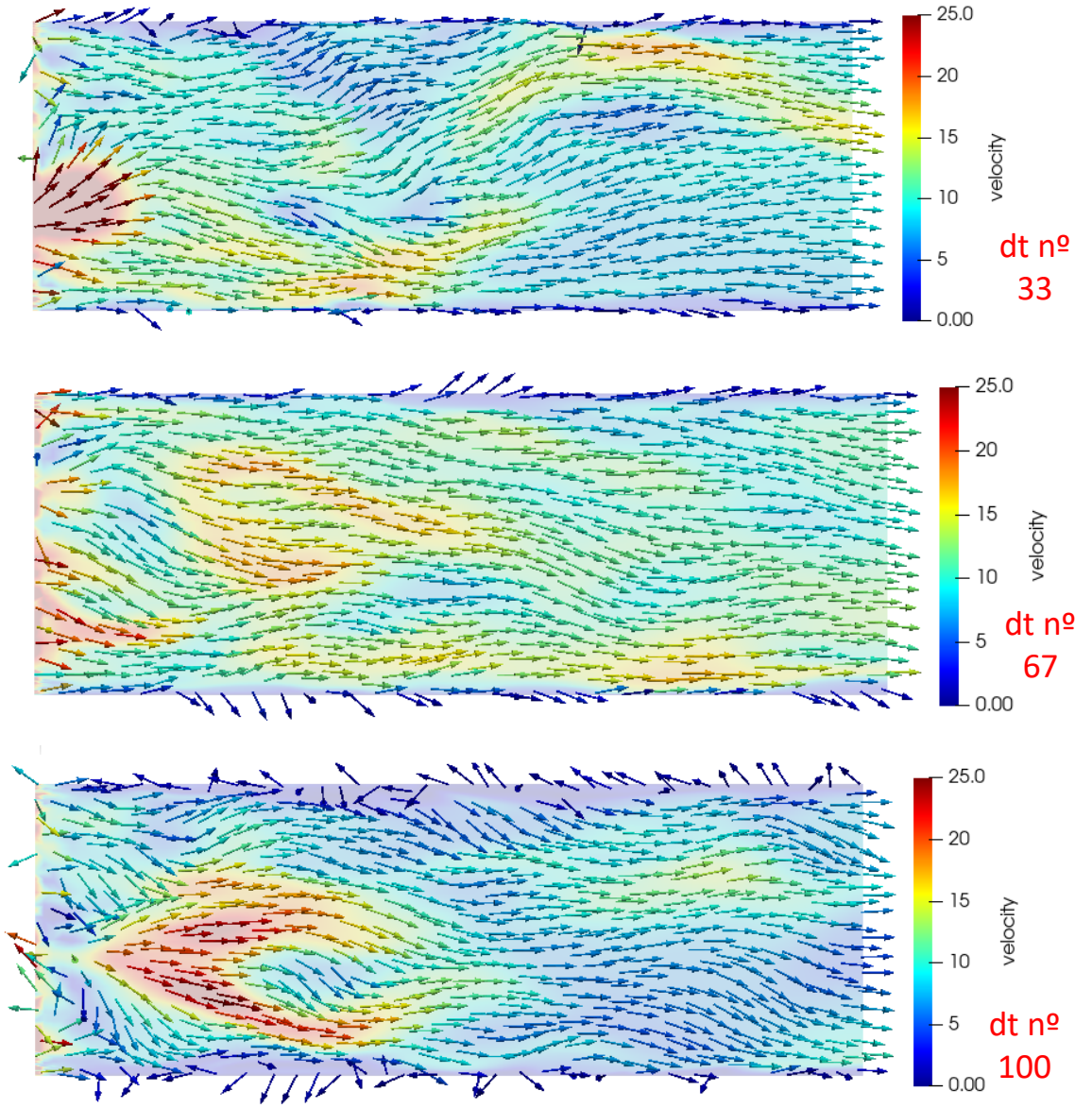
TURBULENCE_MODEL_PARAMETERS {
    synthetic_turbulence_input_type = k_omega
}
```

```
# +-------------------------------------------------------------------------+
# | Simple Boundary Condition                                               |
# +-------------------------------------------------------------------------+

SIMPLE_BOUNDARY_CONDITION( "Inlet" ) {
    surface_sets                    = { "StablePipe_ConPert.SurfaceSet_3.quad4.SolidBody_2_1.hex8" }
    type                            = inflow
    inflow_type                     = velocity
    inflow_velocity_type            = cartesian
    x_velocity                      = 8
    active_type                     = all
    precedence                      = 1
    synthetic_turbulence            = on
    kinetic_energy                  = 500
    eddy_frequency                  = 200
}
```

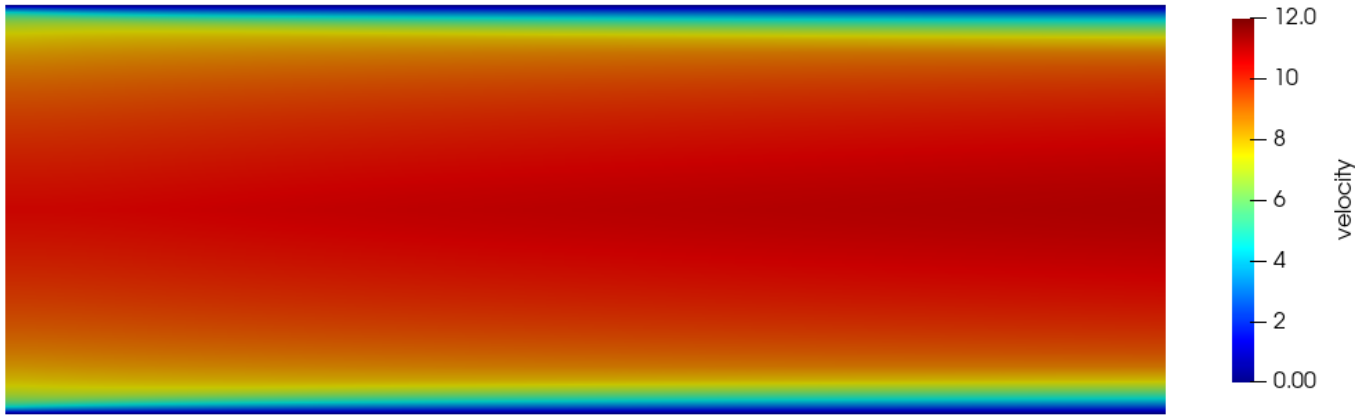
Synthetic/Artificial Turbulence

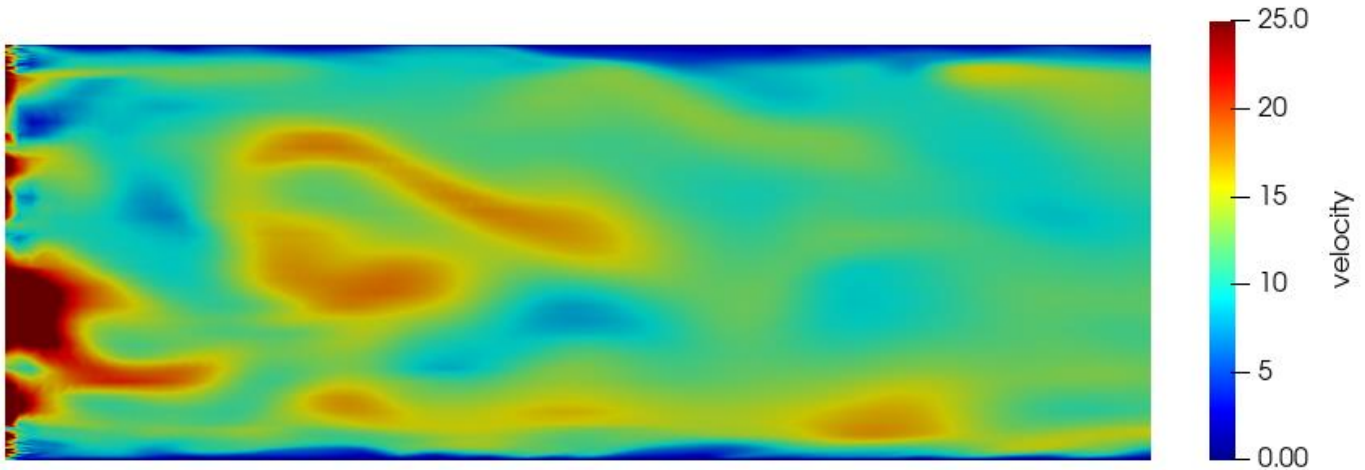
- When artificial turbulence is introduced, all other conditions being equal in the simulation, the results become much more unstable and vortex structures form, which can be captured by the LES method.
- Recirculations appear on the walls. The arrows that appear to be sticking out are due to the representation of the field with uniform arrow size. Their dark blue colour shows that the velocity is actually very low.



Conclusions

- It can be seen that the LES simulation allows us to see the vortex structures formed by the flow in the transient.
- These are strongly conditioned by the inlet parameters, which can greatly vary the results.
- The estimation of the inlet turbulence parameters is not always straightforward, and must be done carefully to obtain realistic flow solutions.





EDIFICIO ANTARES

C/ INDUSTRIAS, 8

28923 – ALCORCÓN (Madrid) – España

Tel.(+ 34) 910 569 228 – (+ 34) 629 87 94 37

. . . . .

info@icemm.es

www.icemm.com