Ennova Technologies

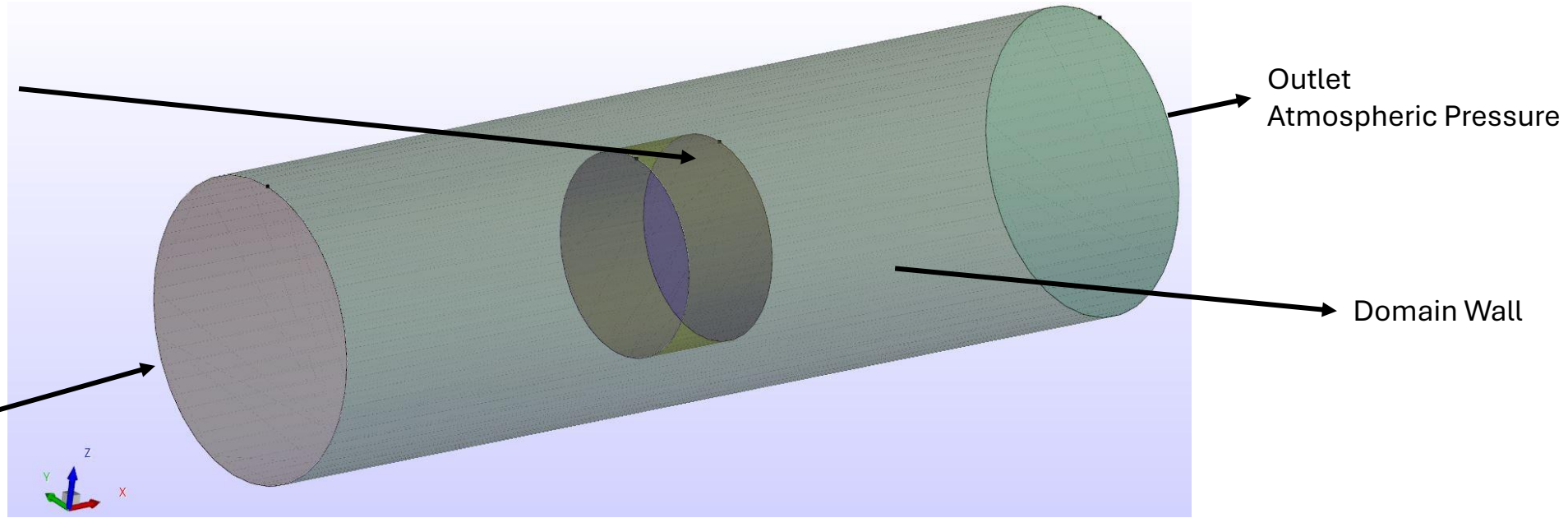
Ennova Release 1.11

Feature Explainer Video : **Scripting Ennova Via Python**

Boundary Conditions

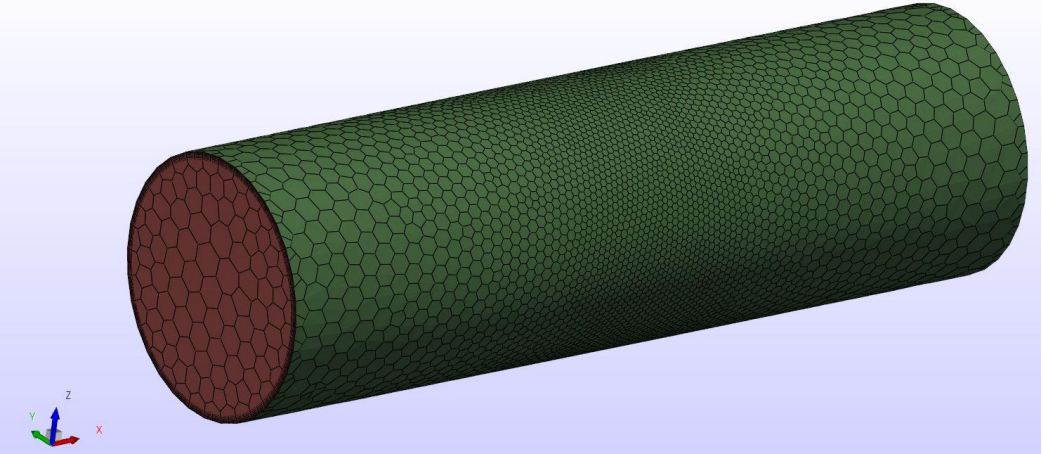
- Mean flow speed : 5 m/s
- Flow model : incompressible
- Mesh size : ~113K cells
- Fluid : Air
- Reference pressure : 1 atm

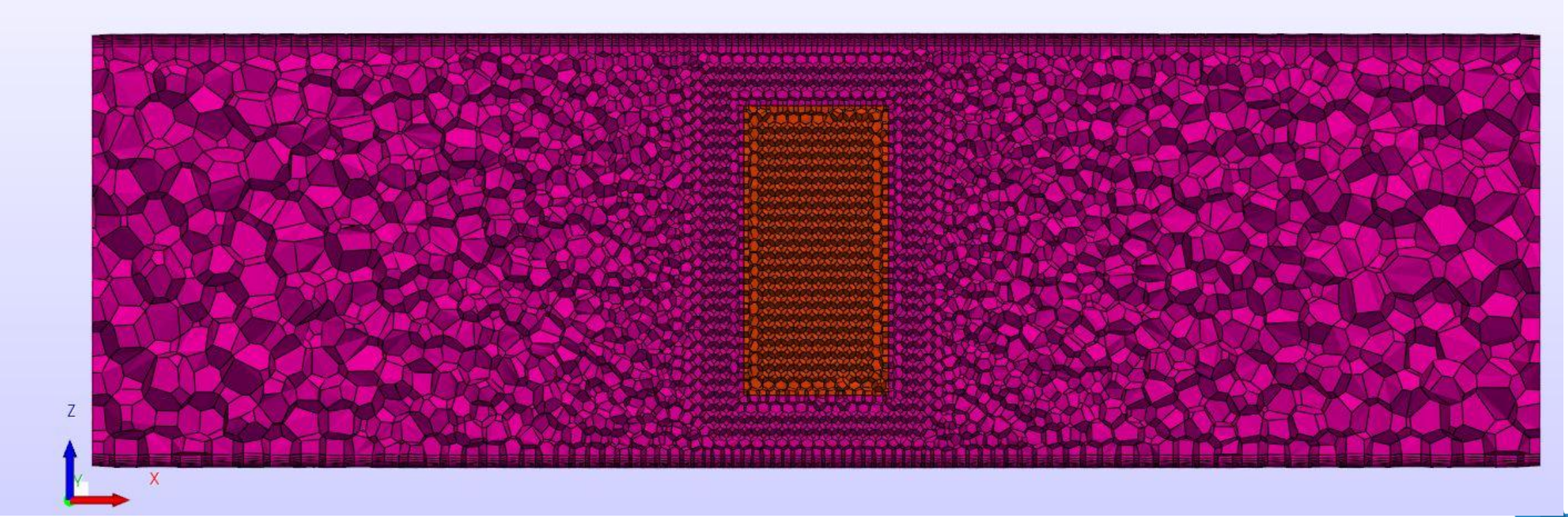
- Turbulence model : kOmegaSST
- Simulation type : Porous Media
- Time management : Steady-state



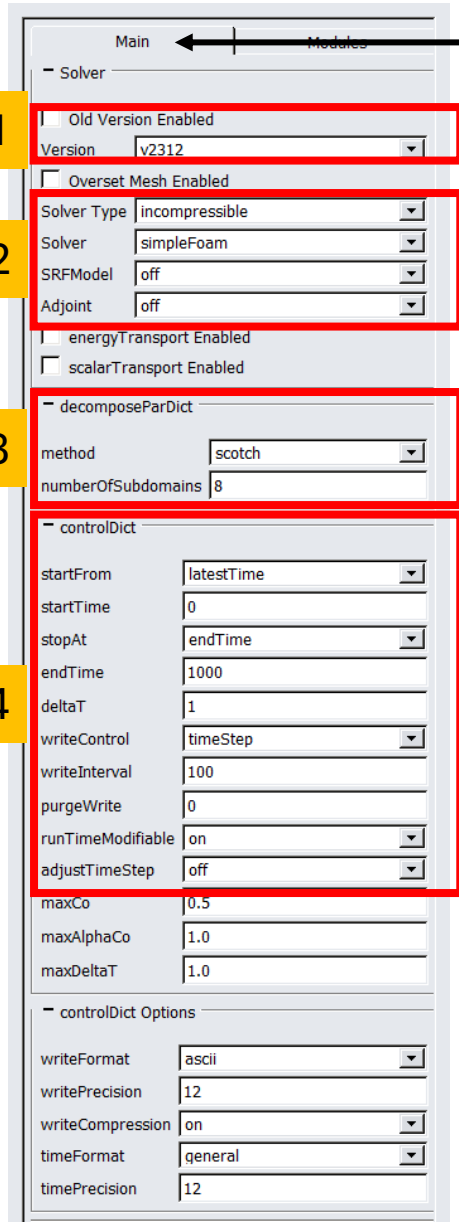
Mesh & Mesh Cut plane

Create uniform size density box around porous zone to have uniform mesh inside porous zone

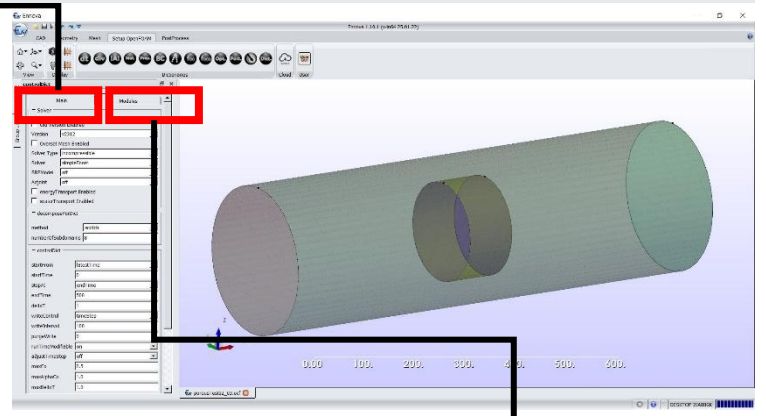


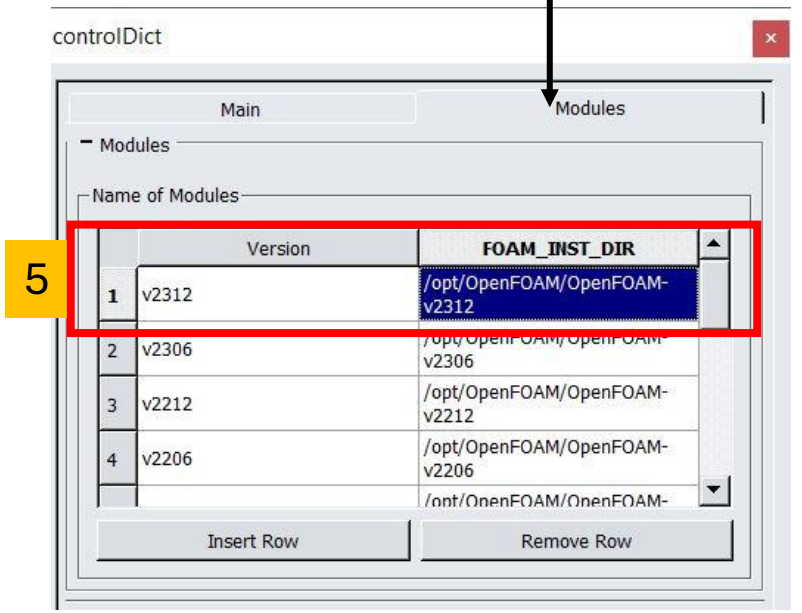


Solver Settings

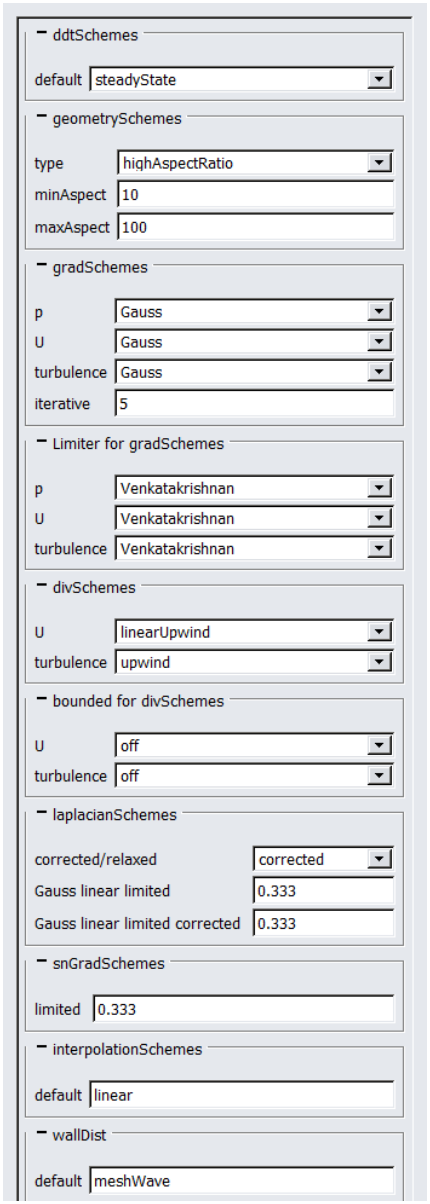


1. Version
   - Select latest version of OpenFOAM
   - Make sure same version is installed where case has to be simulated
   - Here it is v2312
2. Solver Settings
   - Type : incompressible
   - Solver : simpleFoam
     - Transient OpenFOAM solver
3. Decomposition
   - Method : Scotch
     - For equal number of cell distribution across the processors
   - Cores : 8
     - No. of cores available to run case in parallel
     - Optimum is 100K cells/core
4. Simulation Time Setup
   - StartFrom : latestTime
   - endTime : Till convergence is achieved generally 2000 to 4000 iterations
   - deltaT : 1 (Not required for steadystate
   - writeInterval : 100
   - runTimeModifiable : on
     - To edit controlDict file during simulation if required
5. Module
   - OpenFOAM installation path must be specified



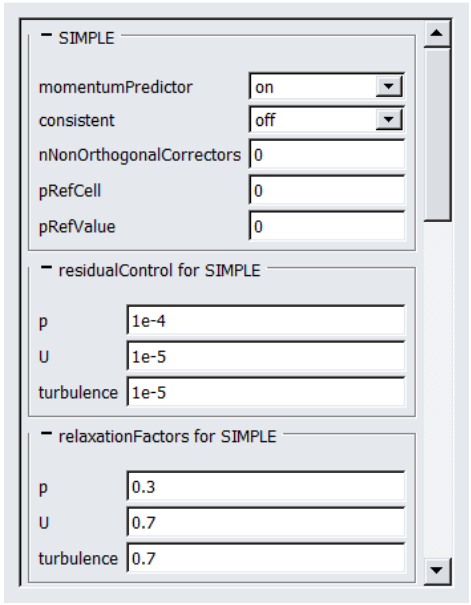


Schemes Settings

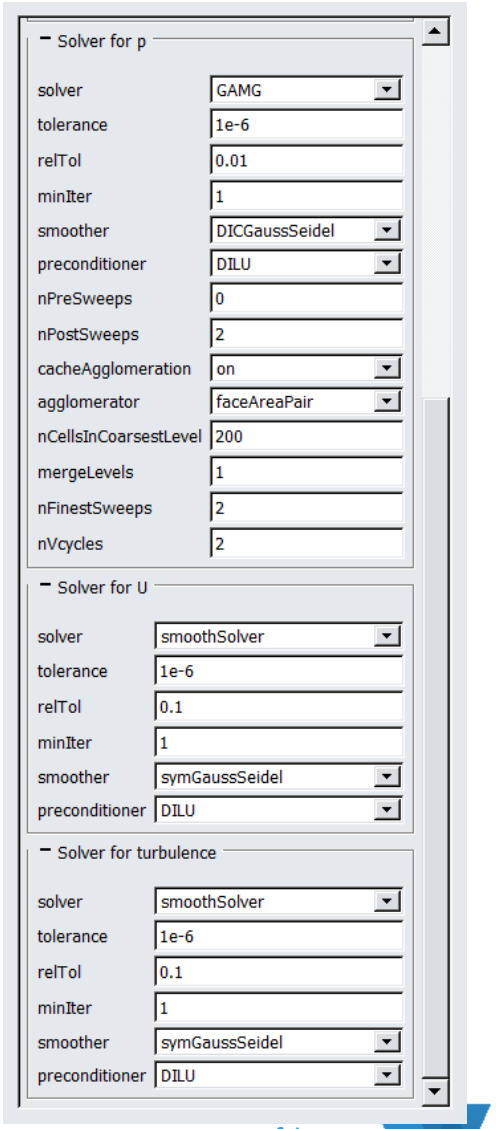


fvScheme

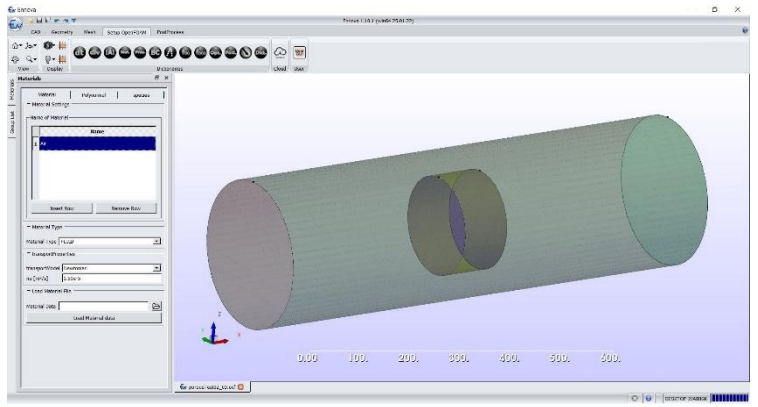
- fvScheme and fvSolution
  - Schemes and simulation convergence tolerance, relaxation factors etc.
- ***Keep it default.***

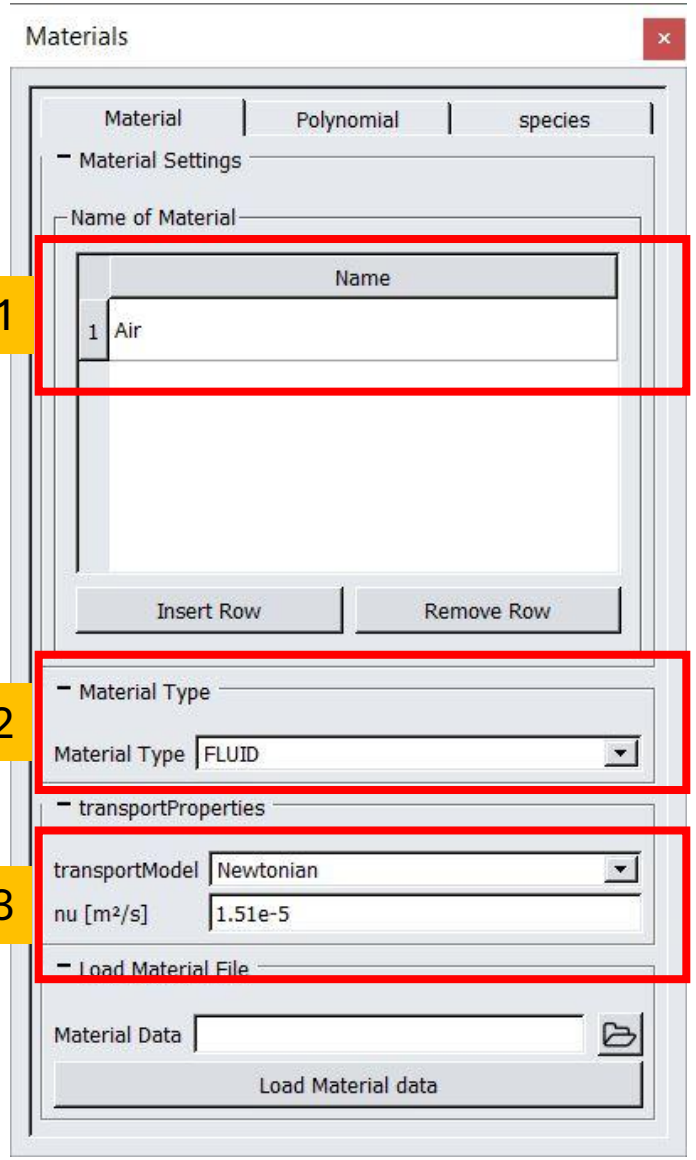


fvSolution



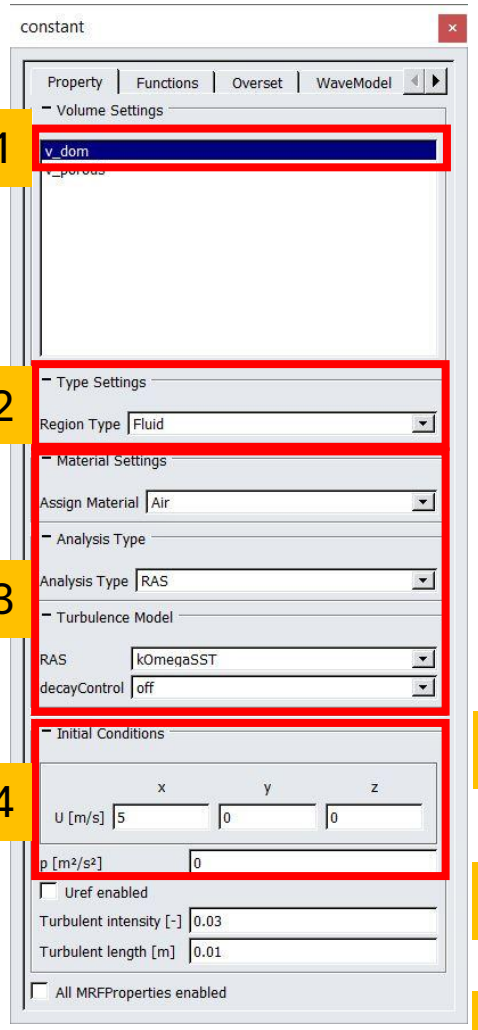
Material Settings





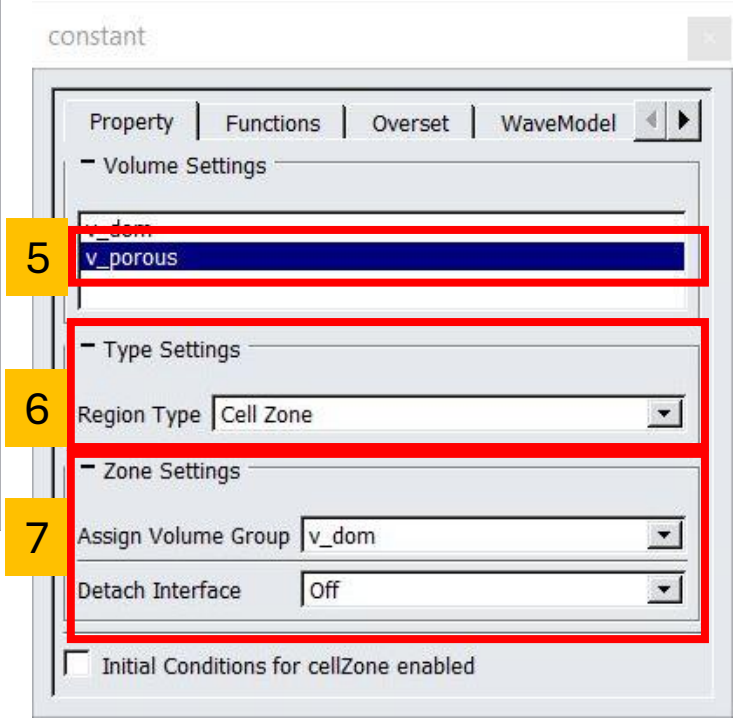
1. Material Name
   - Default is air
   - Double click the Air to update the name
2. Material
   - Material Type : Fluid (Default)
3. transPortProperties
   - transportModel : Newtonian (Default)
   - Nu (Kinematic Viscosity) : $1.51e^{-7}(m^2/s)$

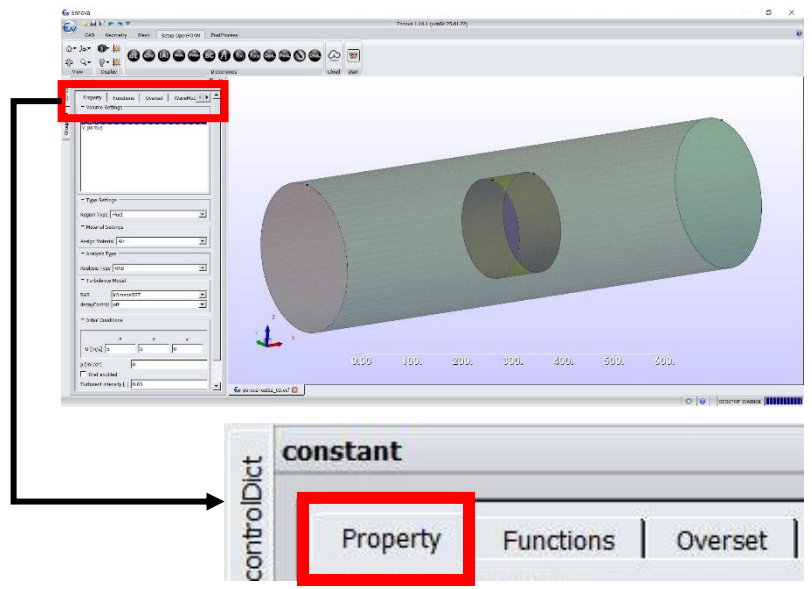
Property Settings



Volume Settings - Property

- Primary Zone : fluid domain
  1. Domain : v_dom
  2. Type Settings
     - Region Type : Fluid (Default)
  3. Material Settings
     - Keep it default
  4. Initial Conditions
     - U (m/s) : 5, 0, 0
     - Supply inlet velocity value

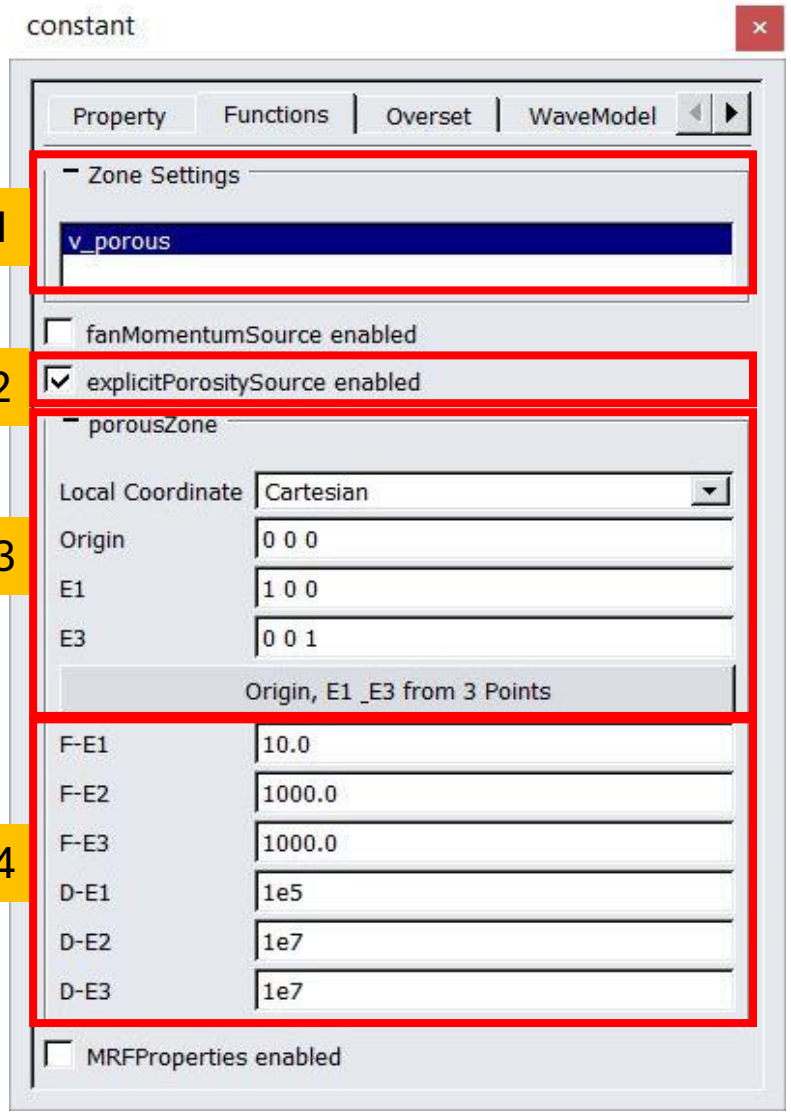


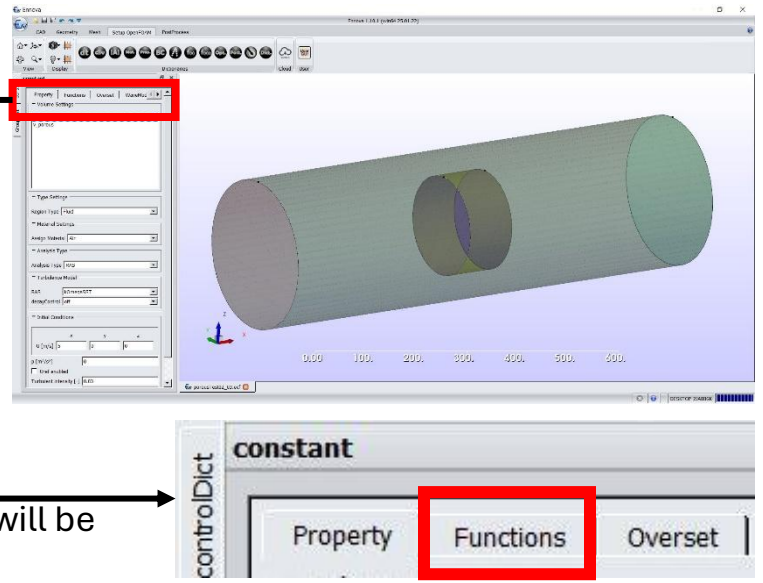


Volume Settings - Property

- Secondary Zone: Porous
  5. Domain : v_porous
  6. Type Settings
     - Region Type : Cell Zone
       - Select it from drop down menu
  7. Zone Settings
     - Assign volume Group : v_dom
       - Assign it with the surrounding zone. In this case it is primary fluid domain v_dom
     - Detach Interface : Off
       - Keep detach interface OFF

Property Settings

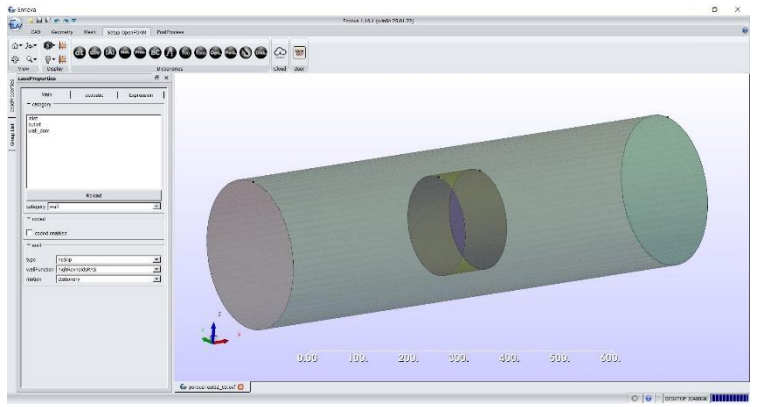




Volume Settings - Functions

- Volume domain which is considered as a cellZone will be available here
  1. Zone Settings : v_porous
  2. explicitPorositySource enabled
     - Check this option for Porous zone
  3. PorousZone
     - Local Coordinate : Cartesian
     - Origin : 0, 0, 0 (As per porous zone location)
     - E1 & E3
       - The local coordinate system is specified by the unit vectors e1 and e3 in the section coordinateSystem.
       - The remaining axis is computed as the cross product e1xe3
  4. DarcyForchheimer Coefficients
     - https://openfoamwiki.net/index.php/DarcyForchheimer

Boundary Conditions



caseProperties

Main | acoustic | Expression

category

inlet
outlet
wall_dom

Reload

category: inlet

coded

coded enabled

inlet

type: subSonic

flowSpecification: fixedVelocity

	x	y	z
U [m/s]	5	0	0

Expression enabled

inlet(Turbulence)

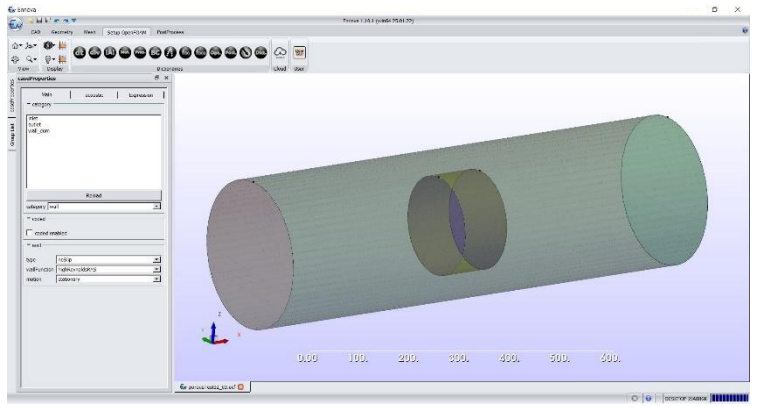
inletOutlet

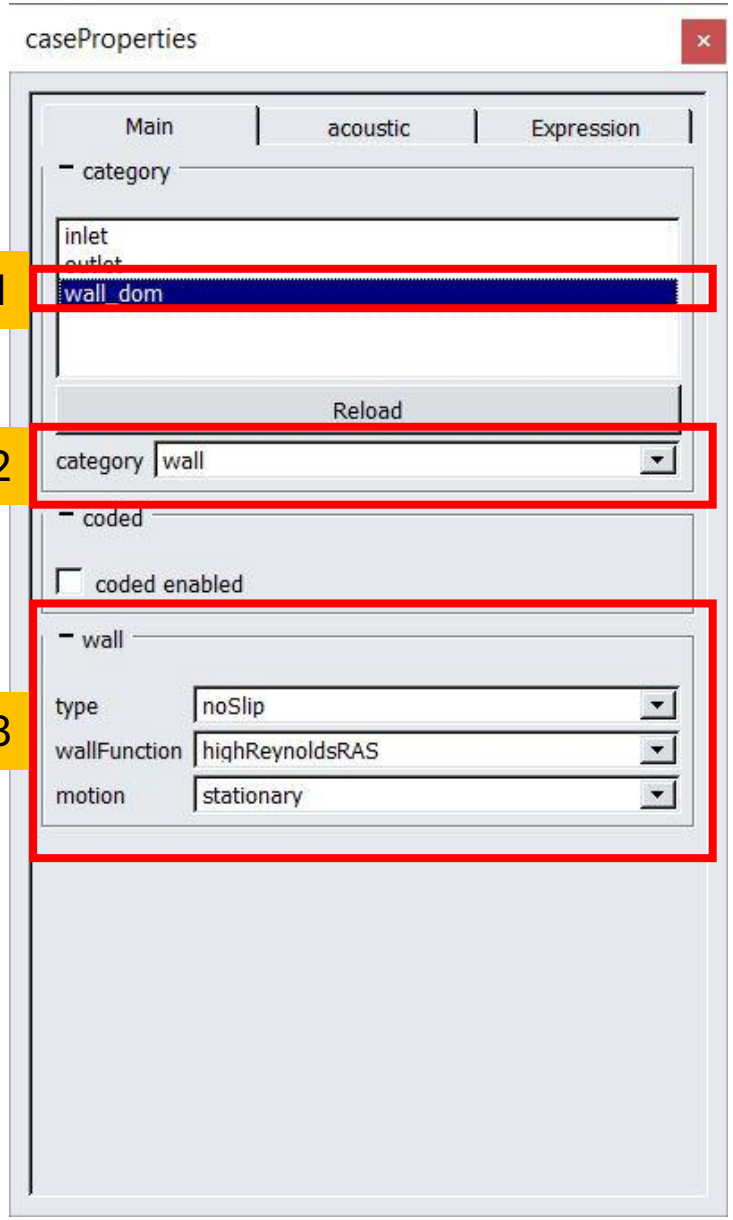
Turbulent intensity [-]: 0.03

Turbulent length [m]: 0.01

- To make sure that porous setting is correct, porous zone group/part name should not be available in the BC section

Boundary Conditions

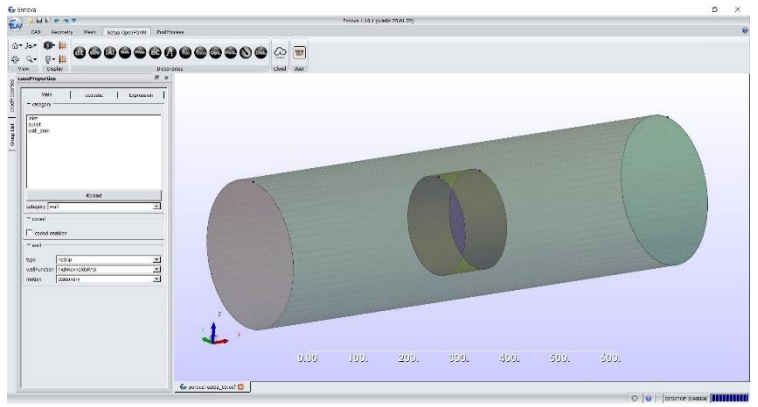


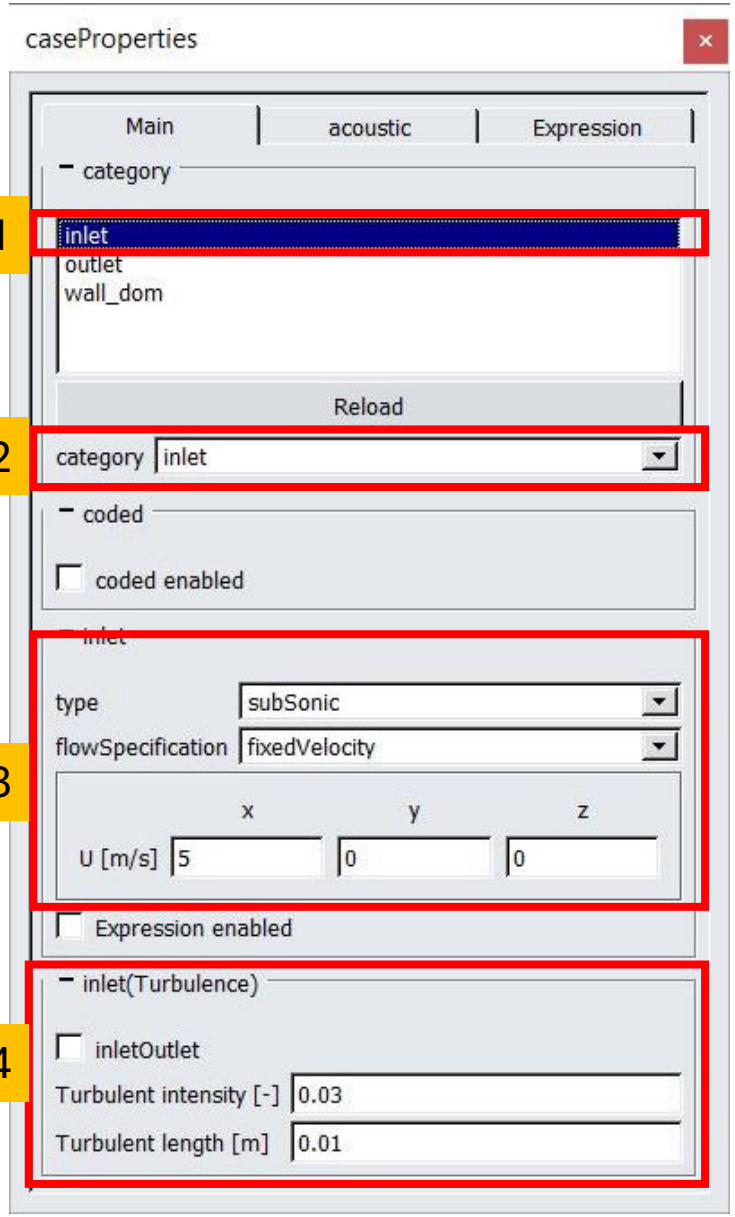


Boundary Conditions - Wall

1. Select all parts which are stationary walls
2. Category : wall
   - Select BC type as wall for all parts except inlet and outlet
3. Settings for wall type BC
   - Keep defaults
   - Make sure that motion is selected as stationary

Boundary Conditions

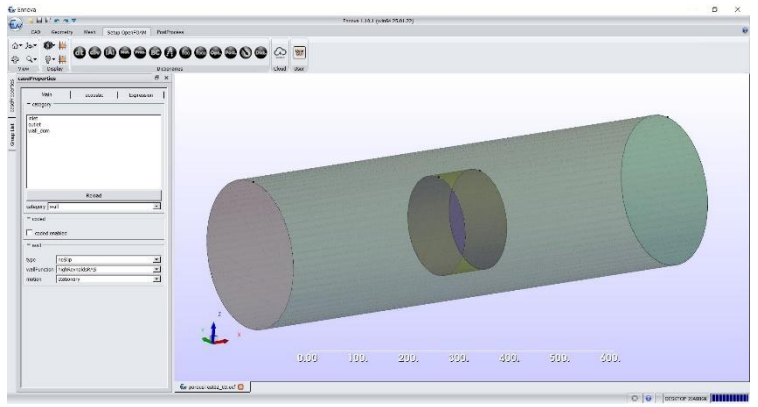


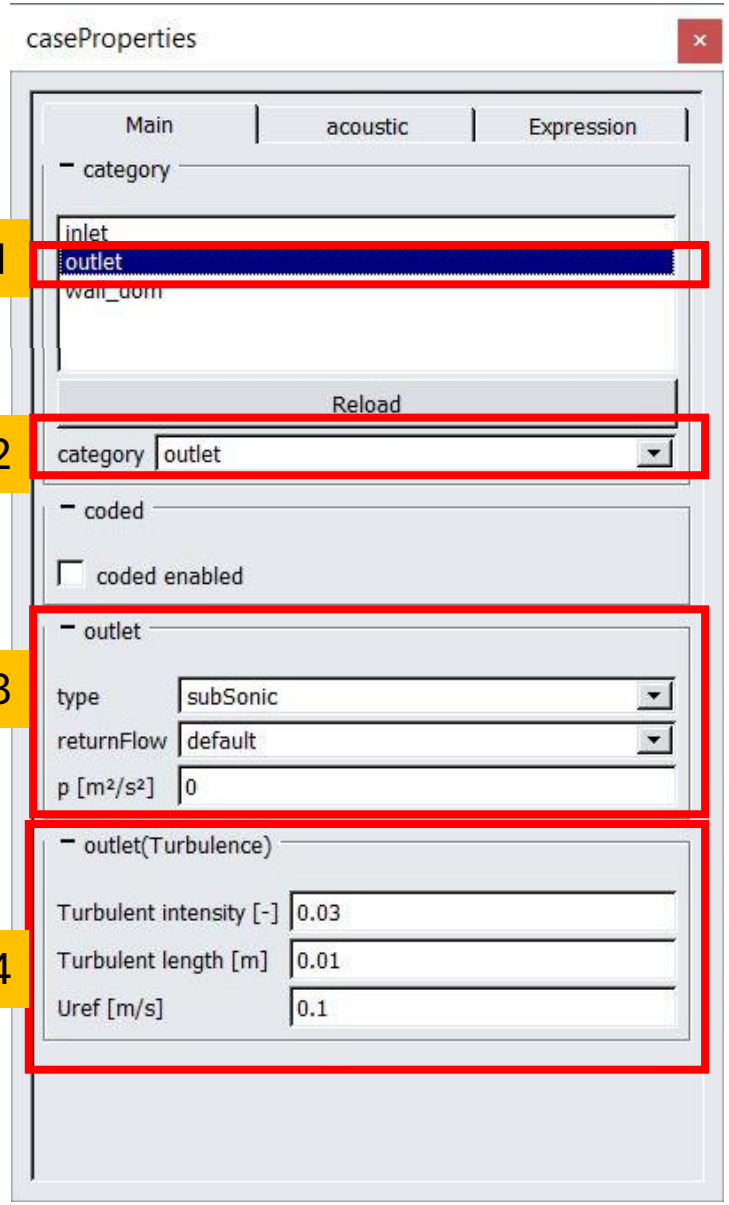


Boundary Conditions - inlet

1. Select part which is inlet to the domain
   - Here it is inlet
2. Category : inlet
   - Select BC type as inlet for inlet
3. Settings for inlet type BC
   - Type : subsonic (default)
   - flowSpecification : select fixedVelocty from drop down
   - Supply inlet velocity value
     - U (m/s) : 5 (in positive X-direction)
4. Turbulence Settings
   - Calculate turbulence intensity and length scale and update values in respective boxes

Boundary Conditions

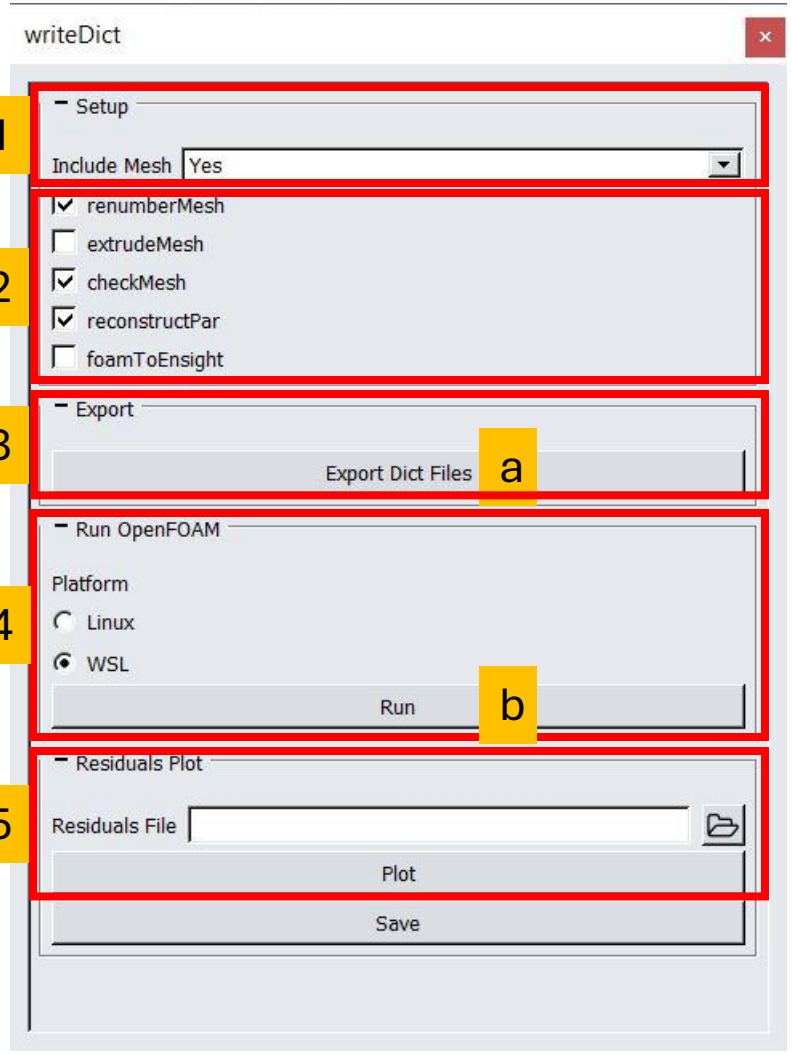




Boundary Conditions - outlet

1. Select part which is outlet to the domain
   - Here it is outlet
2. Category : outlet
   - Select BC type as inlet for outlet
3. Settings for outlet type BC
   - Type : subsonic (default)
   - returnFlow : default
   - Pressure (m^2/s^2) : 0 (Gauge pressure – atmospheric pr)
4. Turbulence Settings
   - Calculate turbulence intensity and length scale and update values in respective boxes

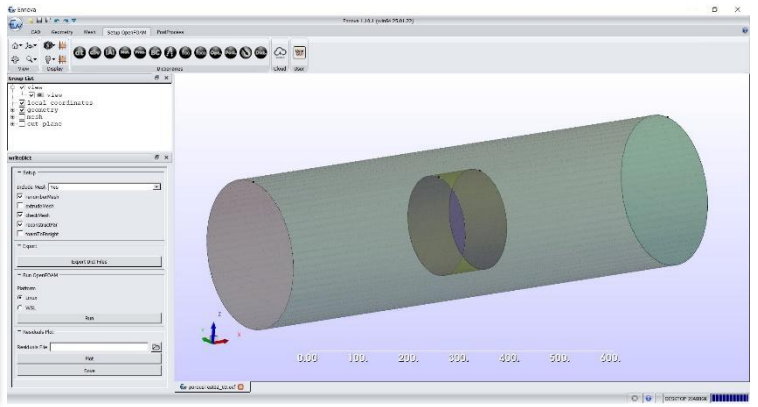
Export OpenFOAM Files



There are two ways of running OpenFOAM case with Ennova,

a. Export Dict Files
   - Exports OpenFOAM case structure files
   - 0, constant, system and Allrun

b. Run
   - Running on WSL directly from Ennova



Setup

1. Include Mesh : Yes
   1. In Ennova we have generated polyhedral volume mesh for OpenFOAM case
2. Optional OpenFOAM operations to be performed before and after the simulation (check the boxes checked in this case)
   - renumberMesh : To reduce memory requirement for each step
   - checkMesh : To check volume mesh quality
   - reconstructPar: Reconstruct result data after completion of a simulation
3. Export File (Explained above)
4. Run OpenFOAM
   - Linux : For running case on Linux (cluster) separately
   - WSL : For running case on from Ennova on local machine
     - Make sure WSL is installed on windows machine
5. Residual Plot
   - Read residual file caseDict/postProcessing/residuals/0/solverInfo.dat
     - caseDict – OpenFOAM case directory name

Update in OpenFOAM File (If required)

- 0/omega

```
{
    type    omegaWallFunction;
    blending    binomial2;
    value    uniform 6.7082e-05;
}
```

Instead of ***binomial2*** there should be ***binomial*** only

Thank You