icemm
SIMULATION ENGINEERING

CFD Tutorial 02

Vortex Shedding: Transient Analysis

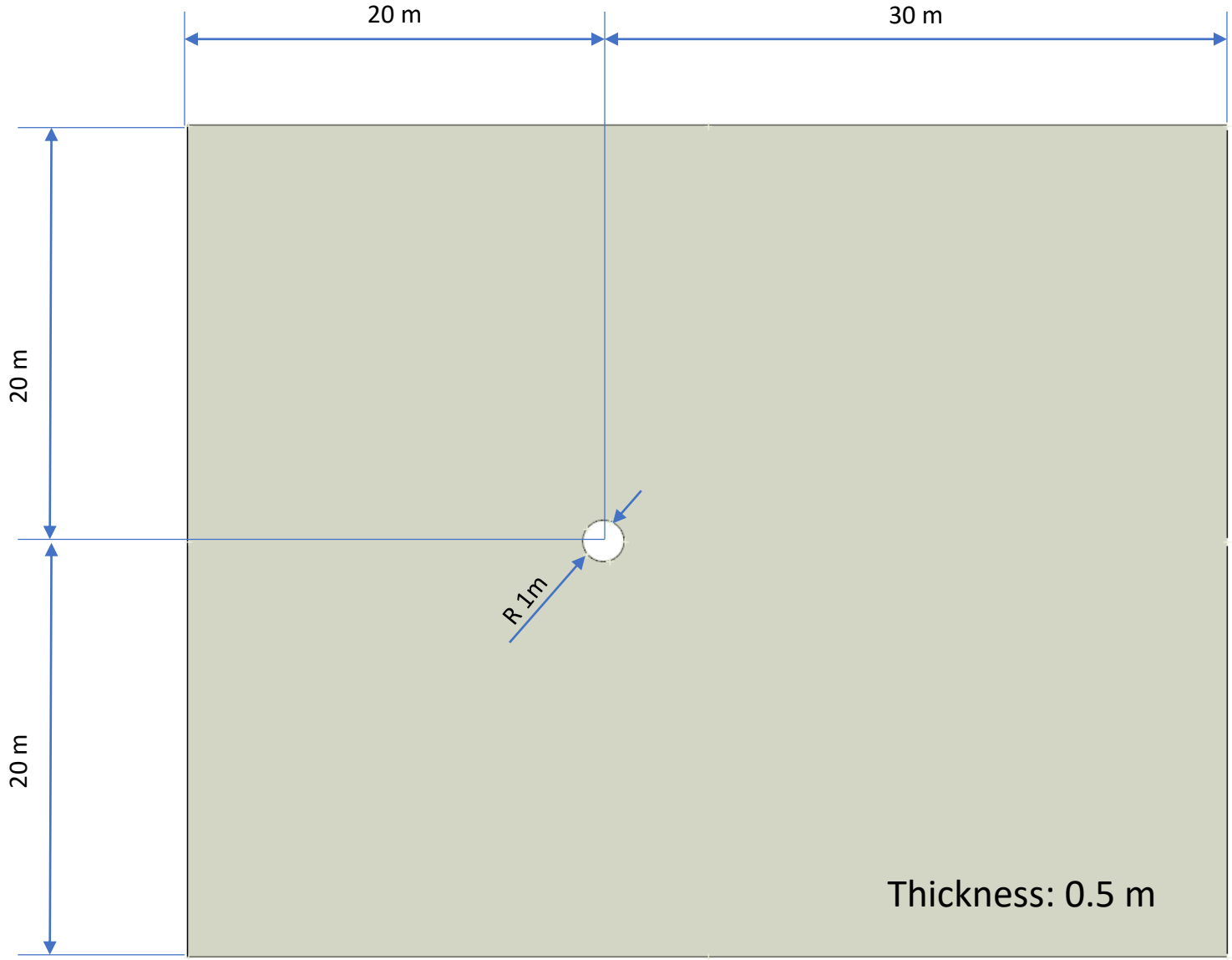
CERTIFICACIÓN DEL SISTEMA DE CALIDAD
DNV·GL
ISO 9001
PYME INNOVADORA

- The aim is to simulate the transient flow that appears around a cylinder when the Reynolds number is close to 100.
- First, the steady case will be solved in order to use the results as the initial condition of the transient simulation.

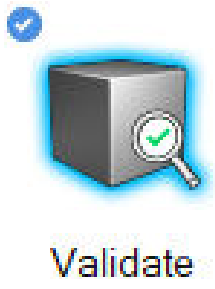
Flow pattern	Reynolds number	Description
	$Re < 5$	No separation, laminar steady flow
	$5 < Re < 45$	Pair of vortices, laminar steady flow
	$45 < Re < 150$	Laminar vortex street, unsteady flow
	$150 < Re < 3 \cdot 10^5$	Transitional unsteady flow
	$3 \cdot 10^5 < Re < 3 \cdot 10^6$	Turbulent unsteady flow
	$Re > 3 \cdot 10^6$	Turbulent vortex street, unsteady flow

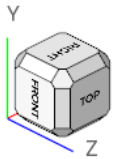
https://www.idealsimulations.com/resources/reynolds-number/

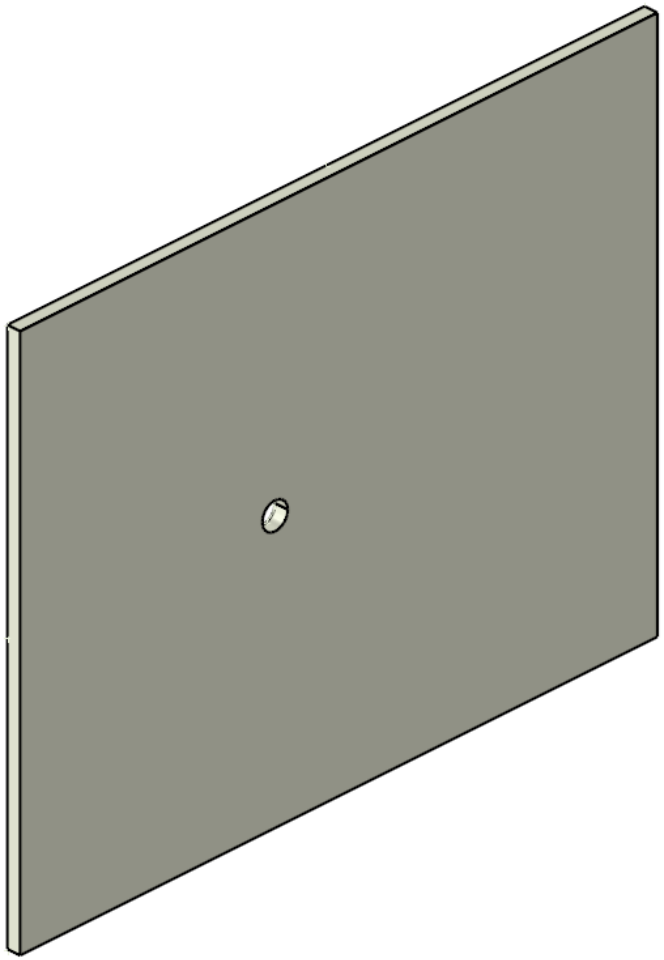
- The domain geometry is shown in the image on the right. As this is a simple geometry, it could be created in Hyperworks CFD. It can also be imported from another CAD software.
- The thickness is chosen to match the size of the element to be used as the analysis is (almost) two-dimensional.



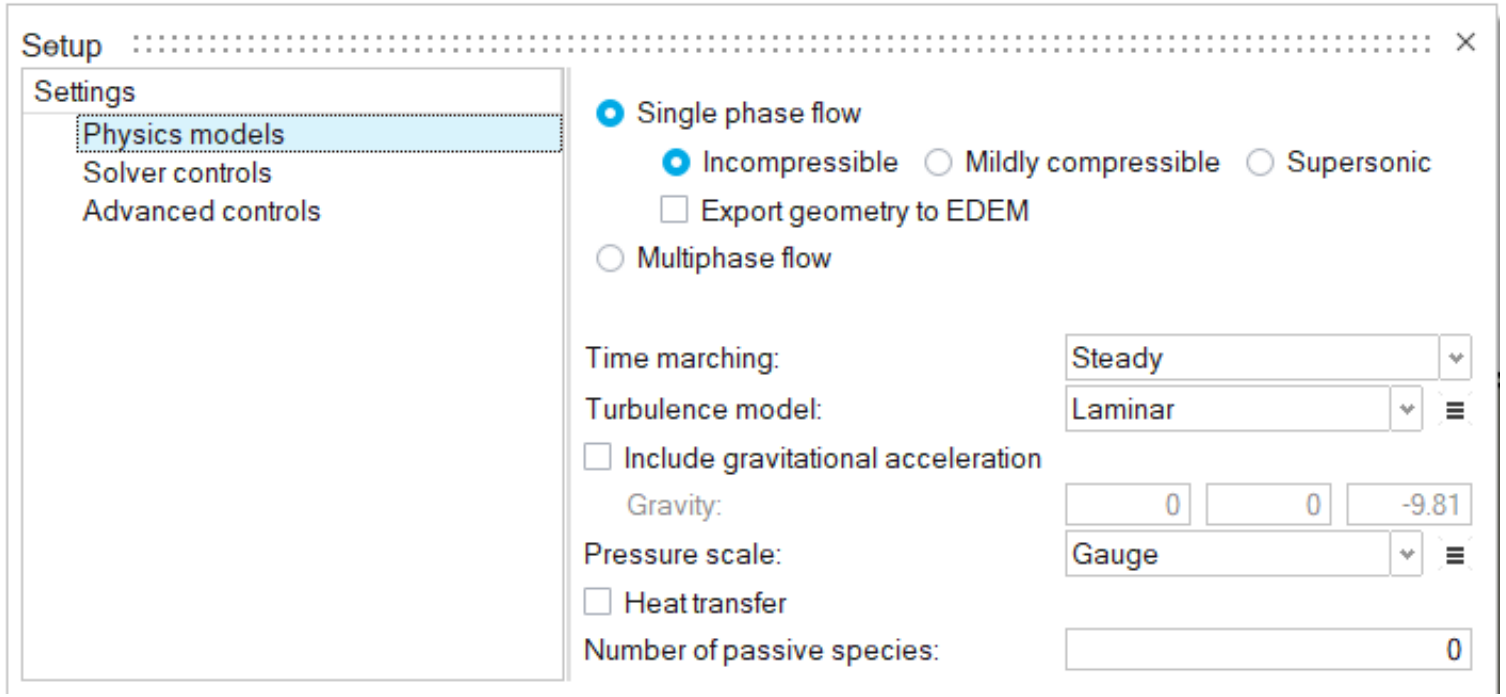
- After building the geometry, click on Validate and check that the blue Tick required for meshing appears.







- First, a stationary analysis is carried out to obtain the initial conditions.
- Laminar turbulence is chosen as the turbulence model based on the estimated Reynolds number.

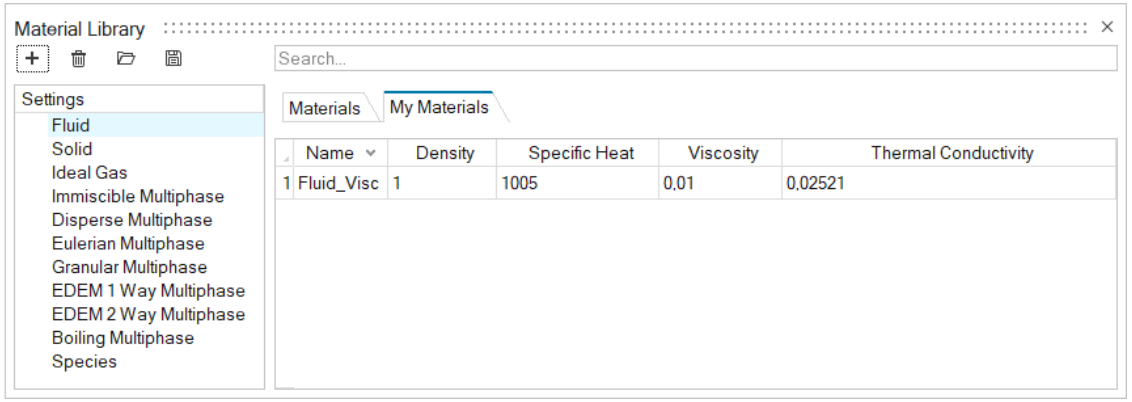


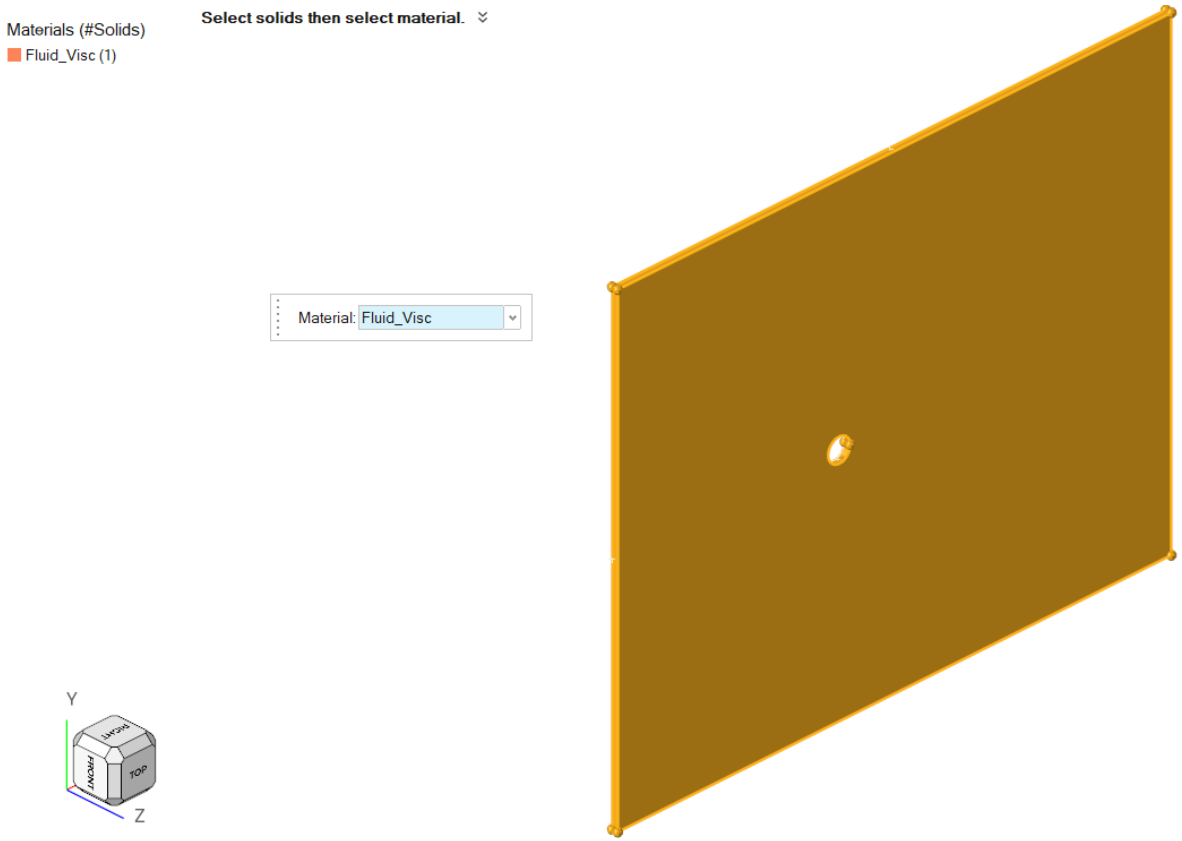
Flow: Material

- Next, a new material is defined with a density of 1 kg/m3 and a viscosity of 0.01 Pa-s.
- Thus we have, defining an inlet velocity of 1 m/s:

$$Re = \frac{UL\rho}{\mu} = \frac{U_{inl} \cdot R_{Cil} \cdot \rho}{\mu} \approx 100$$

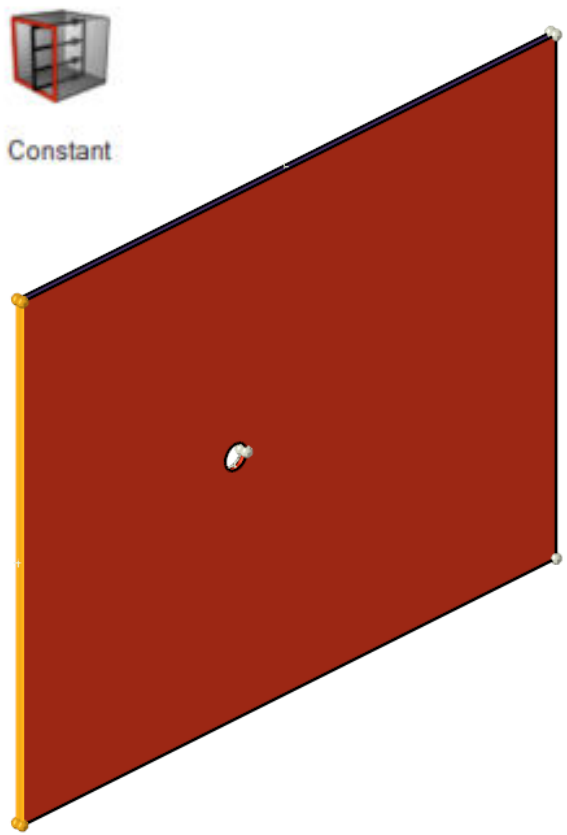
- This material is assigned to the domain

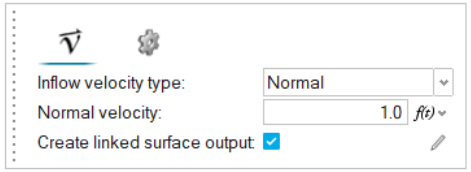


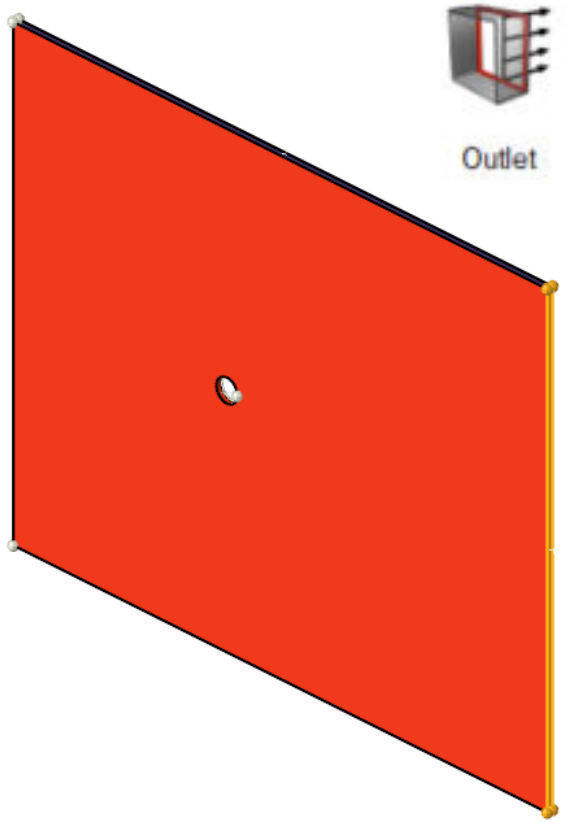


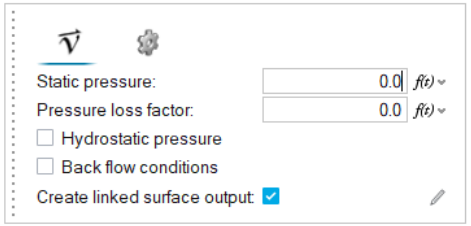
Flow: Boundary Conditions

- The boundaries shown in the picture are defined as inlet and outlet.
- According to the desired Reynolds number, 1 m/s is chosen as the inlet velocity.







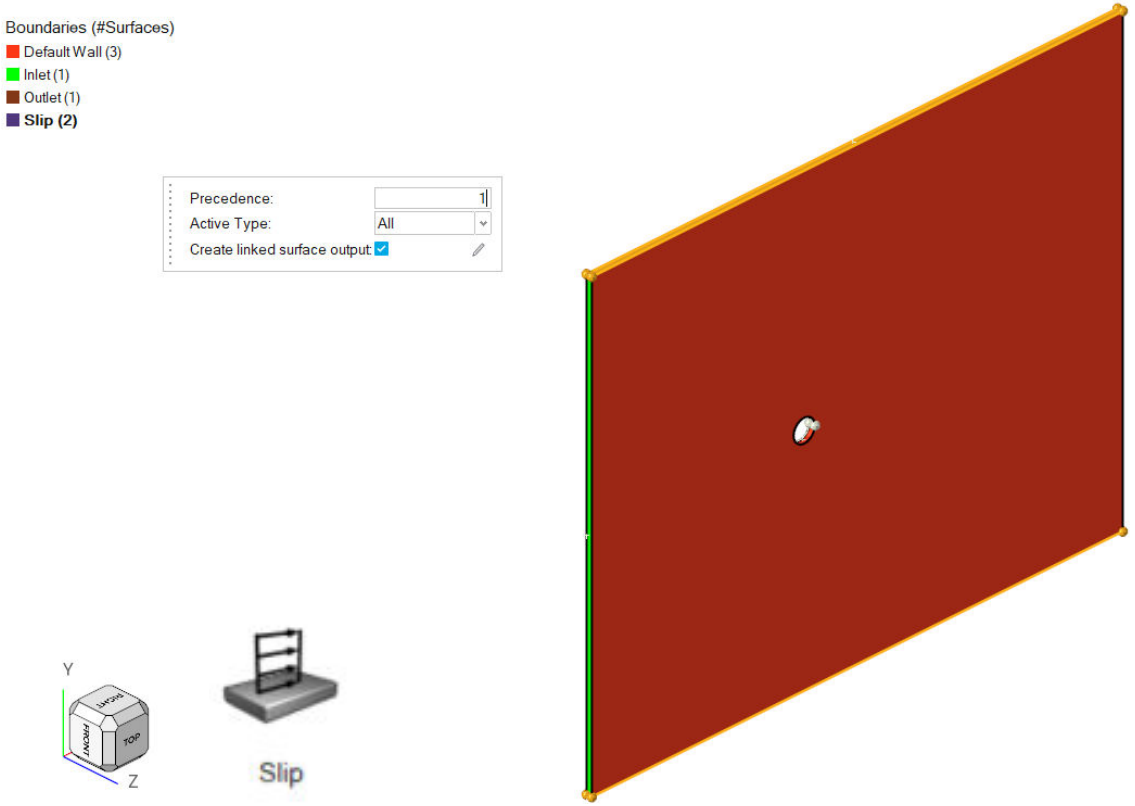


Flow: Boundary Conditions

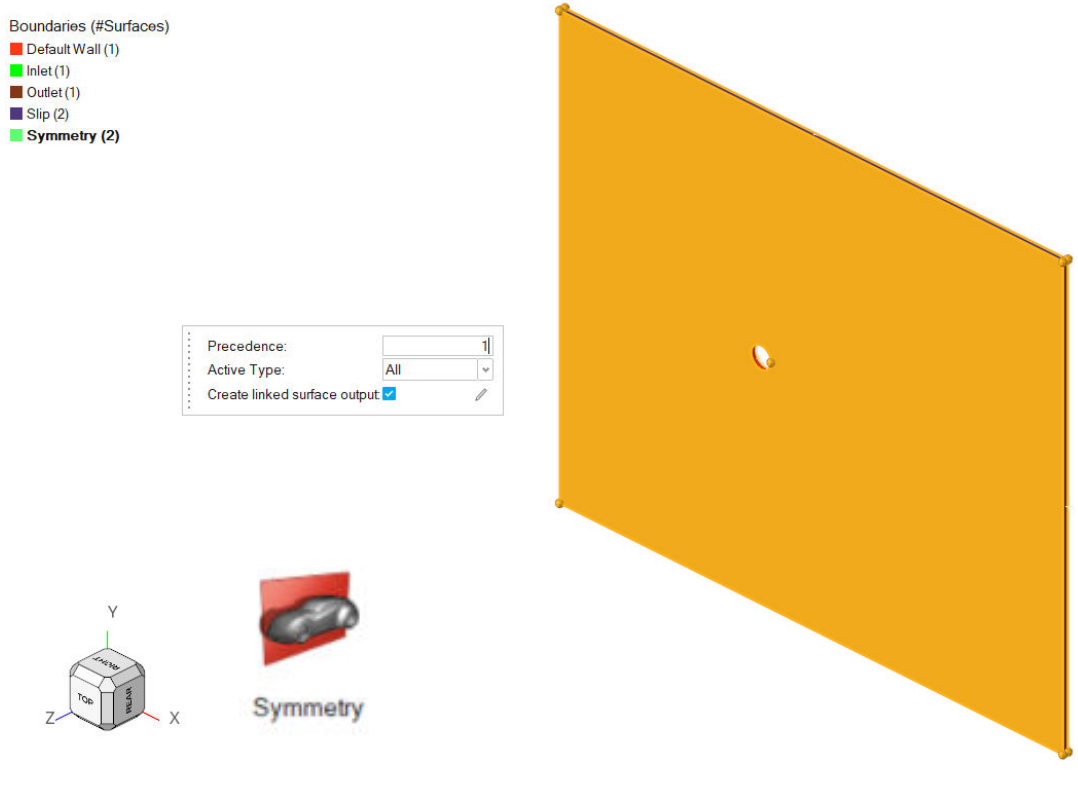
- Slip wall conditions are defined for the upper and lower boundaries.
- For the lateral boundaries, symmetry conditions are defined to maintain the desired two-dimensionality conditions.
- The cylinder wall is left as No-Slip Wall.



No Slip

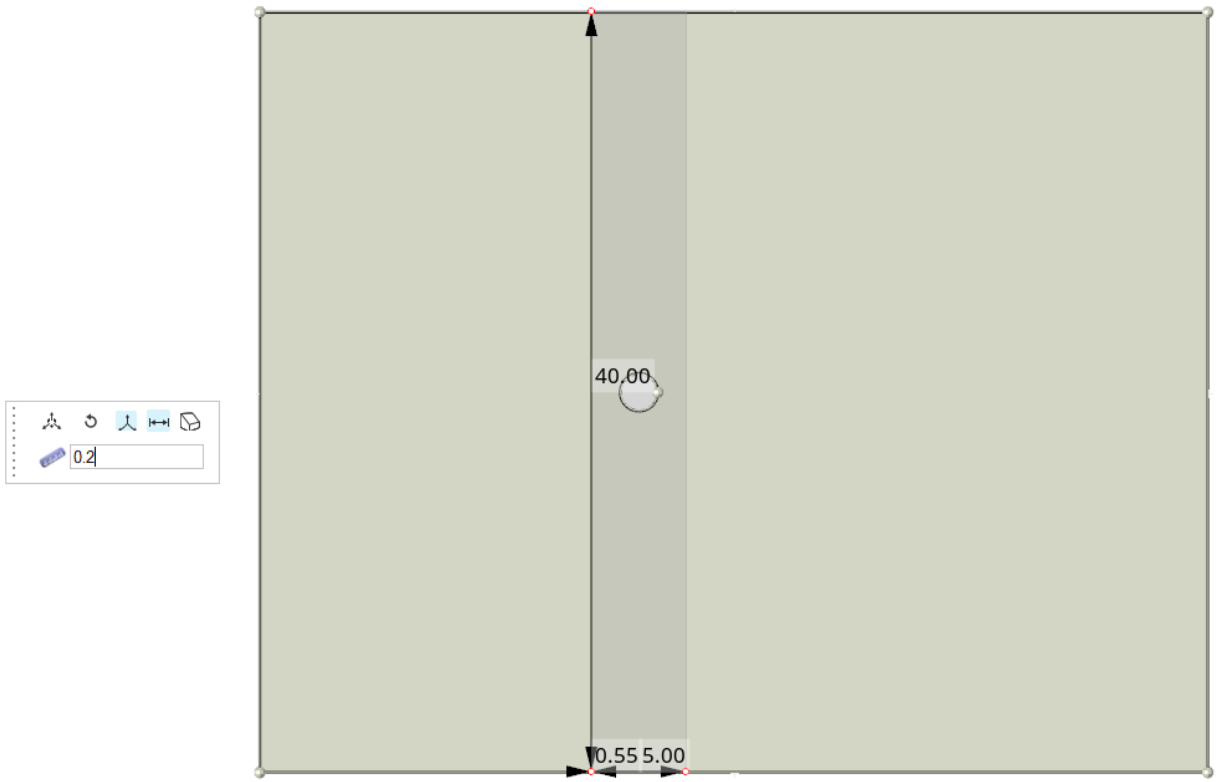


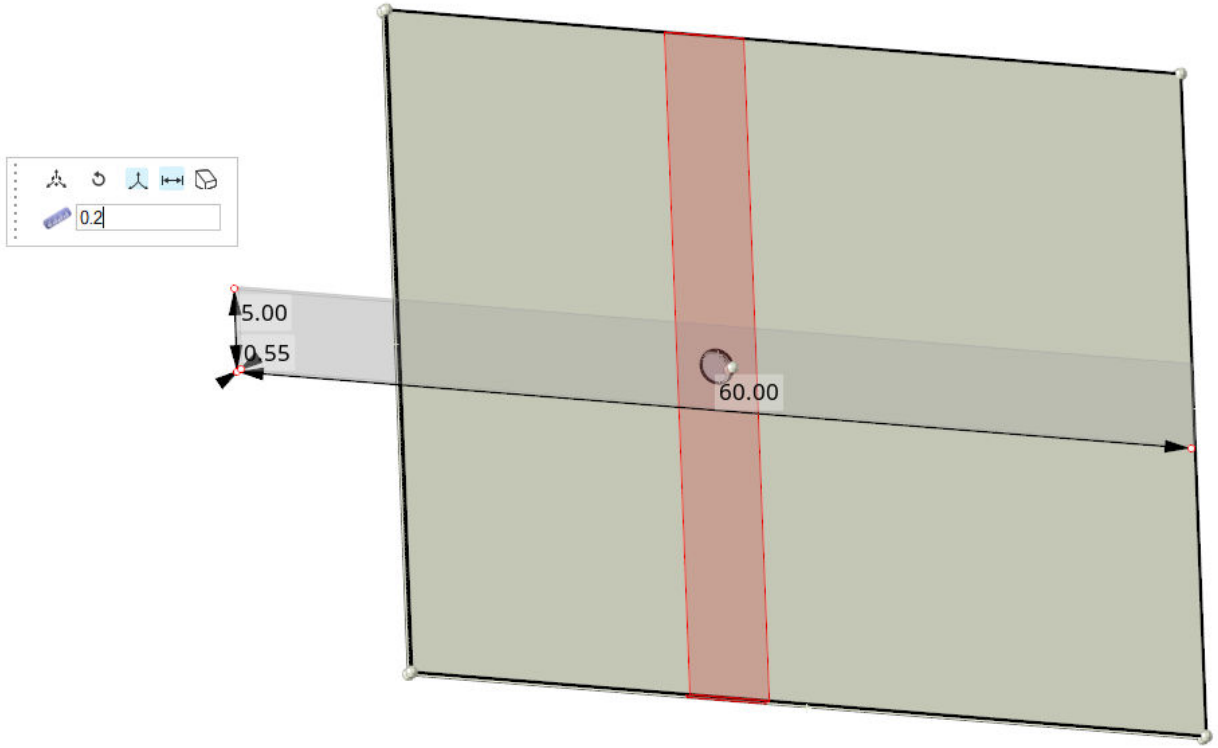
Slip



Symmetry

- To mesh the domain, we first create two prismatic refinement zones in the sections close to the cylinder as shown in the images. The element size will be 0.2 m in these regions.



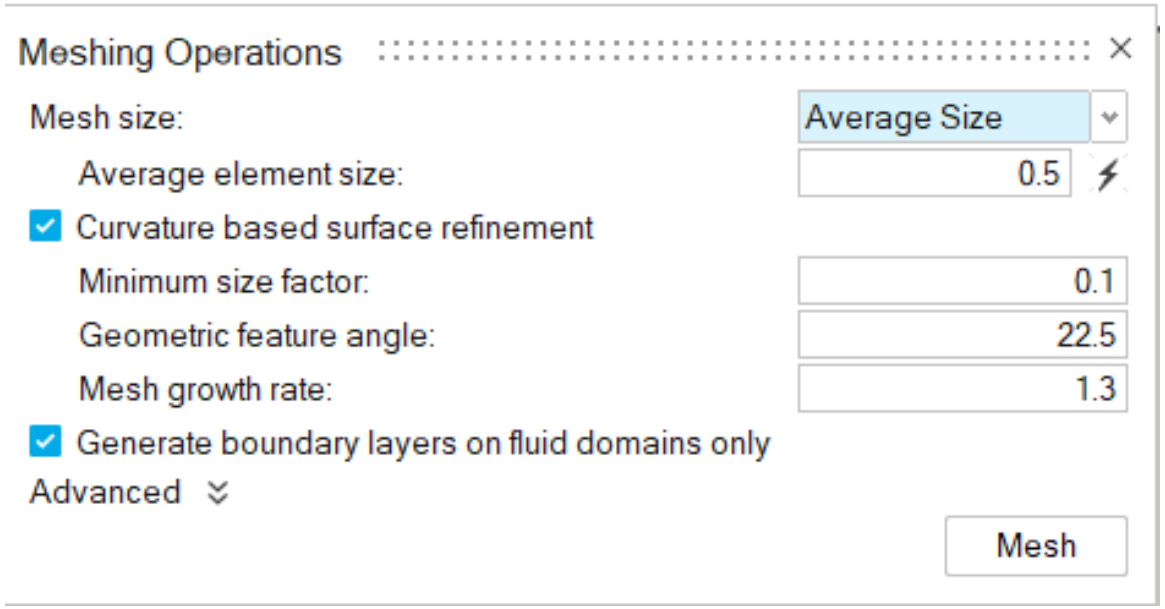


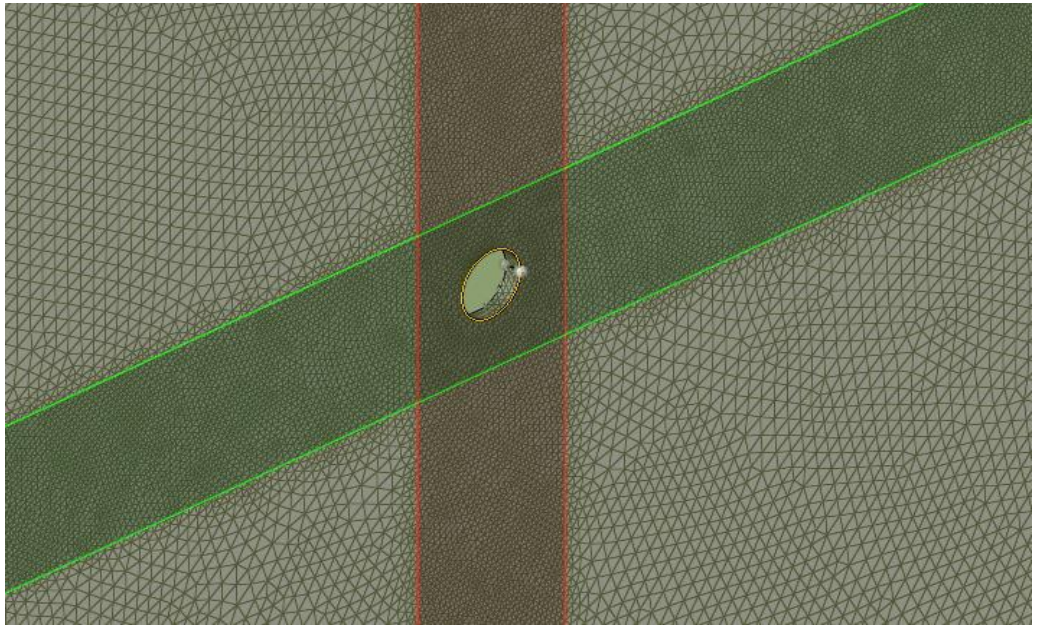
- Then select Volume Mesh and choose 0.5 m as the average element size, equal to the thickness of the domain.
- Click on Mesh to generate the mesh, which should look like the one in the image.
- By clicking on the icon that appears in blue below the Measure option, you can obtain a count of the nodes and elements generated.



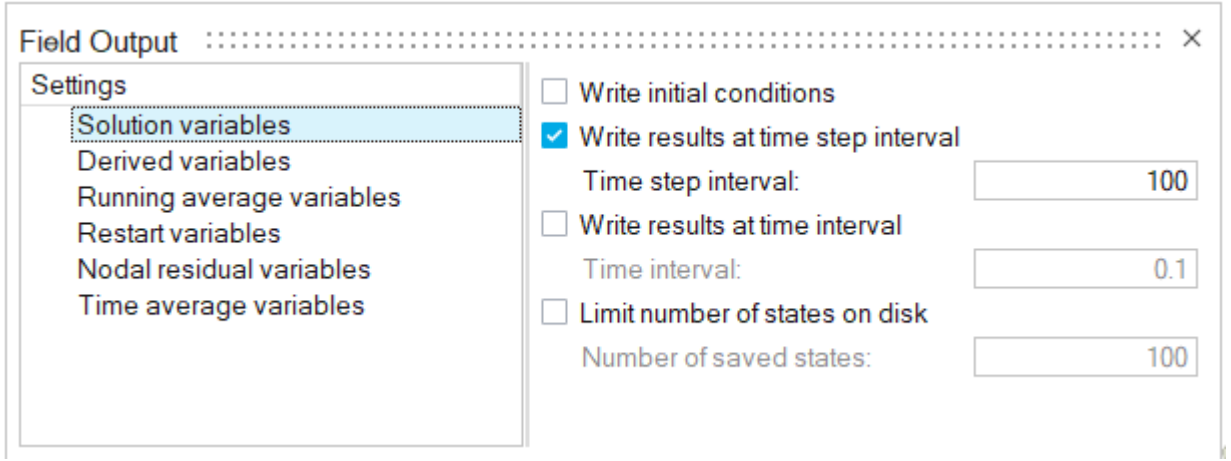
Entity Count

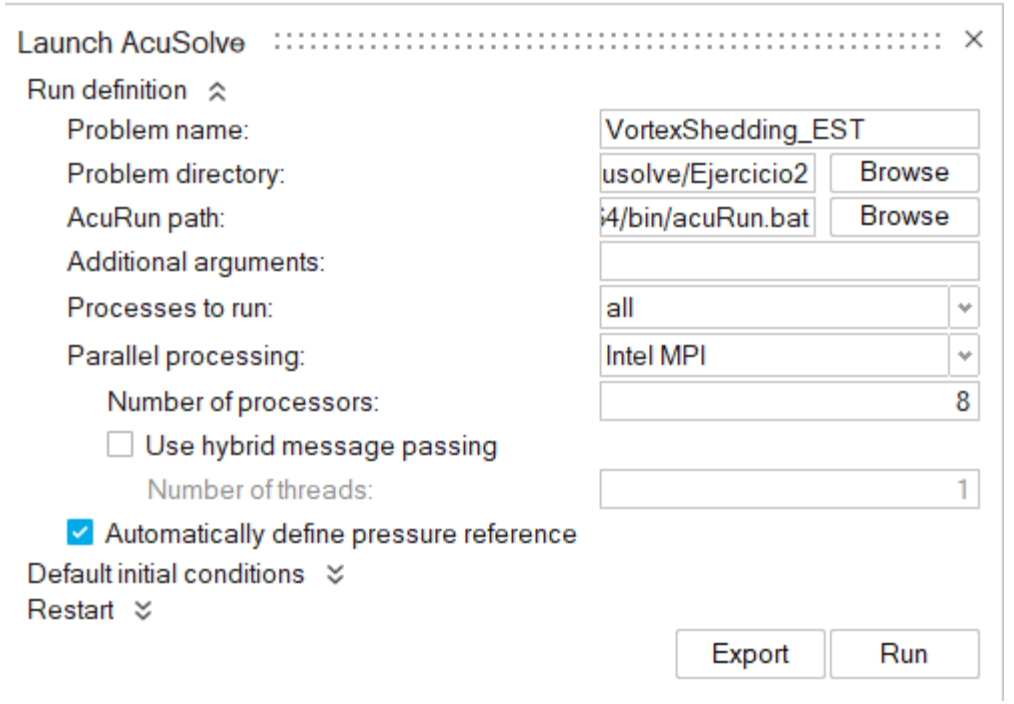
Entity	Count
Points	10
Lines	20
Surfaces	7
Solids	1
Parts	1
Nodes	83115
2d Elements	83360
3d Elements	369575





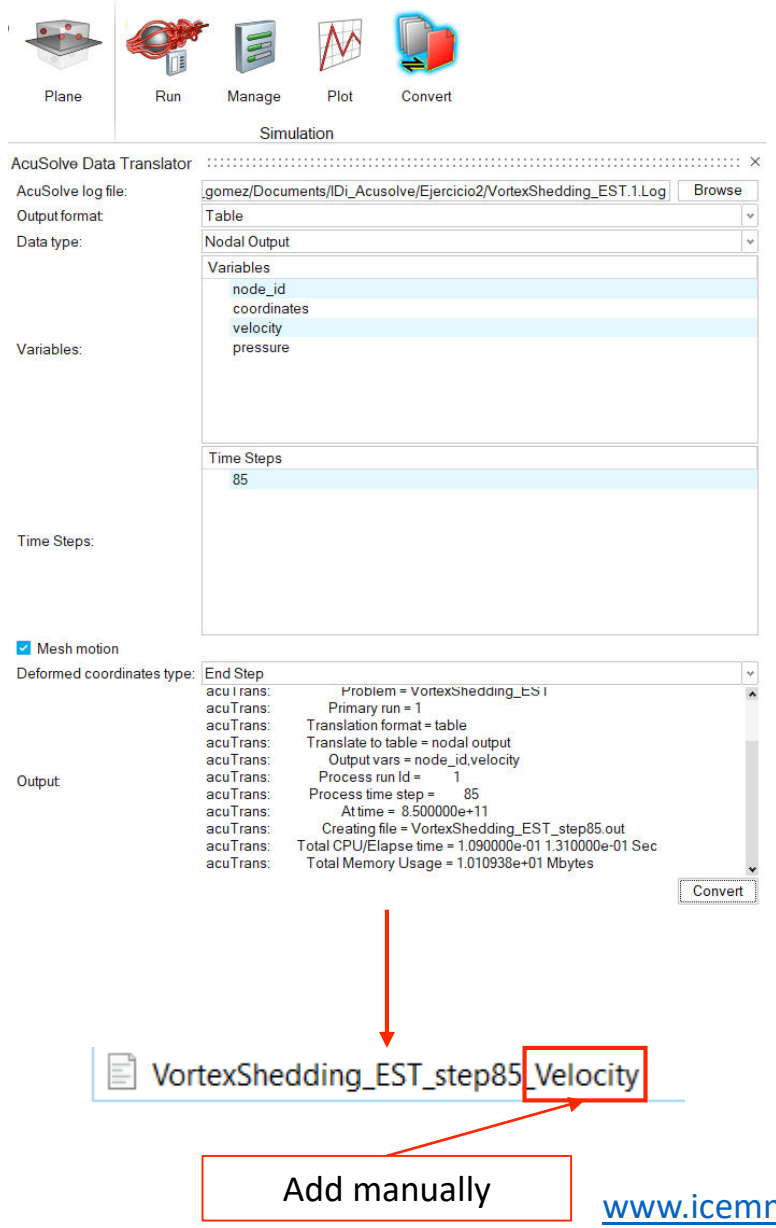
- In the Solution ribbon, check that inside the Field Output menu, the Time Step Interval field has the value 100 in both Solution Variables and Derived Variables.
- After that, give a suitable name to the problem, like VortexShedding_EST, select the folder where to generate the files in the Problem Directory and press Run to start the simulation.



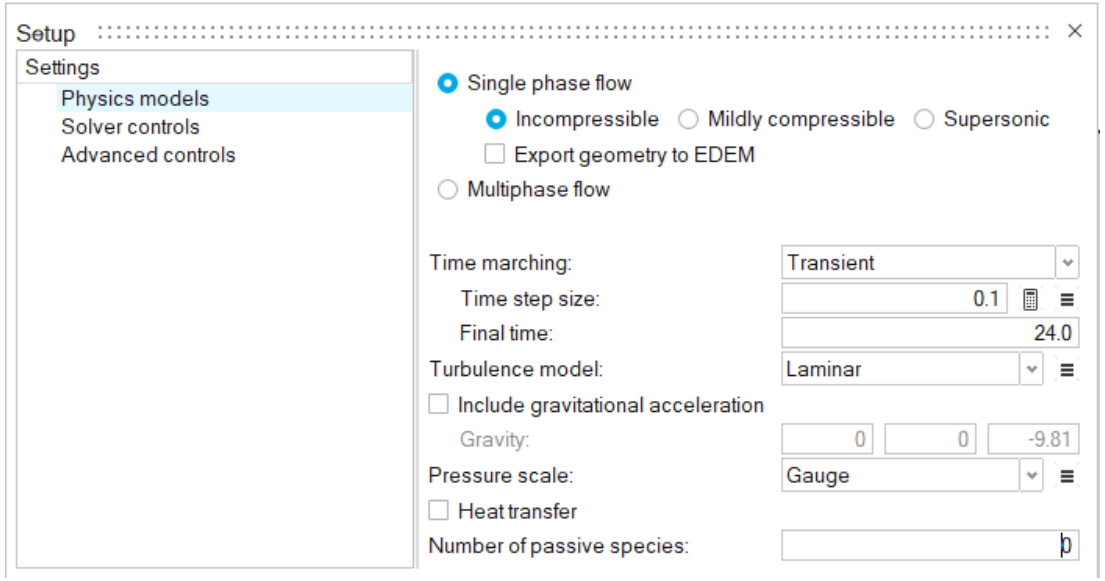


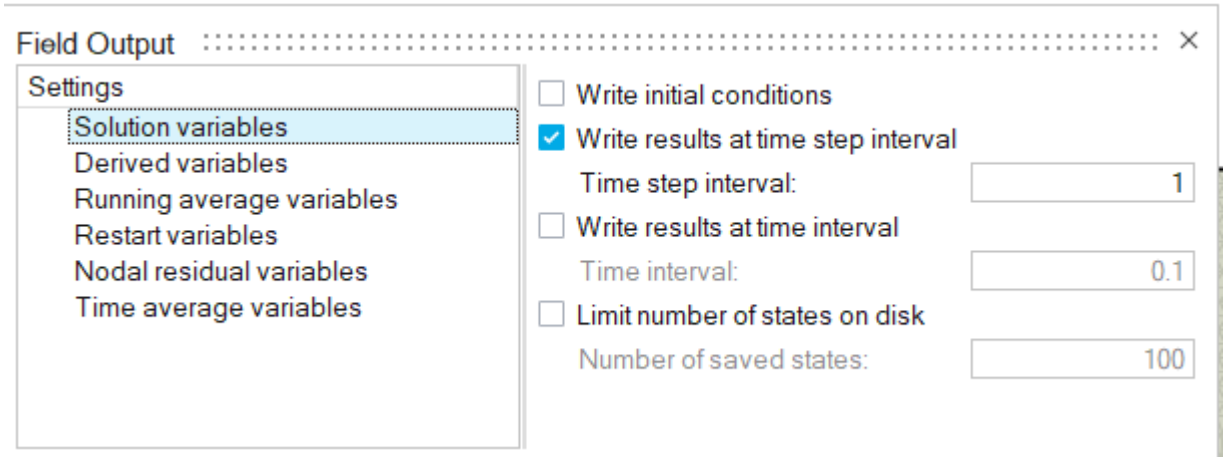
Export results

- Once the simulation is complete, its results can be viewed by importing the generated .log from the Post ribbon.
- To export the results as required, in the Solution tab, select Convert. The .log of the stationary simulation is chosen as the Acusolve log file.
- The following fields should be Output format: Table and Data type: Nodal Output.
- Under Variables, first select node_id and holding the ctrl key also select velocity. Select the last Time Step and press Convert.
- This generates an .out file in the folder where the .log is. To export various files without overwriting, they must be manually renamed from the explorer, for example by adding _Velocity at the end.
- Then repeat the process but select node_id and pressure in Variables and add _Pressure to the name of the .out file.
- This files can also be used to plot variables or represent them in external tools.



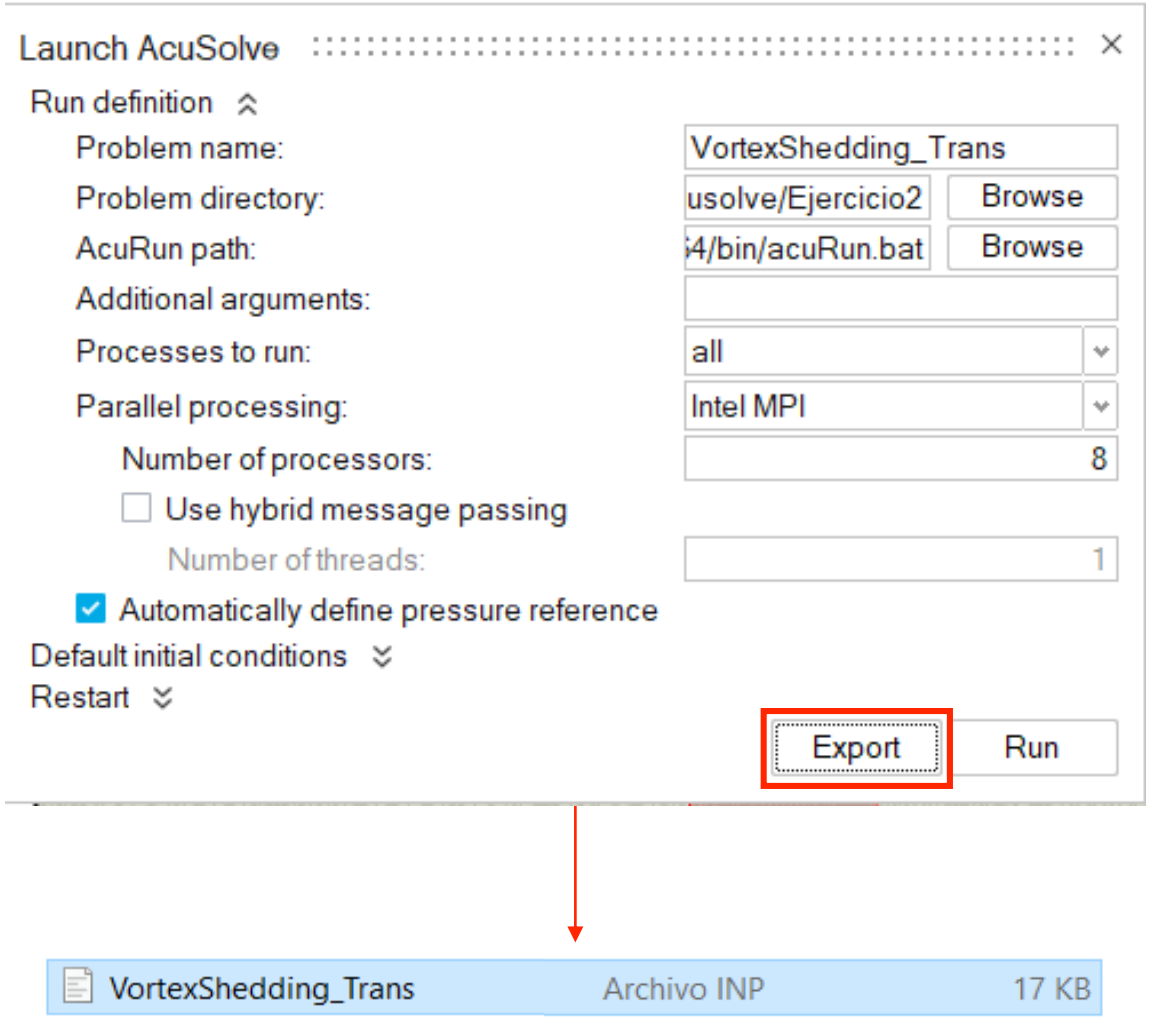
- The next step is the transient analysis. The geometry, boundary conditions and meshing must not be changed with respect to the steady state.
- The only changes to be made are:
  - In Flow>>Physics: Select Time marching: Transient and choose 0.1 as Time Step and 24.0 as Final Time.
  - In Solution>>Field Output: Change the Time Step Interval for writing solutions to 1 in both Solution variables and Derived variables.
- The time step is chosen based on known results for this type of problem.





Solution: Export

- In this case, in the Acusolve launcher menu, give the file a suitable name and click the Export button.
- This generates in the Problem Directory the .inp file that must be modified to define the initial conditions from the steady state simulation.



Modify the inp file.

- Inside the .inp file, replace the text in the red box with the text in the green box.
- The .out files must be in the same folder as the .inp file.
- The syntax on the left-hand side should be as shown, paying attention to names and singular or plural endings.

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

NODAL_INITIAL_CONDITION( "pressure" ) {
    selection_type                  = all
    type                            = constant
    default_value                   = 0
    precedence                      = 0
    satisfy_boundary_condition      = off
}
```

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

NODAL_INITIAL_CONDITION( "pressure" ) {
    selection_type                  = all
    type                            = nodal_values
    variable                        = pressure
    default_value                   = 0
    nodal_values                    = Read("VortexShedding_EST_step85_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                  = {0.001,0.001,0.001}
    nodal_values                    = Read("VortexShedding_EST_step85_Velocity.out")
    precedence                      = 0
    satisfy_boundary_condition      = off
}
```

Launch simulation externally

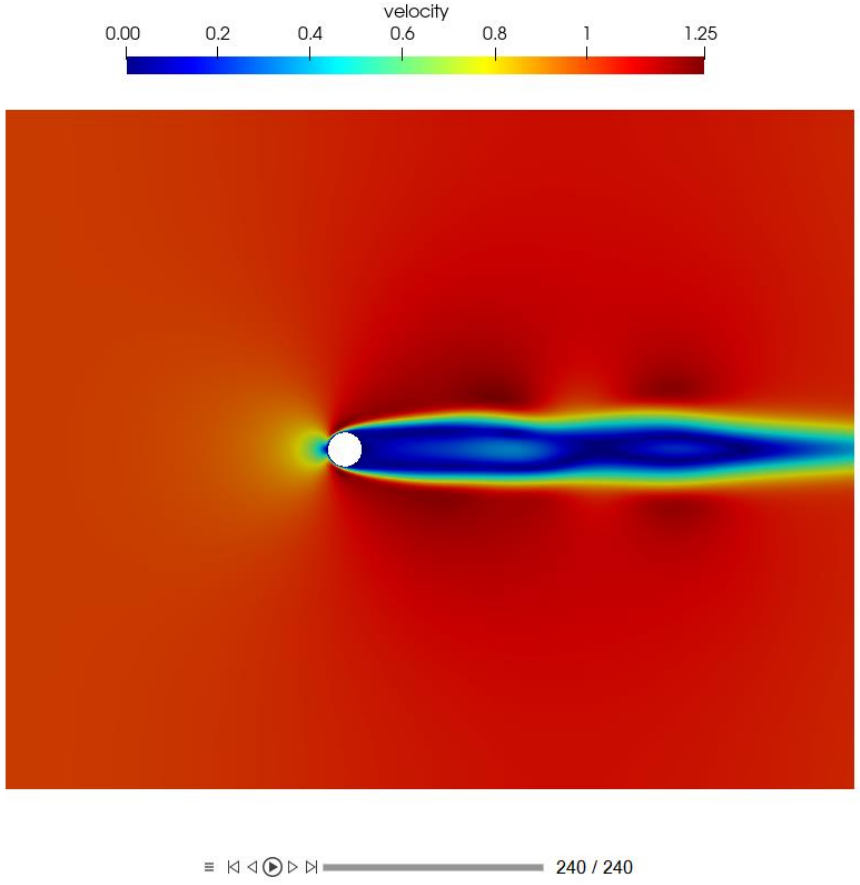
- To run a modified .inp a launcher file must be created.
- This is a text file in the form of the image shown on the slide, being:
  - "C:\Program Files\Altair\2022\hwcfdsolvers\acusolve\win64\binacurun.bat" the absolute path to the acurun.bat file of the version used.
  - VortexShedding_Trans the name of the problem associated with .inp
  - The number after -np marks the number of processors to use, 8 in this case.
- Save the final file with the .cmd extension.
- The simulation is then launched by clicking on the launcher from the explorer. The progress of the simulation can be followed in the usual way from Hyperworks CFD by loading the generated .log into the plot tool.

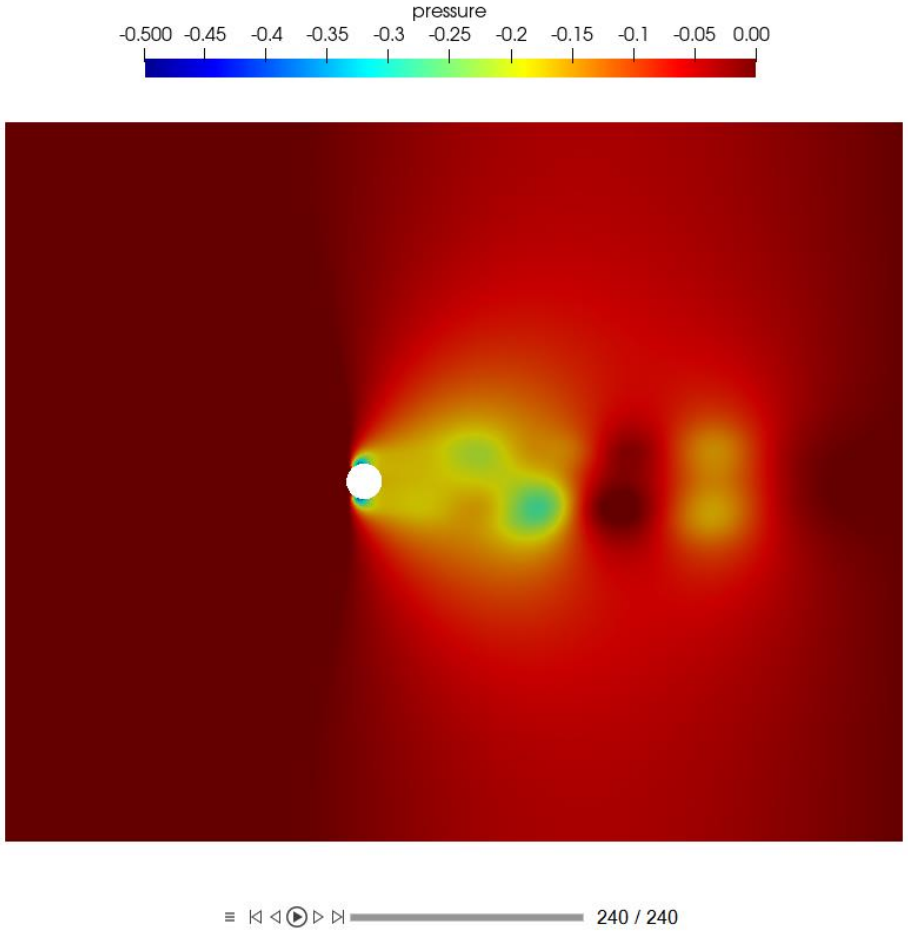
```
call  "C:\Program Files\Altair\2022\hwcfdsolvers\acusolve\win64\bin\acurun.bat" -pb VortexShedding_Trans -np 8

@ rem move *.dat .\06_Results
@ rem move *.odb .\06_Results
```



- When the simulation is finished, the .log can be opened from the post tab as usual.
- As it is a transient simulation and the output frequency was set to 1 time step, a frame selector appears at the bottom.
- The first frame corresponds to the solution of the steady state case. The animation ends up showing the expected evolution to a vortex shedding.

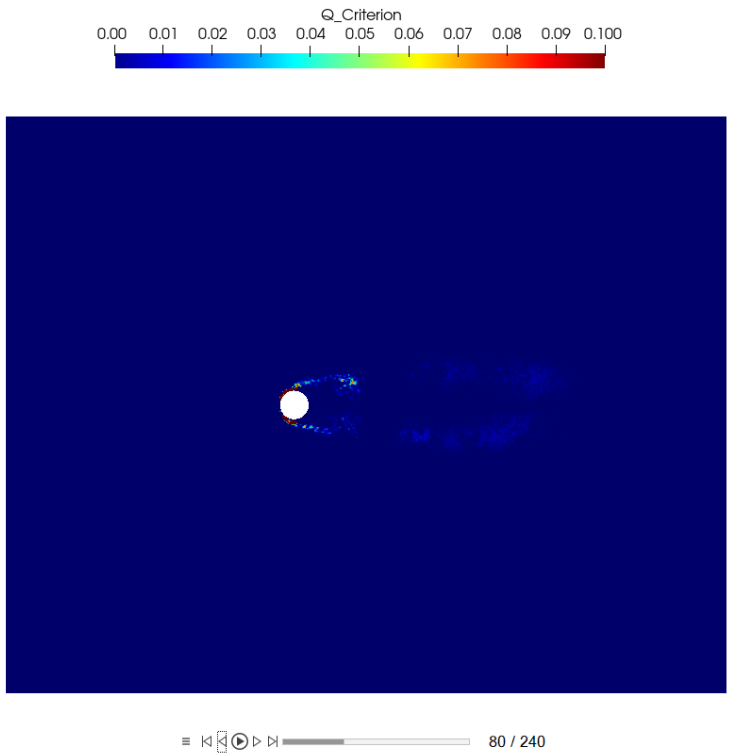


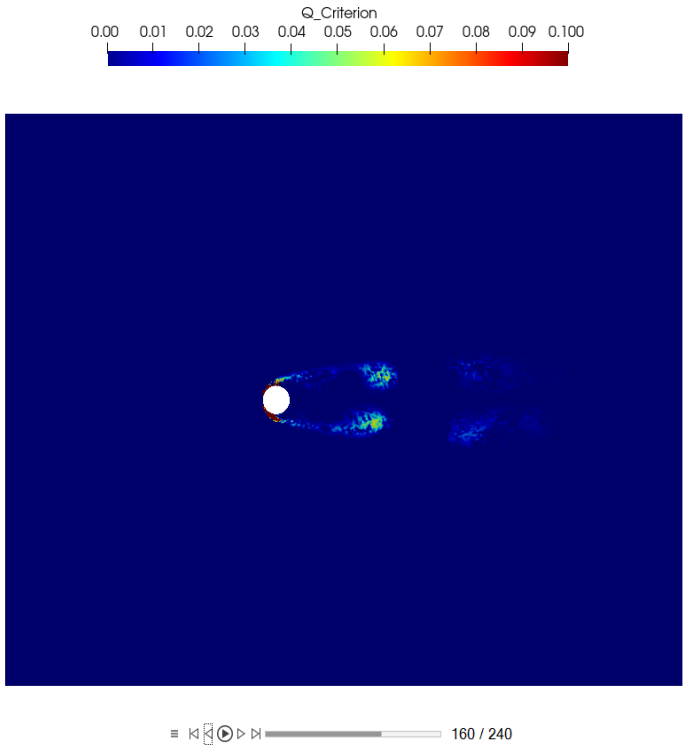


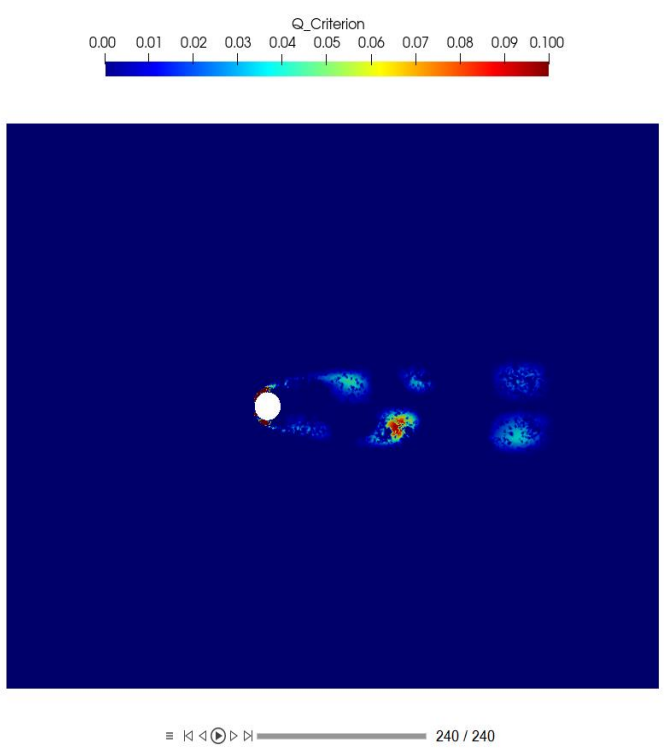
Post: Results

- To better see the vortices generated, a variable called Q_Criterion can be defined from Calculate: Derived Data Calculator using the velocity gradient as an auxiliary variable.
- The expressions are shown in the table on the right. The expression in version 2022.0 gives an error when used on a surface, so it is displayed on a central slice plane, where the definition is somewhat worse but allows us to visualize the eddies structure.

Name	Expression
gradVel	gradient(velocity)
Q_criterion	-0.5*(gradVel[:,0,0]*gradVel[:,0,0] + gradVel[:,1,0]*gradVel[:,0,1] + gradVel[:,2,0]*gradVel[:,0,2] + gradVel[:,0,1]*gradVel[:,1,0] + gradVel[:,1,1]*gradVel[:,1,1] + gradVel[:,2,1]*gradVel[:,1,2] + gradVel[:,0,2]*gradVel[:,2,0] + gradVel[:,1,2]*gradVel[:,2,1] + gradVel[:,2,2]*gradVel[:,2,2])



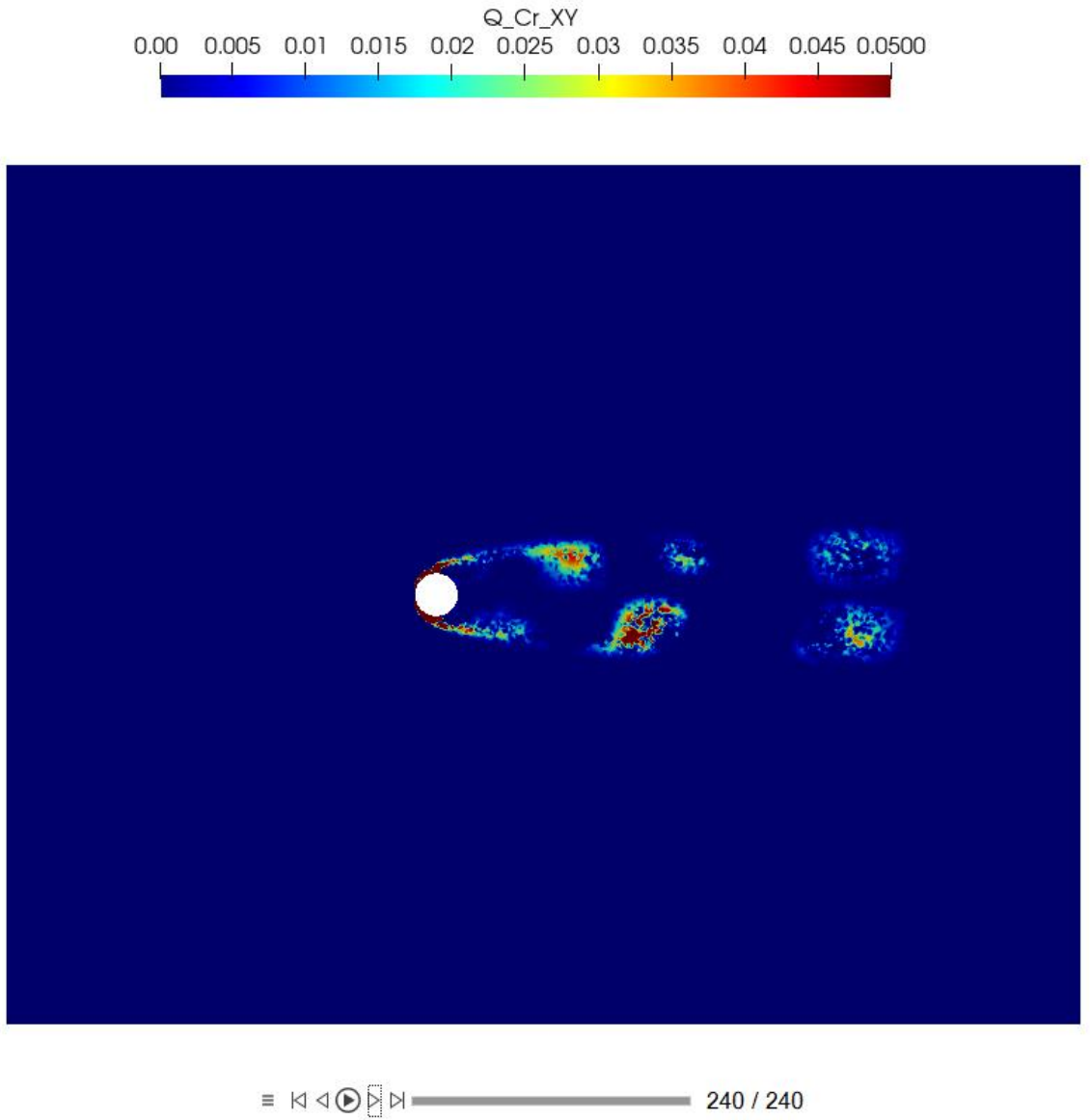




- Somewhat more visible results are obtained using the two-dimensional Q_Criterion expression, although it also fails on surfaces. It is defined as:

$$Q_Cr_XY = gradVel[:,0,0] * gradVel[:,0,0] - gradVel[:,1,0] * gradVel[:,0,1]$$

- The animation obtained by pressing the Play button shows how the generated vortices advance with the time steps.
- By changing the input velocity, other cases could be simulated depending on the Reynolds number.
- If the Reynolds number is raised over a certain limit, the turbulence model used in the simulation should be changed to one able to represent the unsteadiness of the flow.



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