Ennova Technologies

Propeller Test Case

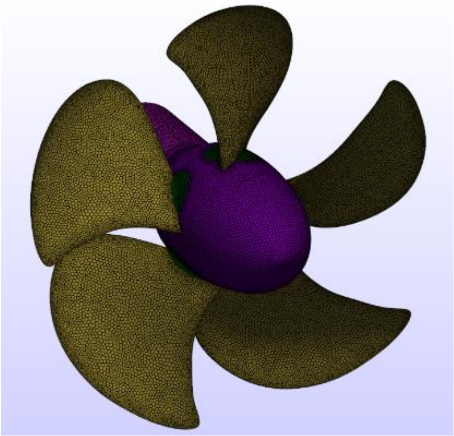
OpenFOAM case setup with Ennova

December 27th 2024

Ennova Technologies

ennova-cfd.com

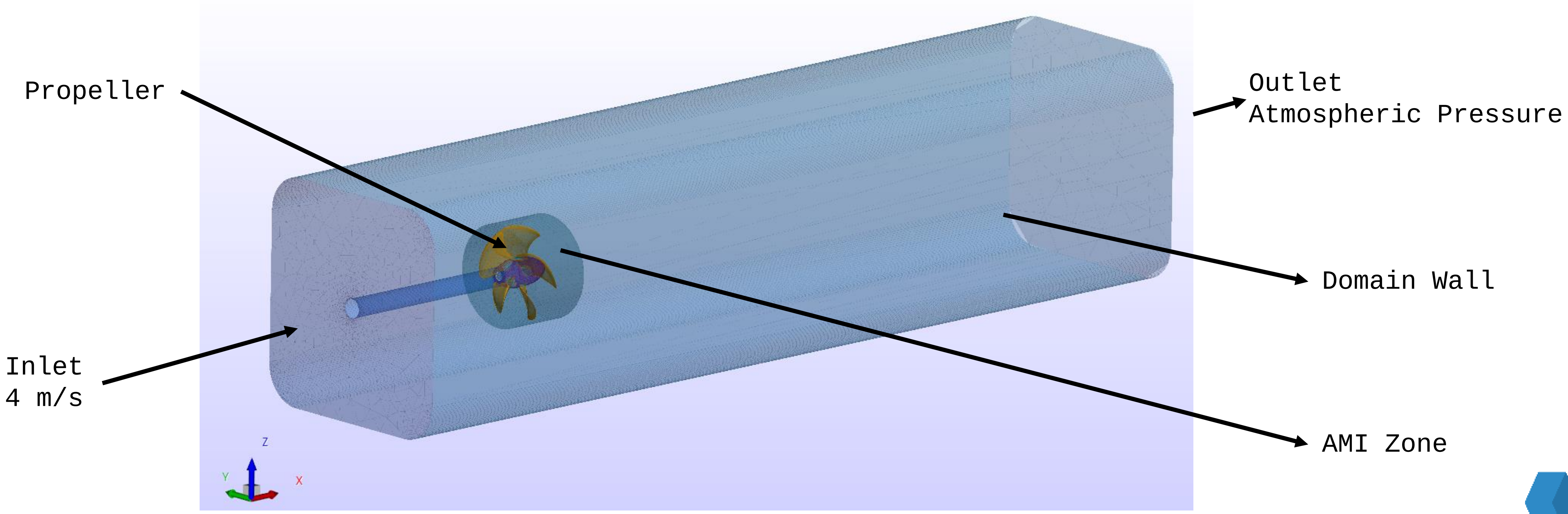
Case : AMI - Transient Potsdam Propeller



Boundary Conditions

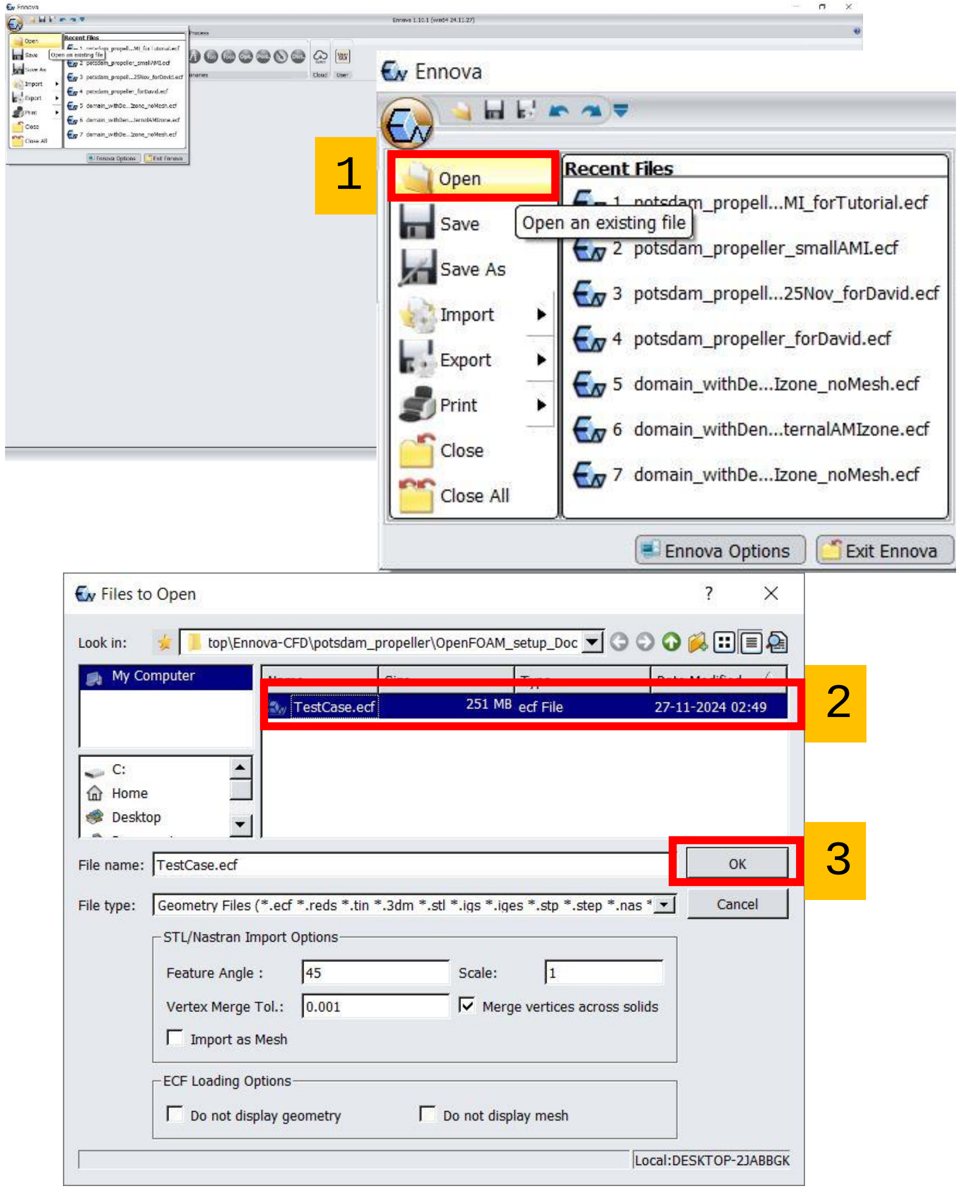
- Mean flow speed : 4 m/s
- Rotation speed : 900 RPM
- Flow model : incompressible
- Mesh size : 1.53M cells
- Simulation point : 1 (J = 1.0667)
- Fluid : Water
- Reference pressure : 1 atm

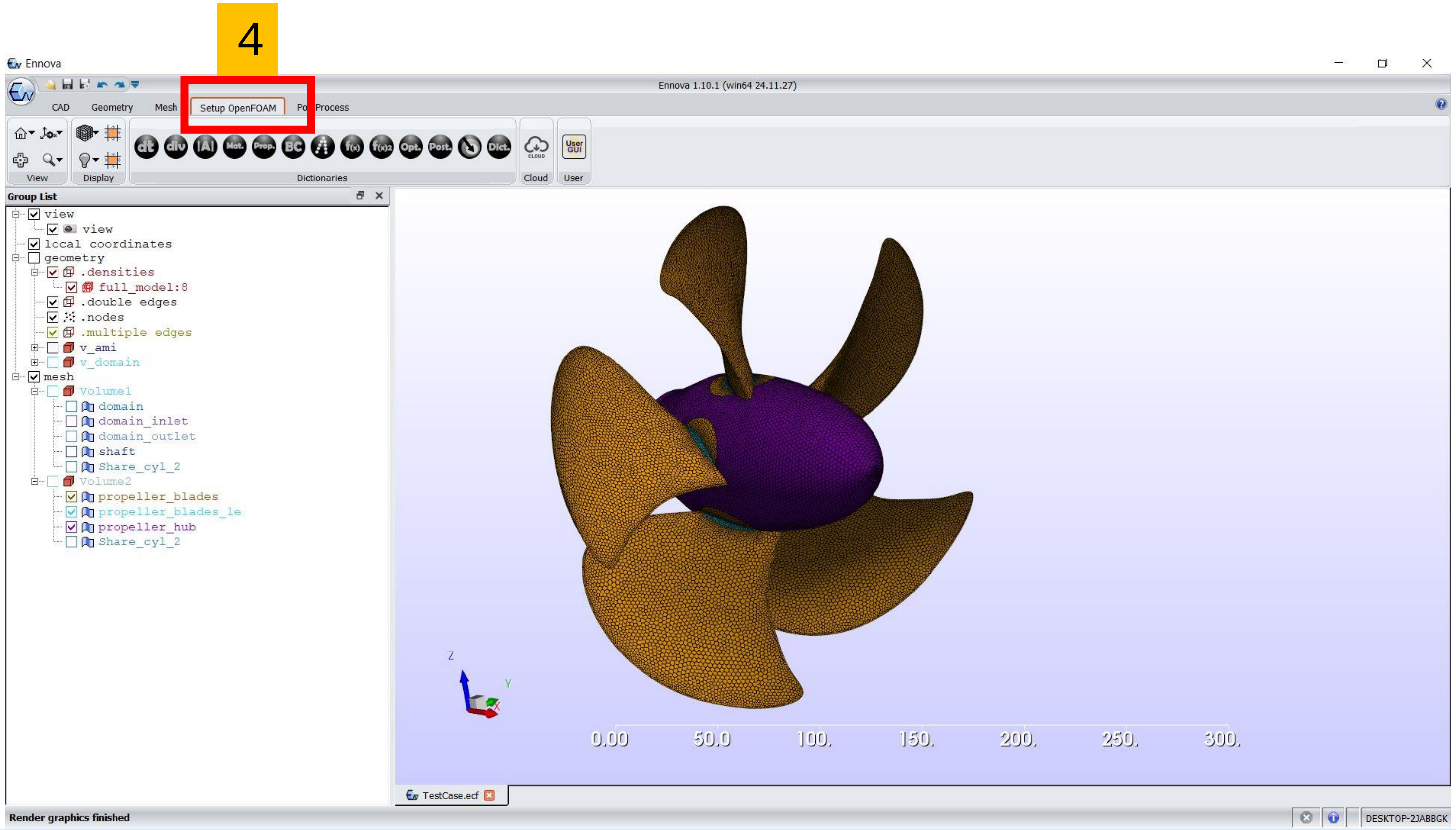
- Reference density : 997.71 kg/m 3
- Dynamic viscosity : 9.559 × 10-4 Pa·s
- Turbulence model : kOmegaSST
- Simulation type : Propeller
- Time management : Transient
- Rotational Direction : (-1, 0, 0)



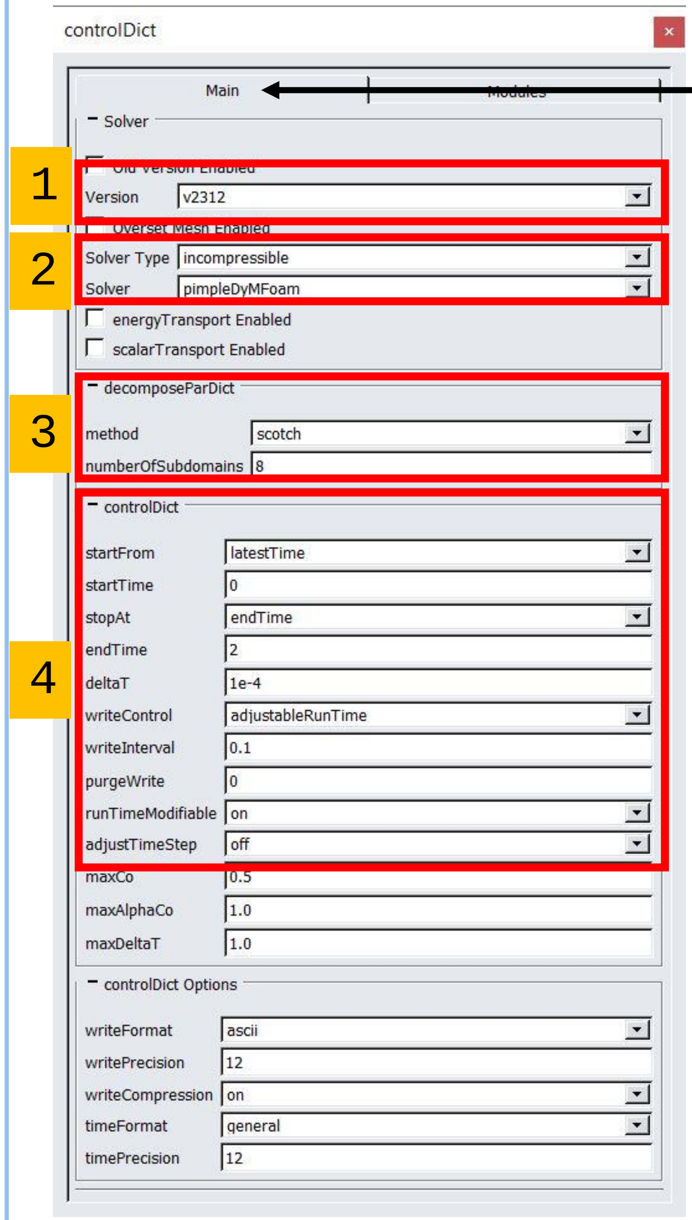
Reading ECF (Volume Mesh File)

1. Open File
2. Select File
3. Ok to load the file
4. Click on Setup OpenFOAM

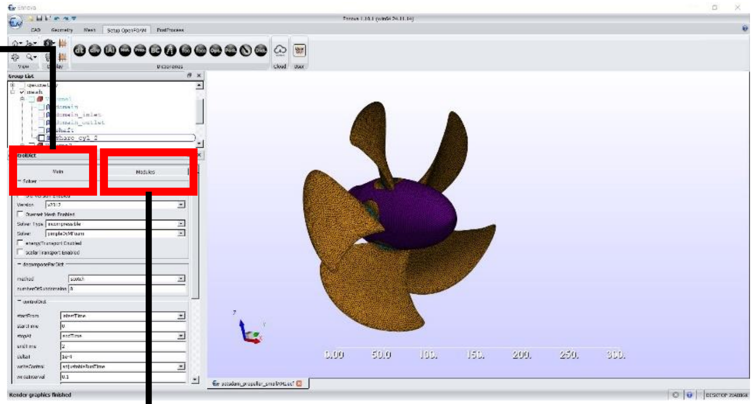


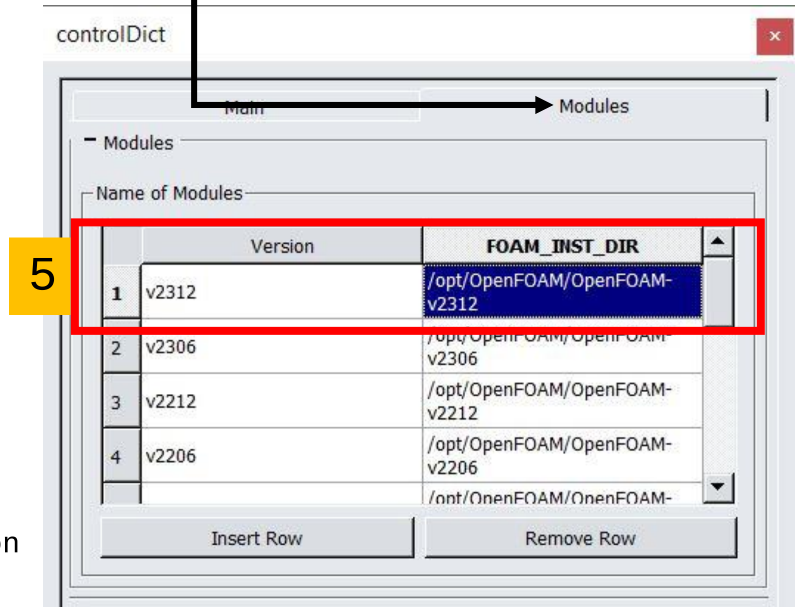


Solver Settings

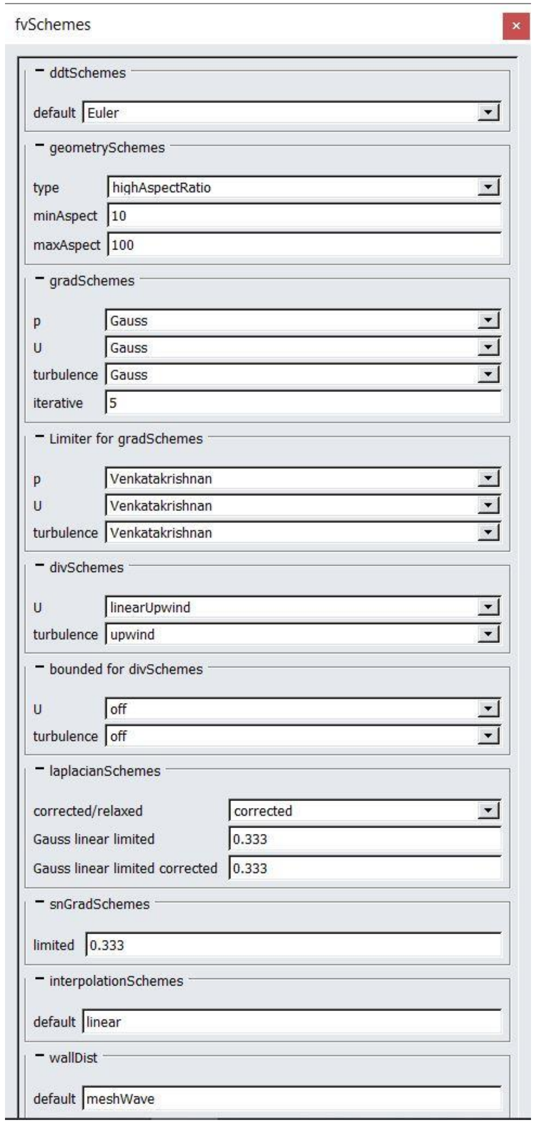


1. Version
   - Select latest version of OpenFOAM
   - Make sure same version is installed where case has to be simulated
   - Here it is v2312
2. Solver Settings
   - Type : incompressible
   - Solver : pimpleDyMFOAM
     - Transient OpenFOAM solver
3. Decomposition
   - Method : Scotch
     - For equal 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 : 2.0 (in seconds)
   - deltaT : 0.0001
   - writeInterval : 0.1 (in seconds)
   - runTimeModifiable : on
     - To edit controlDict file during simulation if required
5. Module
   - OpenFOAM installation path must be specified



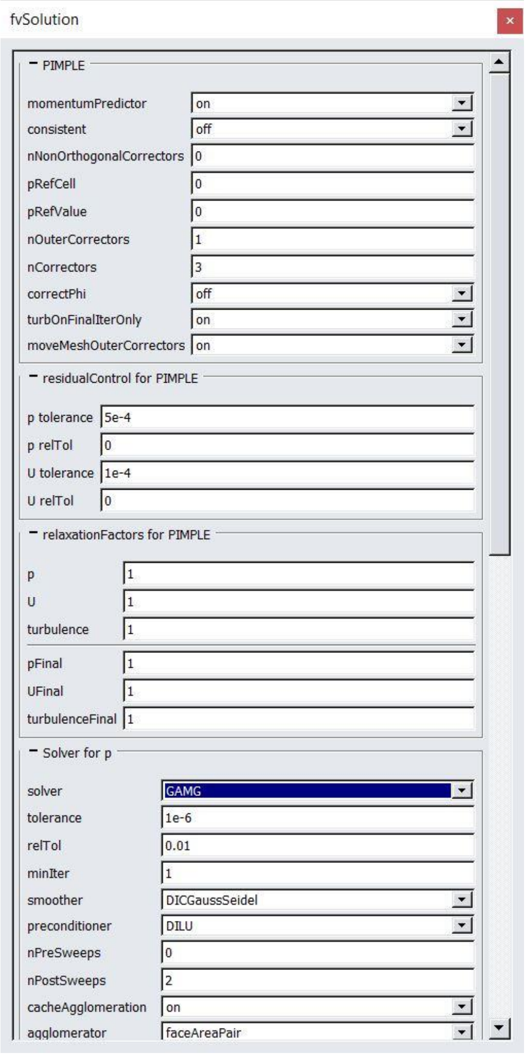


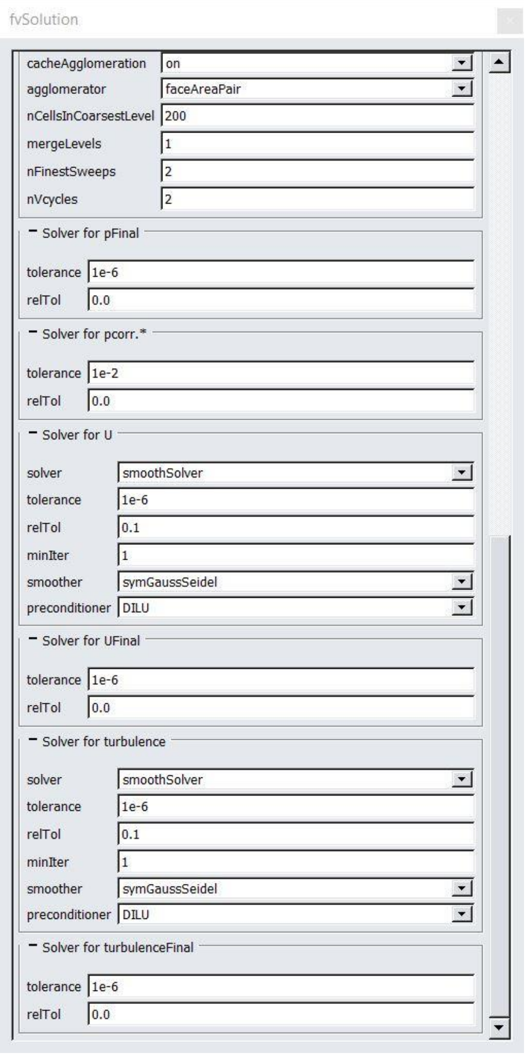
Schemes Settings



fvScheme

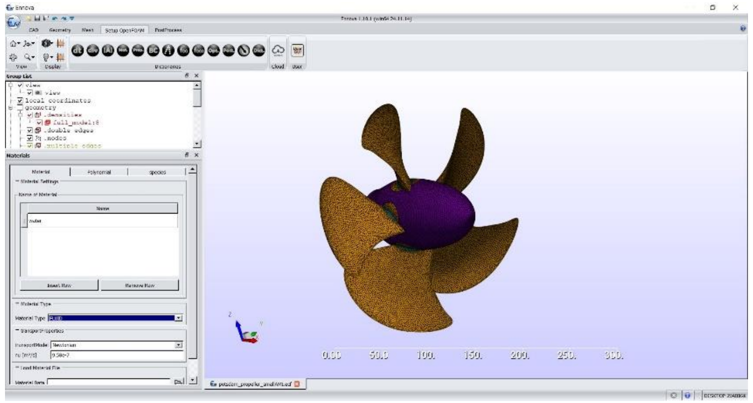
- fvScheme and fvSolution
  - Schemes and simulation convergence tolerance, relaxation factors etc.
- ***Keep it default.***

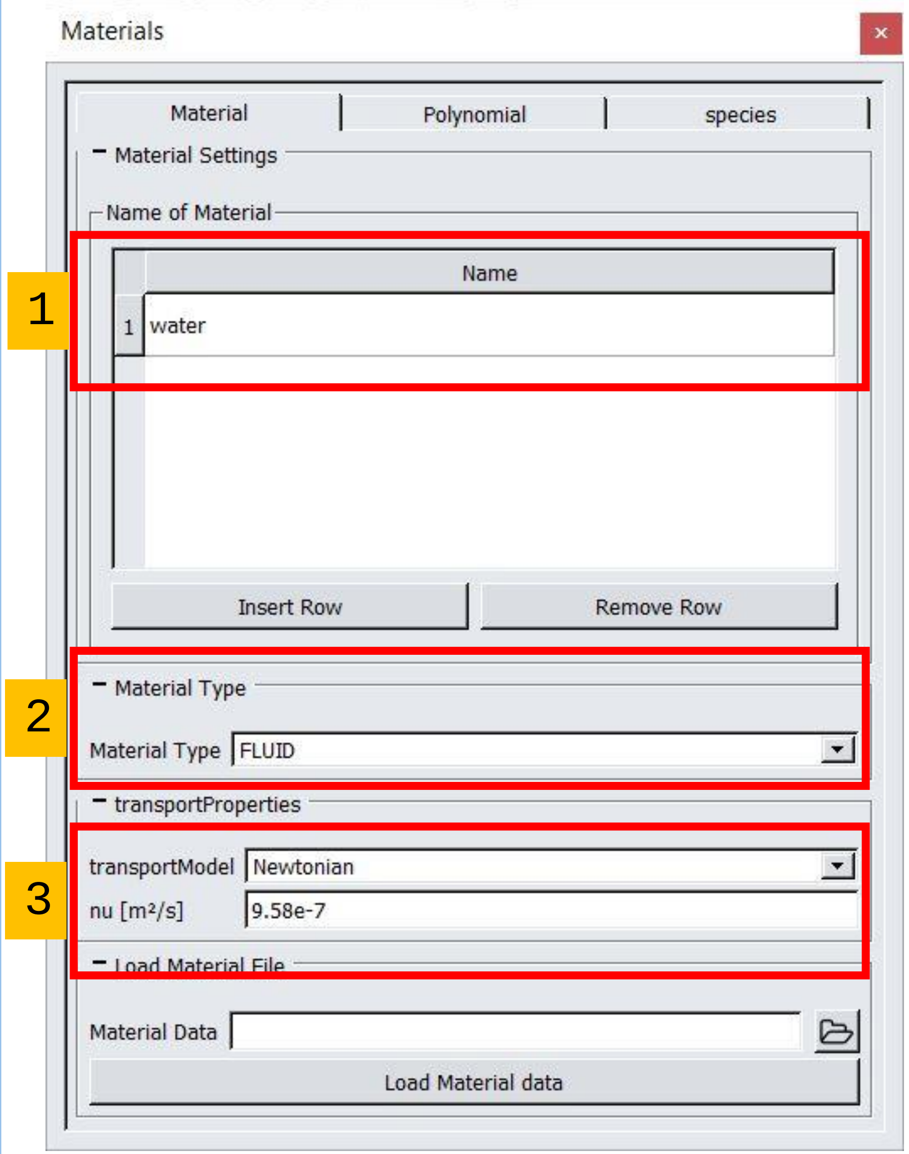




fvSolution

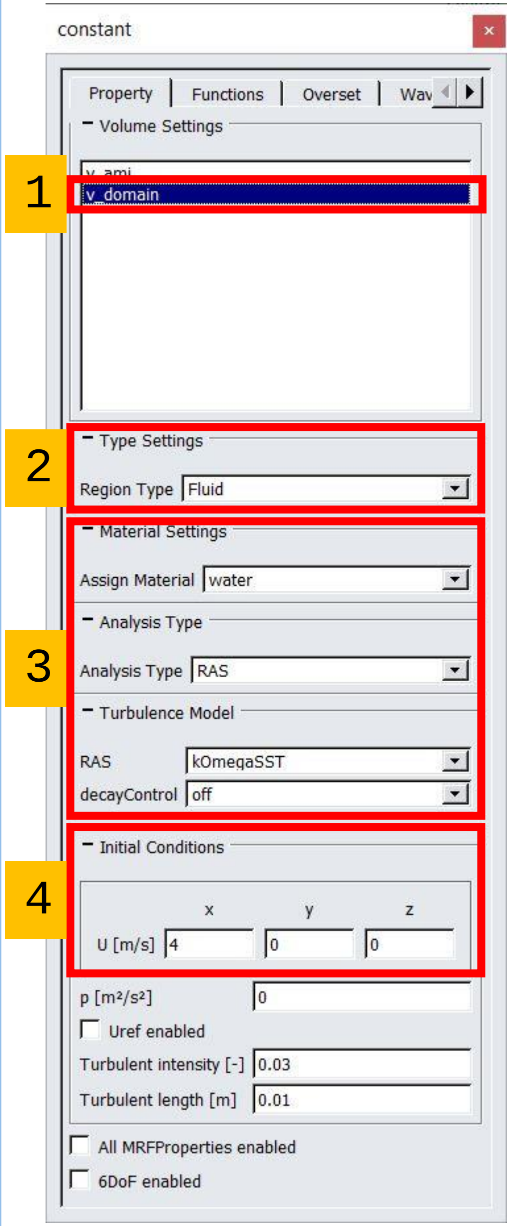
Material Settings





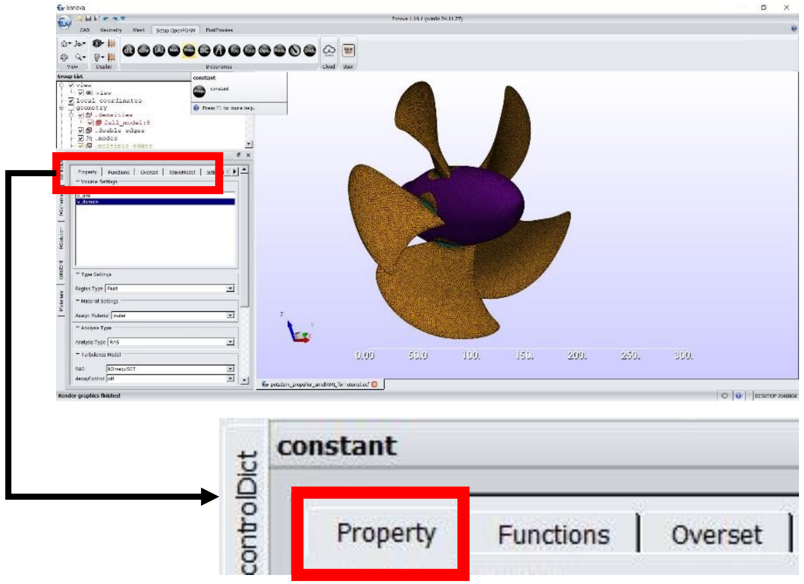
1. Material Name
   - Default is air
   - Double click the Air to update the name
2. Material
   - Material Type : Fluid (Default)
3. transPortProperties
   - transportModel : Newtonian (Default)
   - Nu (Kinematic Viscosity) : $9.58e^{-7}(m^2/s)$ updated value for water

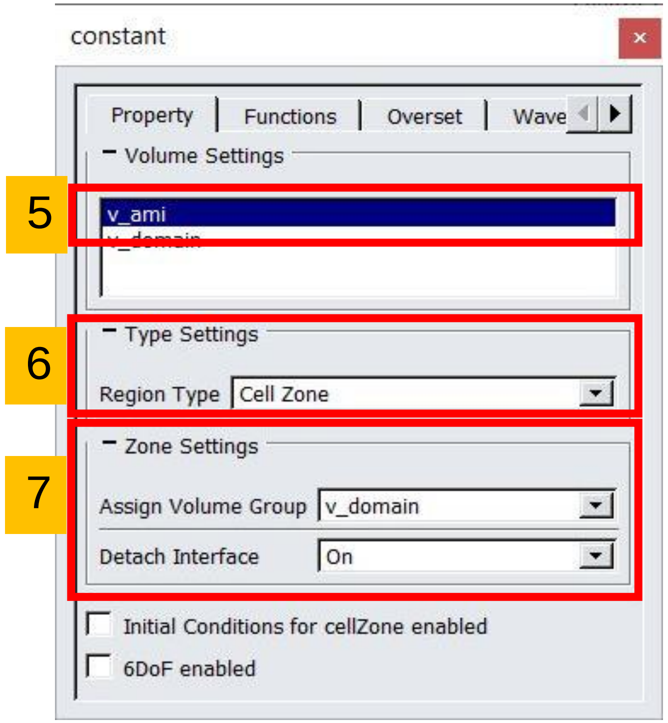
Property Settings



Volume Settings - Property

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

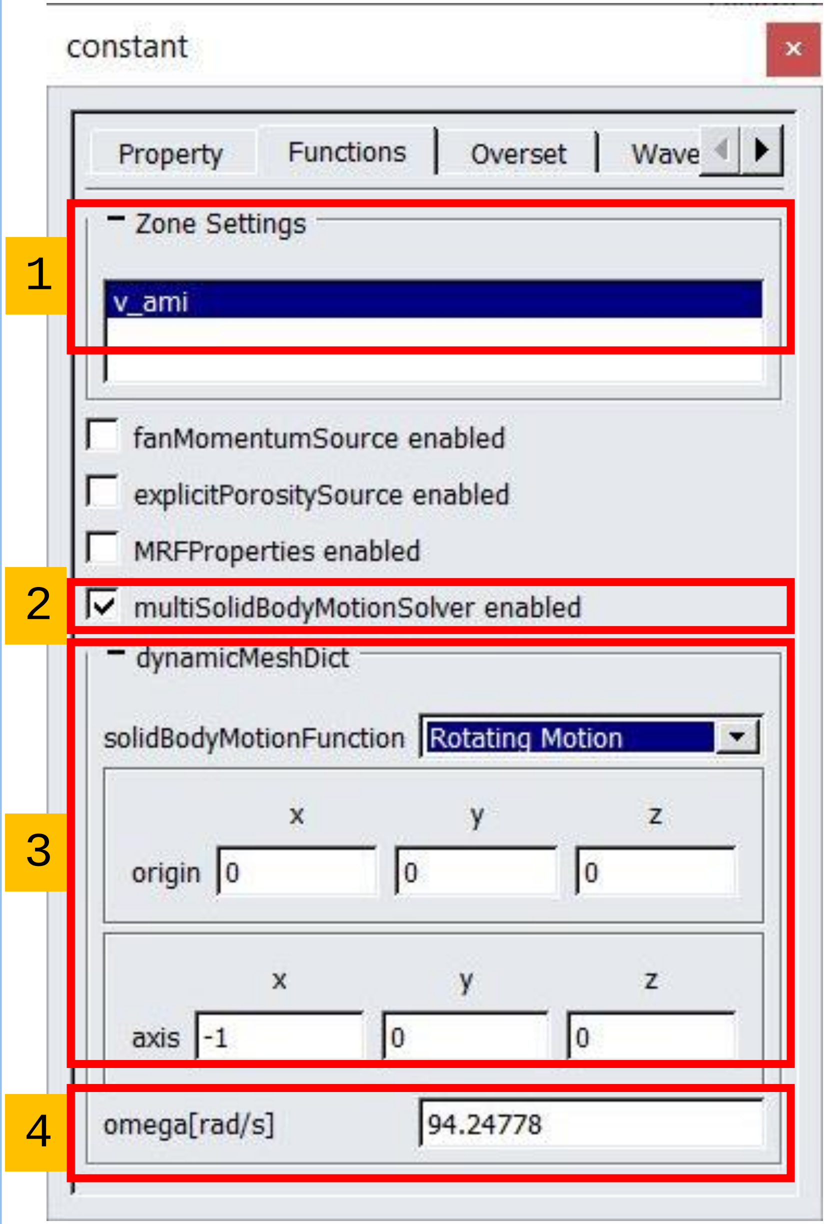


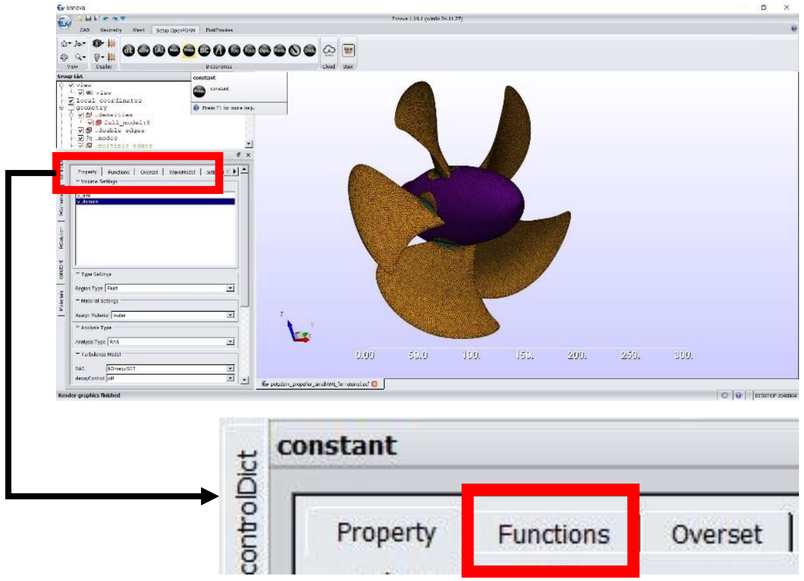


Volume Settings - Property

- Secondary Zone: AMI
  5. Domain : v_ami
  6. Type Settings
     - Region Type : Cell Zone
       - Select it from drop down menu
  7. Zone Settings
     - Assign volume Group : v_domain
       - Assign it with the surrounding zone. In this case it is primary fluid domain v_domain
     - Detach Interface : On
       - Keep detach interface ON because this cellZone is rotating

Property Settings

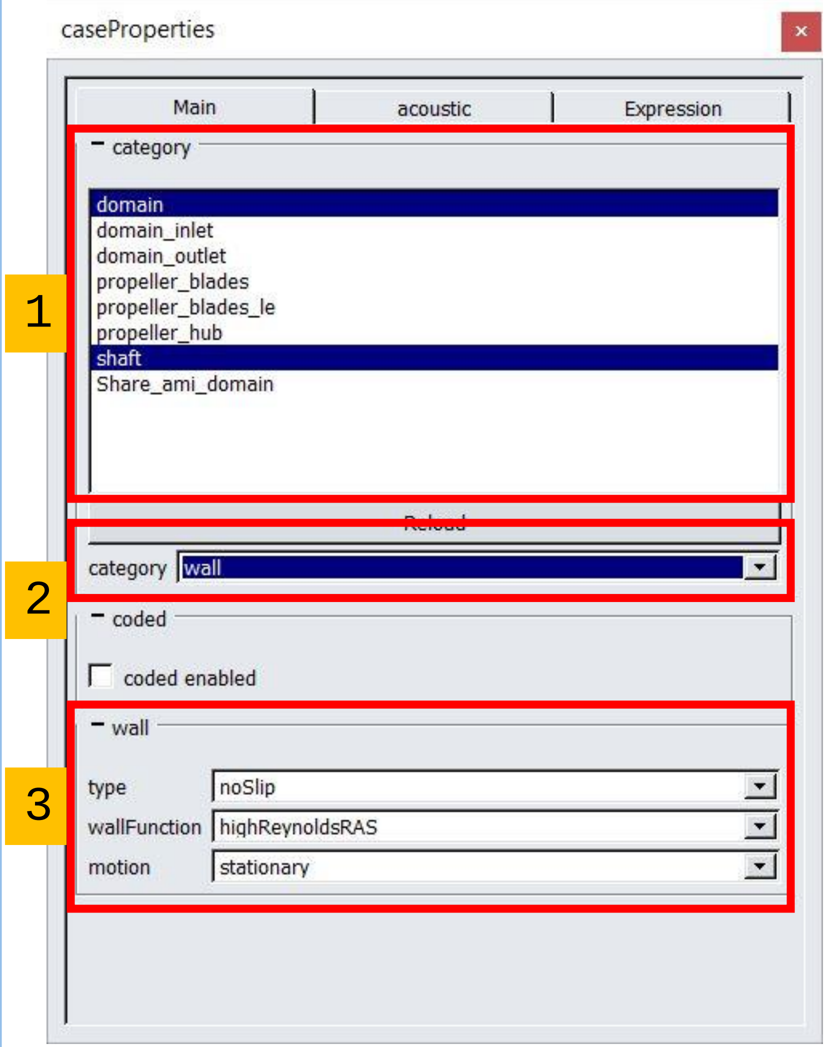


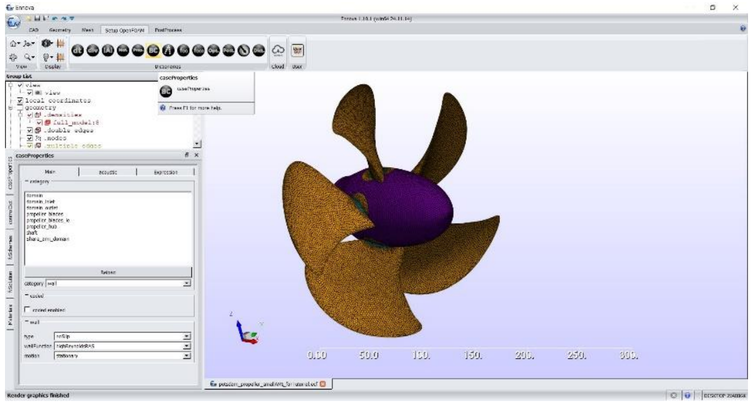


Volume Settings - Functions

- Volume domain which is considered as a cellZone will be available here
  1. Zone Settings : v_ami
  2. Motion Solver
     - Check multiSolidBodyMotionSolver enabled
     - For mesh motion of v_ami
  3. Motion
     - solidBodyMotionFunction : Rotating Motion
     - Origin : 0, 0, 0
     - Axis : -1, 0, 0
       - Select rotation axis according to right hand thumb rule
  4. Propeller revolution in rad/s
     - Omega : 94.24778 (900 rpm)

Boundary Conditions

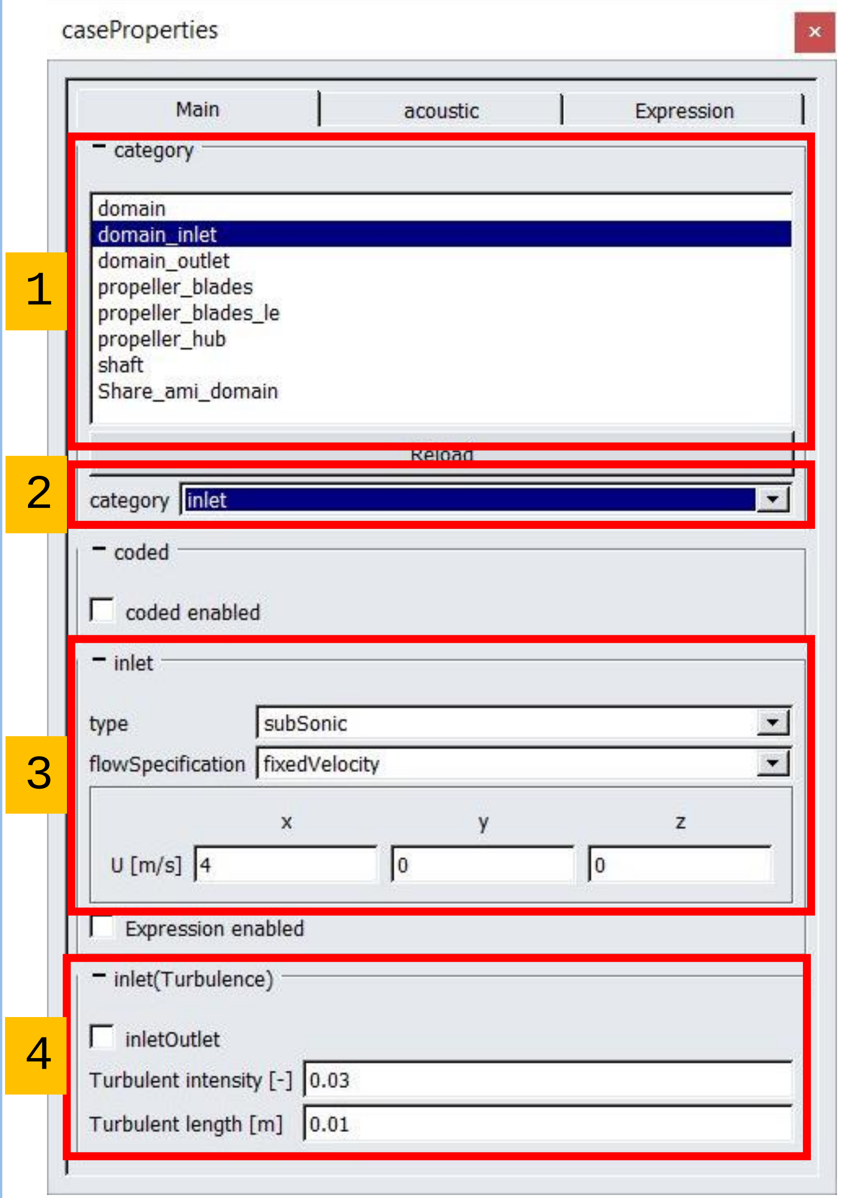


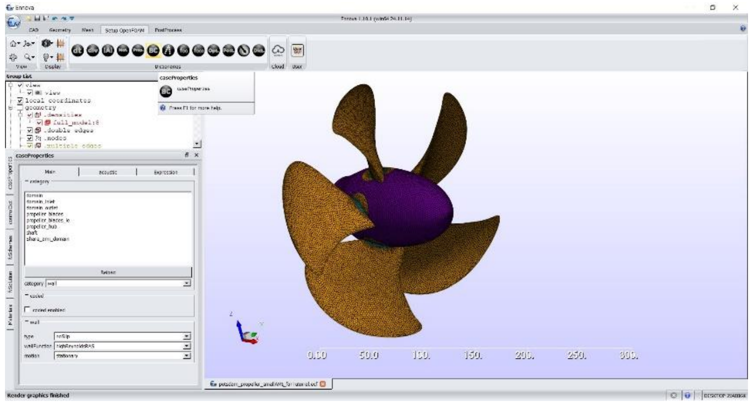


Boundary Conditions - Walls

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

Boundary Conditions

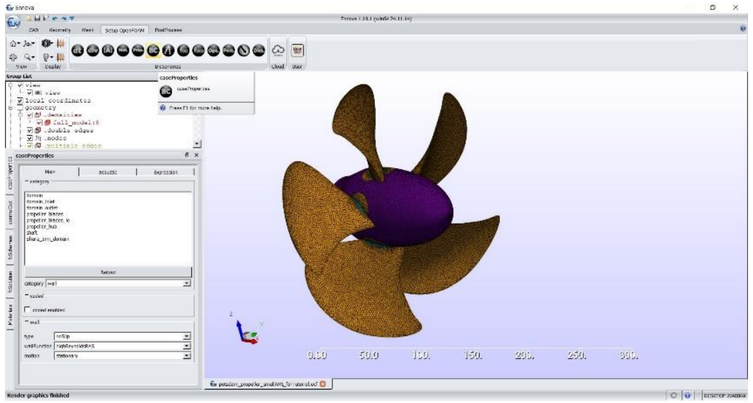


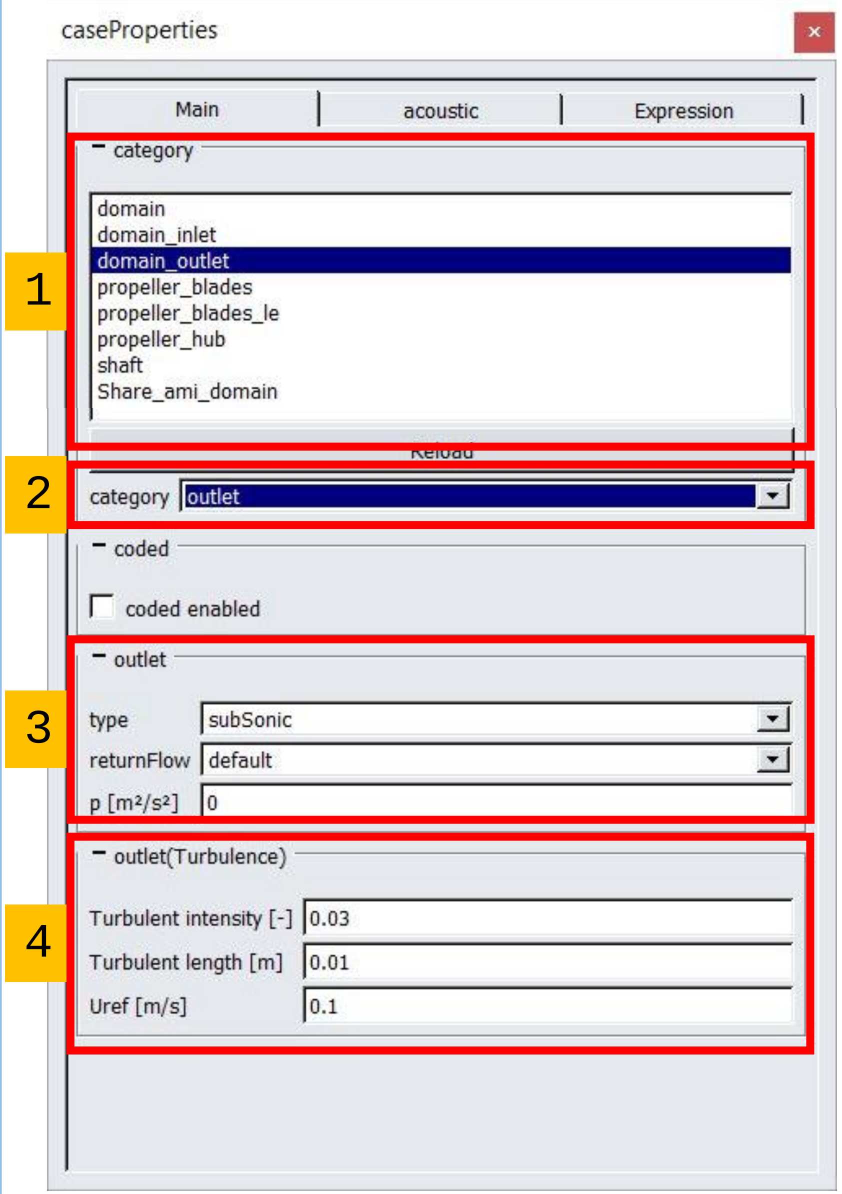


Boundary Conditions - inlet

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

Boundary Conditions

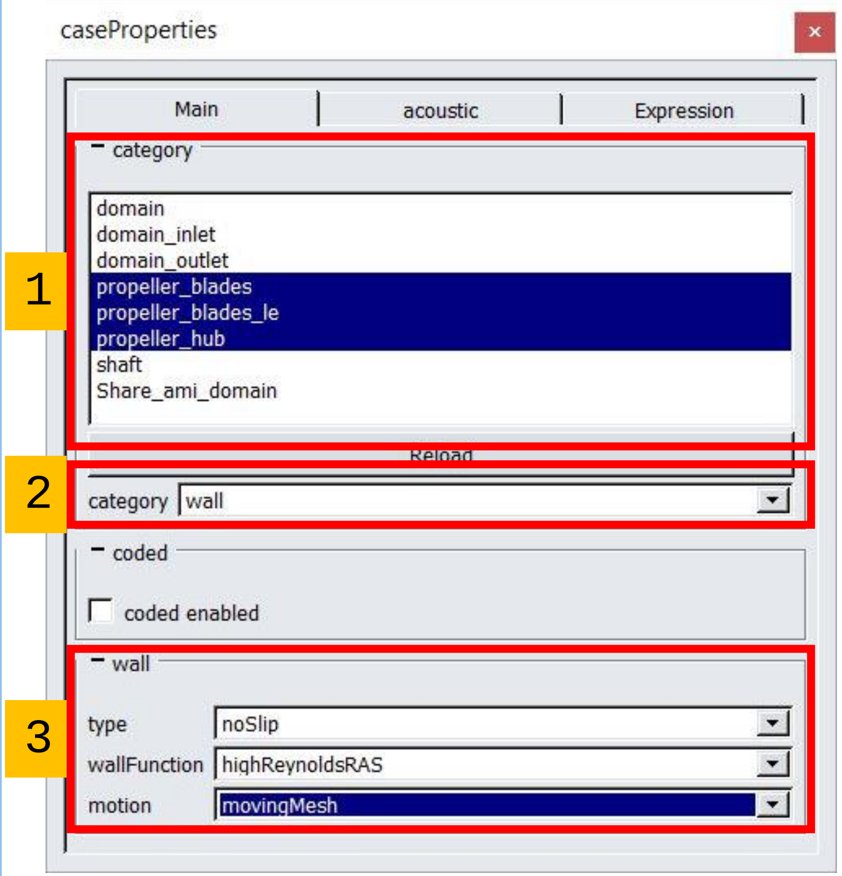


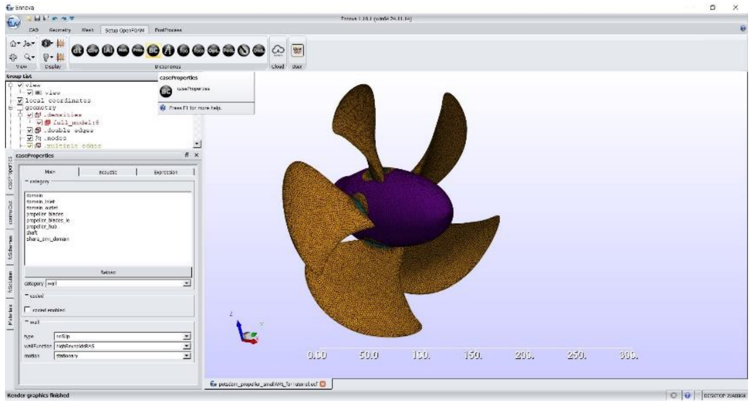


Boundary Conditions - outlet

1. Select part which is outlet to the domain
   - Here it is domain_outlet
2. Category : outlet
   - Select BC type as inlet for domain_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

Boundary Conditions

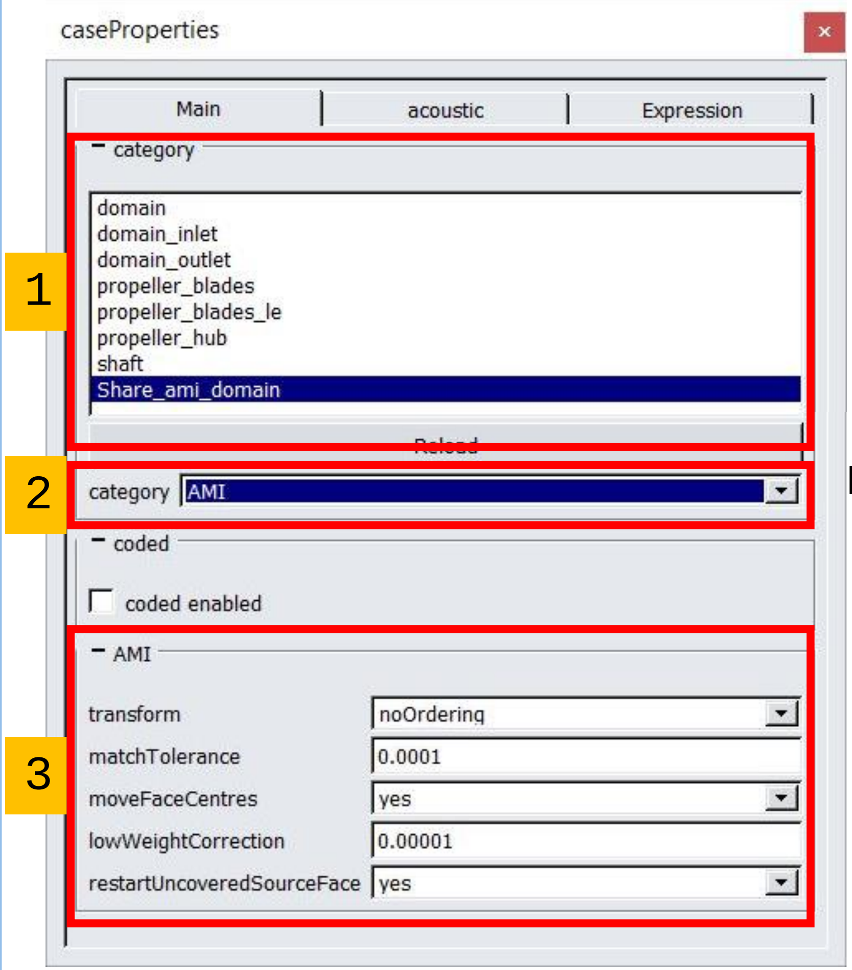


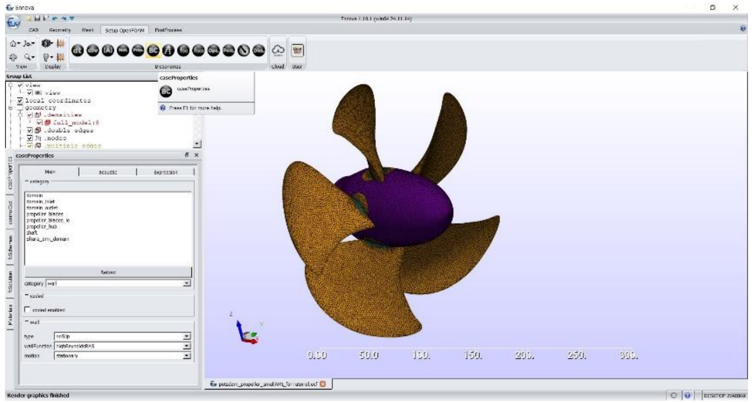


Boundary Conditions - Moving Wall

1. Select parts which are moving
   - Here it is propeller which is divided into 3 parts
2. Category : wall
   - Select BC type as wall for all propeller parts
3. Settings for moving wall type BC
   - Type : noSlip (default)
   - wallFunction : highReynoldsRAS (default)
   - Motion : movingMesh
     - All propeller parts are inside AMI domain
     - Hence, these should be given as moving mesh motion

Boundary Conditions

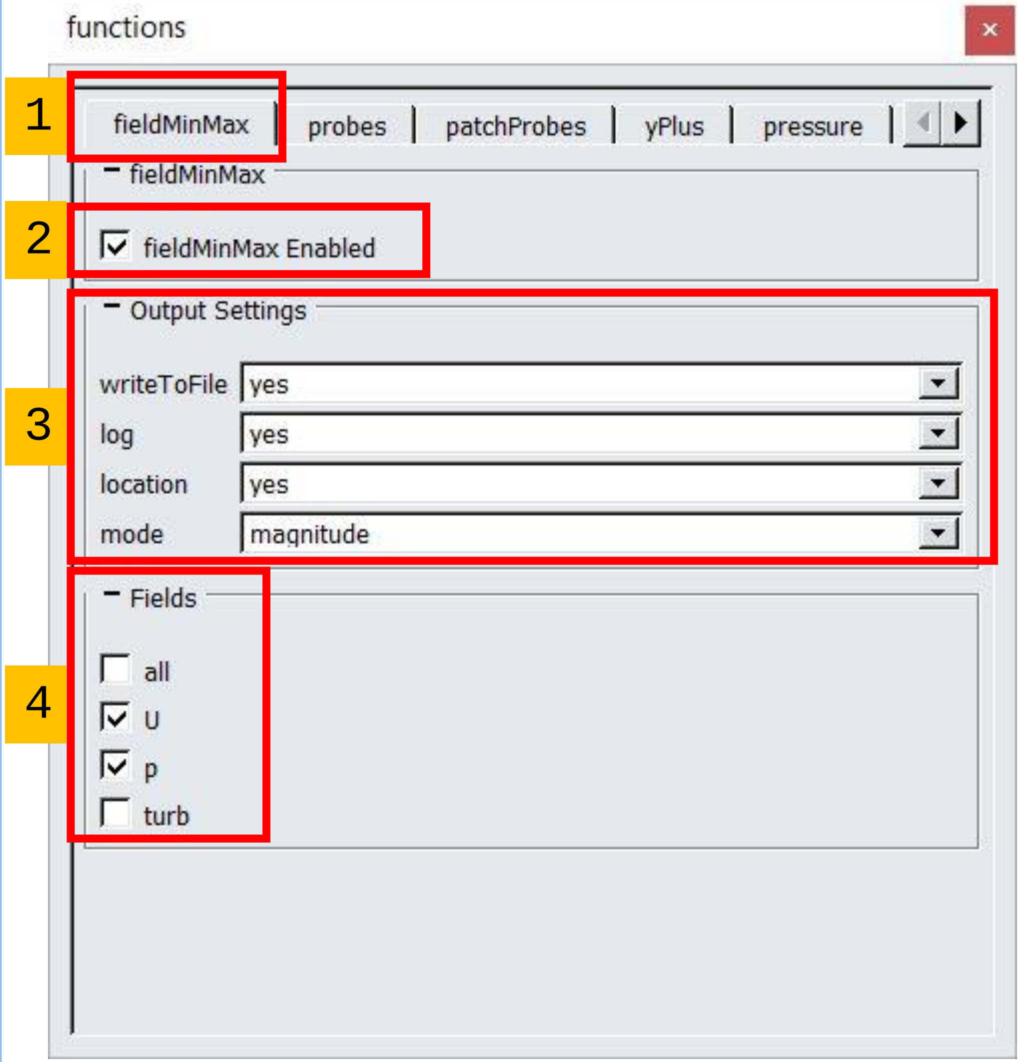


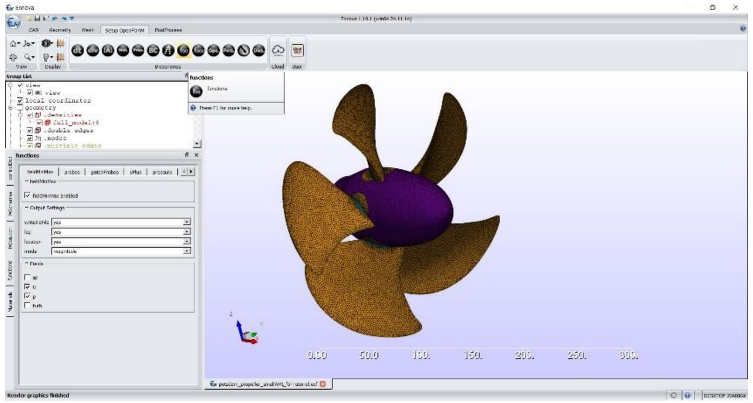


Boundary Conditions - AMI

1. Select parts which are interface for AMI and fluid domain
   - Here it is Share_ami_domain
     - Ennova by defaults gives prefix to parts which share 2 volumes
2. Category : AMI
   - Select BC type as AMI for Share_ami_domain part
3. Settings for AMI type BC
   - Keep everything as defaults

Post Processing

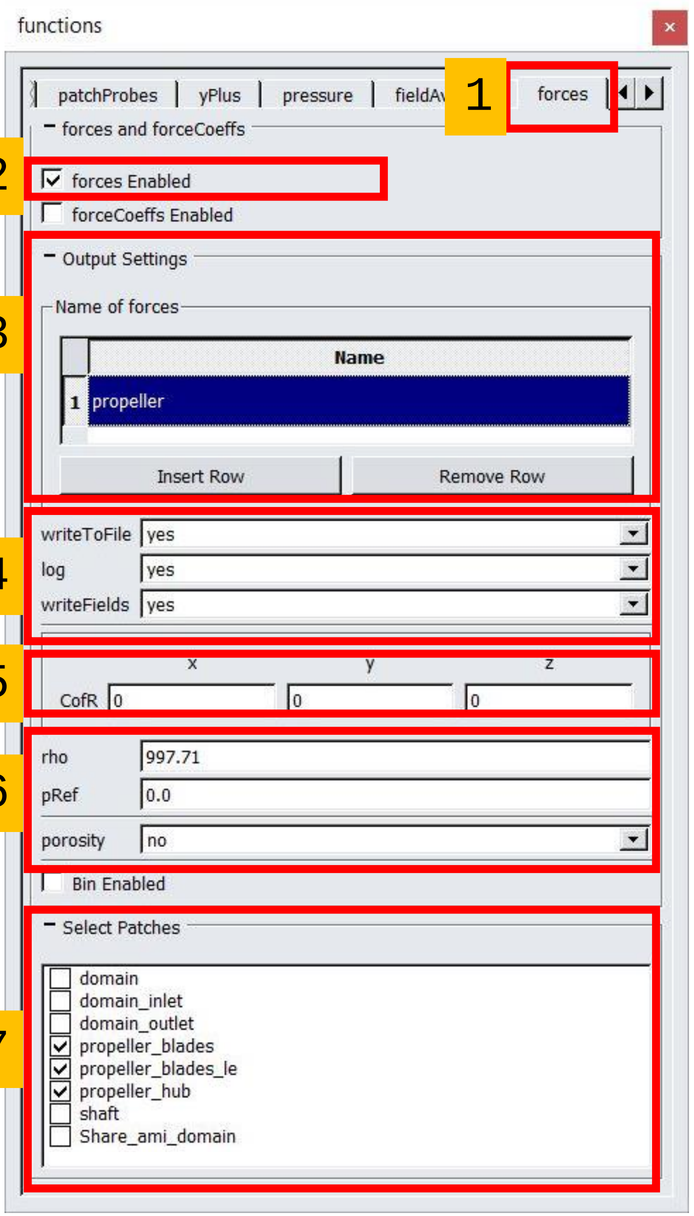


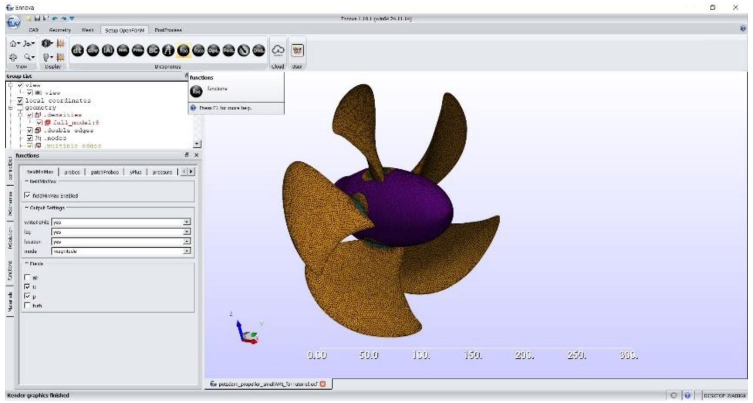


Post Processing

1. Function : fieldMinMax
    - This utility prints the min and max values of selected fields
2. Check box to output the field min and max values
3. Output settings
    - writeToFile : Yes
        - To print all min max values in text file
    - Log : Yes
        - To print values in solver log file
    - Location : Yes
        - Gives co-ordinates of min max values
    - Mode : magnitude (default)
4. Check boxes against fields for which min & max output is needed

Post Processing

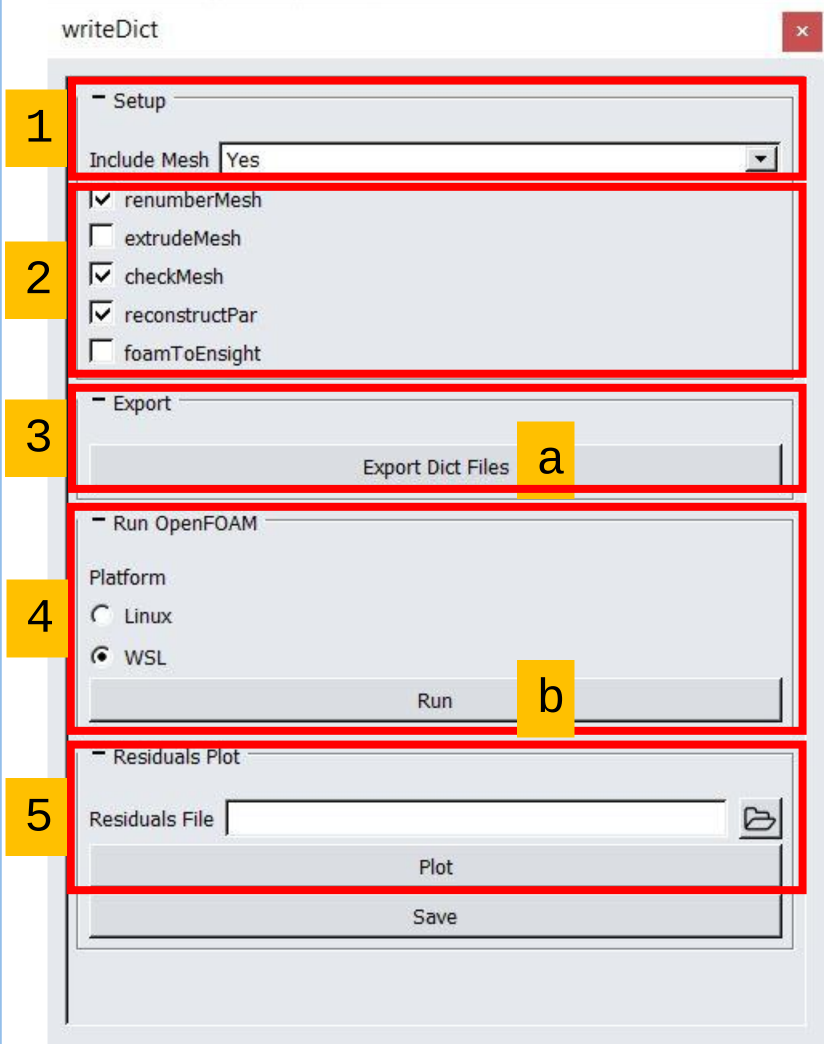




Post Processing

1. Function : forces
    - This utility prints the force and moments on the selected surface patches
    - Force and moment values are needed to find thrust and torque of the propeller
2. Check box to output the force and moments on the selected surface patches
3. Name : text file output name
    - Click on insert rows to add file in which output is required
4. Output settings
    - writeToFile : Yes (To print all min max values in text file)
    - Log : Yes (To print values in solver log file)
    - writeFiels : Yes
        - To write force and moment data on surface patches
        - Check this box if force and moment contours are needed on surface patches
5. CofR : Give co-ordinates of center of roation for moment calculation
6. Field properties
    1. Provide density value of primary fluid. Here it is water
7. Check surface patches on which force and moments are to be calculated

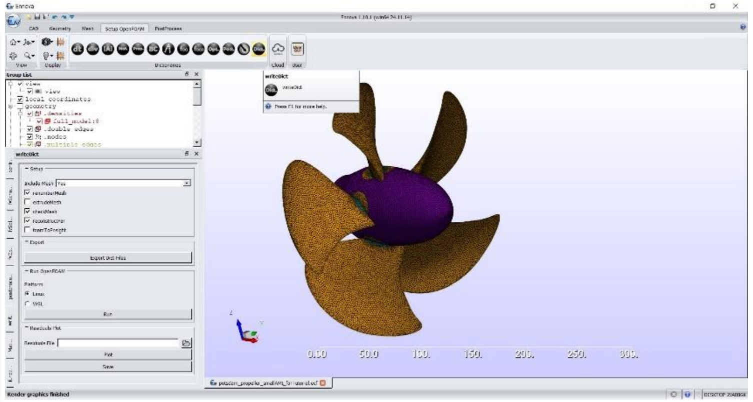
Export OpenFOAM Files



There are two ways of running OpenFOAM case with Ennova,

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

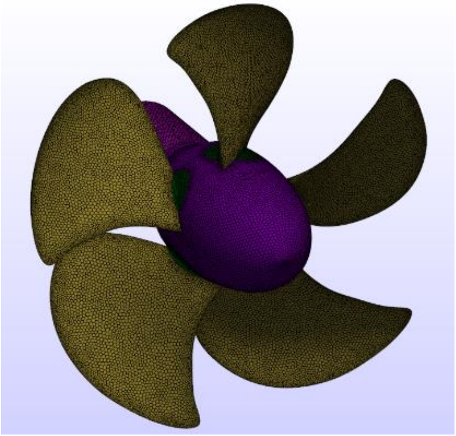
b. Run
   - Running on WSL directly from Ennova



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

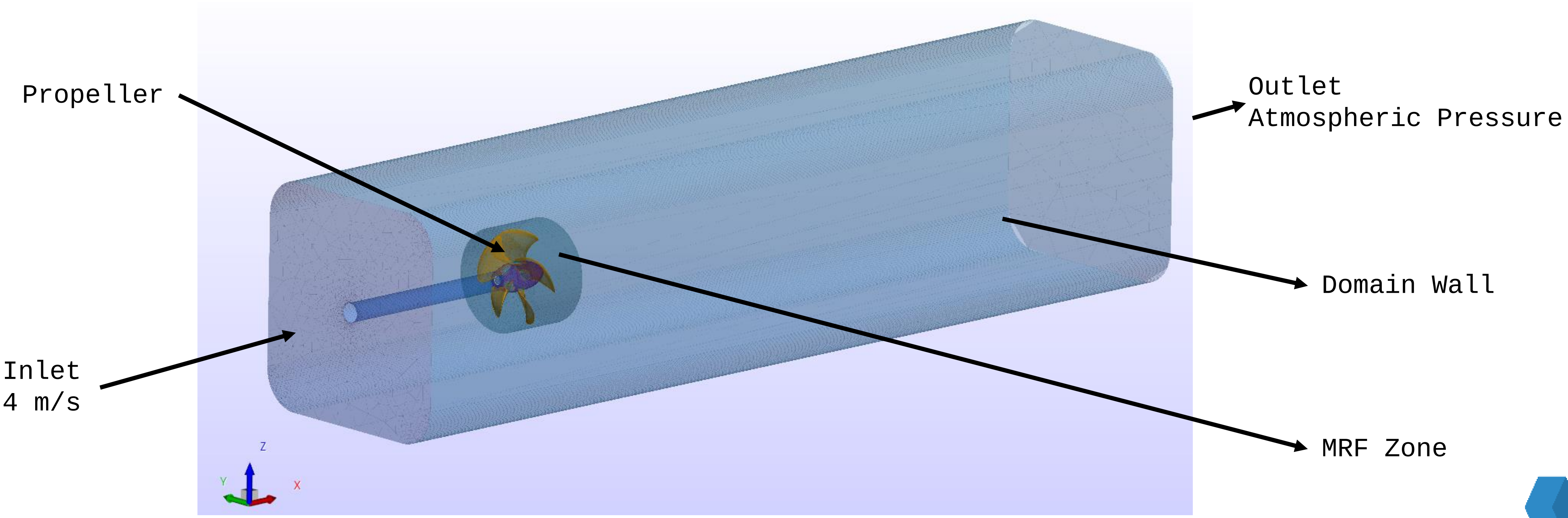
Case : MRF - SteadyState
Potsdam Propeller



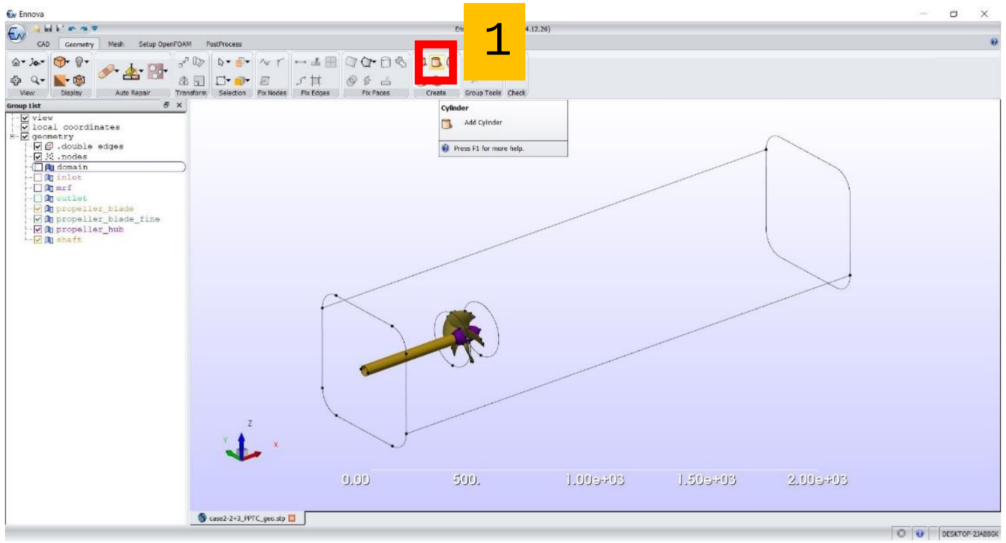
Boundary Conditions

- Mean flow speed : 4 m/s
- Rotation speed : 900 RPM
- Flow model : incompressible
- Mesh size : 1.53M cells
- Simulation point : 1 (J = 1.0667)
- Fluid : Water
- Reference pressure : 1 atm

- Reference density : 997.71 kg/m^3
- Dynamic viscosity : 9.559×10^{-4} Pa·s
- Turbulence model : kOmegaSST
- Simulation type : Propeller
- Time management : Transient
- Rotational Direction : (-1, 0, 0)

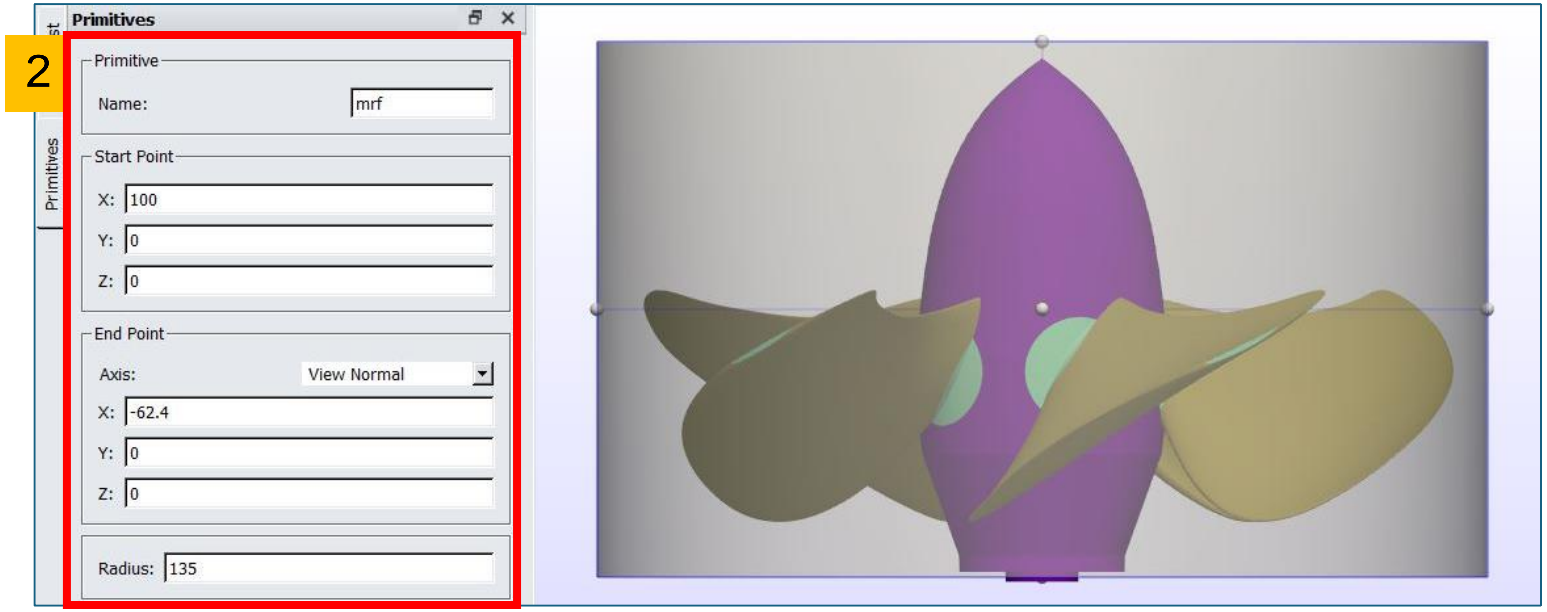


Creating MRF region



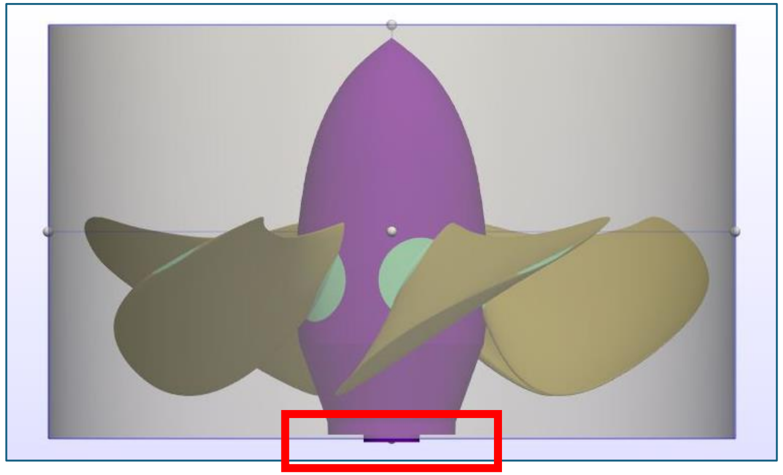
Load CAD or existing ecf (CAD) file

- Make sure All parts have been named properly
- Keep only propeller (with hub) visible
- Steps
  1. Select create primitive parts (Cylinder)
  2. Supply required size and direction for MRF region



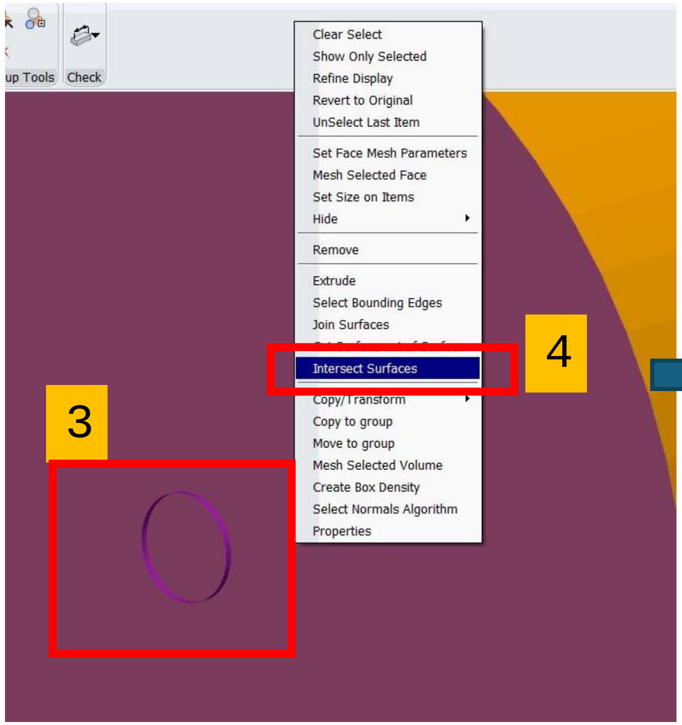
- All the rotating parts should be completely inside MRF region
- MRF region should be as close as possible to the rotating parts
  - 3-4 cells should fit between the gap

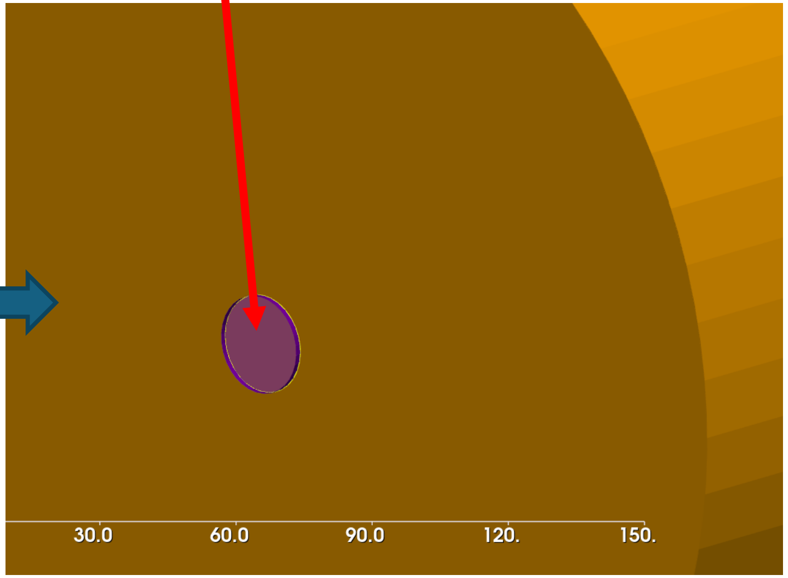
Creating MRF region



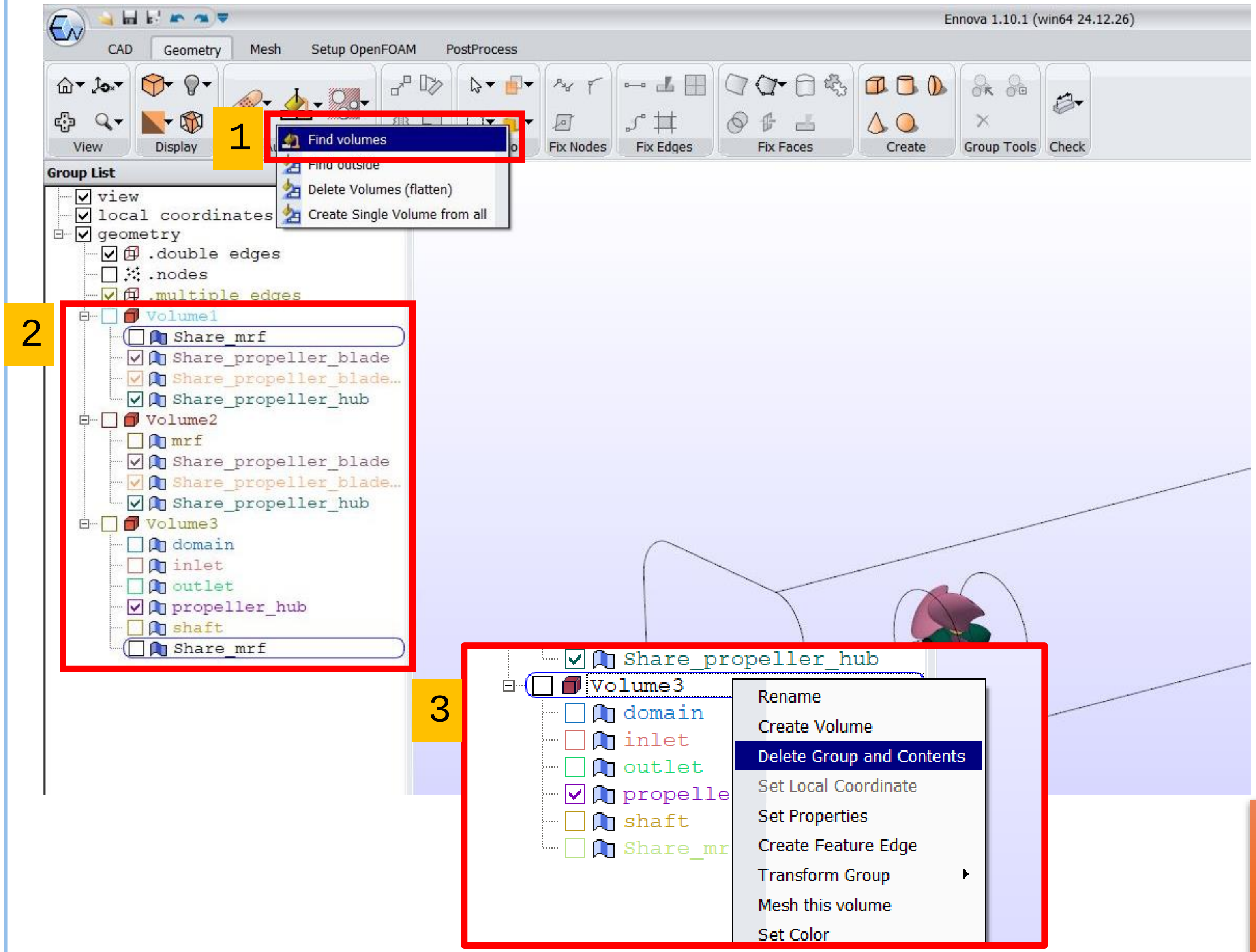
- Steps
  3. Select MRF and Hub surfaces
  4. Select Intersect Surface to Intersect MRF and hub region

- A yellow line (represents multiple edges) will appear after intersection operation
- **DON'T DELETE** THE Highlighted portion at the Center





Creating MRF region –Volumes

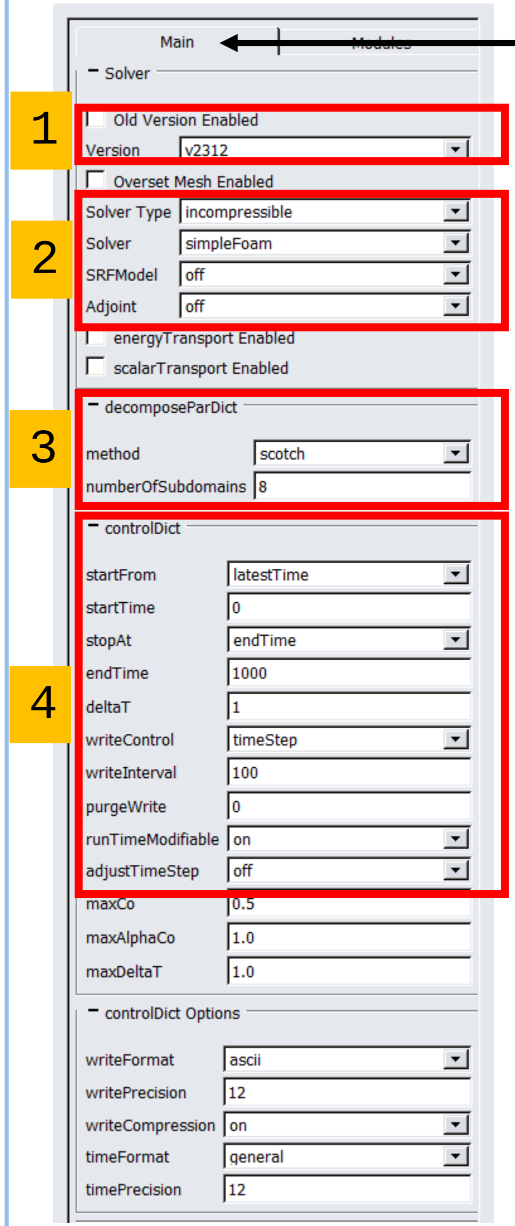


1. Select Find Volumes from Geometry tab under Auto-repair section
2. 3 closed volumes are created
   - Volume1
     - Fluid Domain
   - Volume2
     - MRF Region
   - Volume3
     - Propelled region

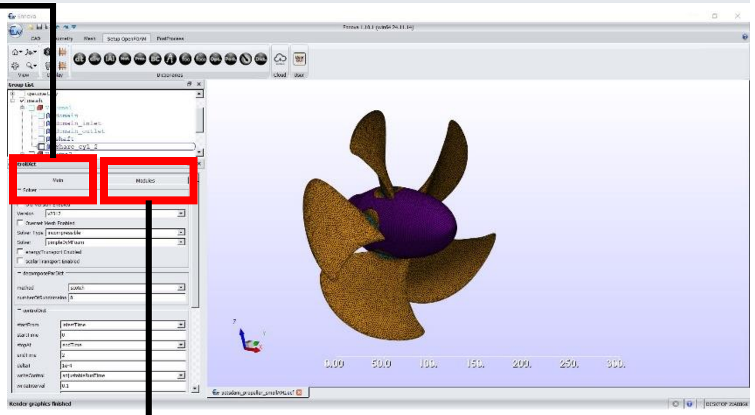
3. Volume3 is solid volume of propeller which is not required
   1. Right click on Volume3 and select ***Delete Group and its Content***

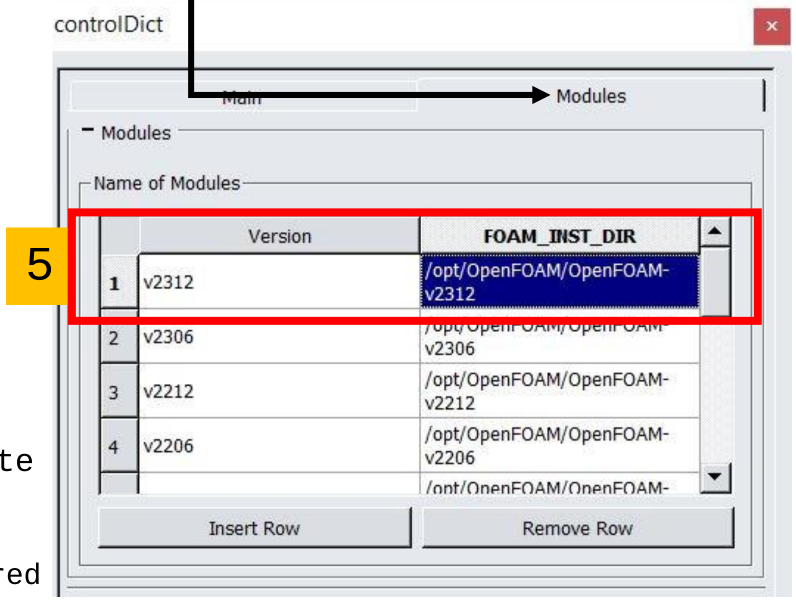
After these operations generate surface and volume mesh as usual and then move to case set-up

Solver Settings

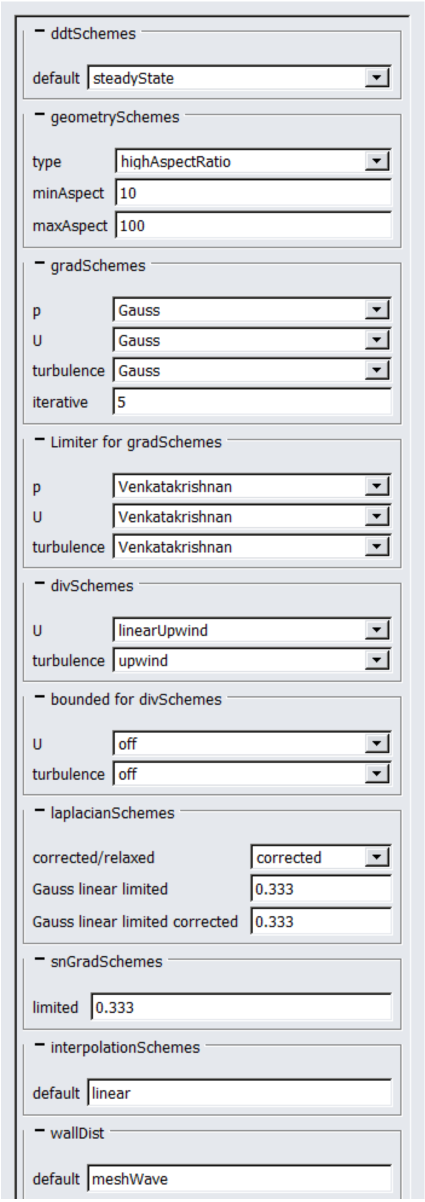


1. Version
   - Select latest version of OpenFOAM
   - Make sure same version is installed where case has to be simulated
   - Here it is v2312
2. Solver Settings
   - Type : incompressible
   - Solver : simpleFoam
     - Transient OpenFOAM solver
3. Decomposition
   - Method : Scotch
     - For equal 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



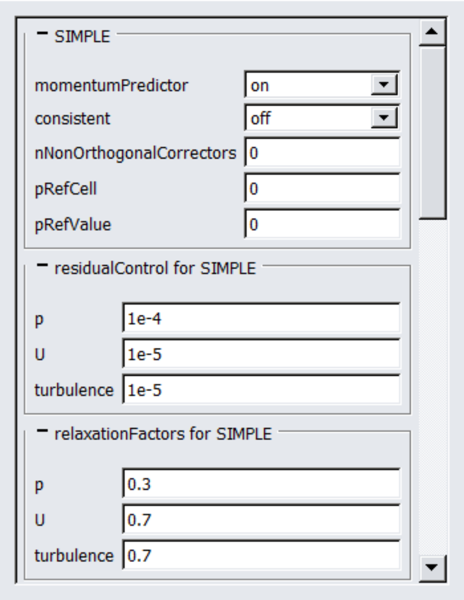


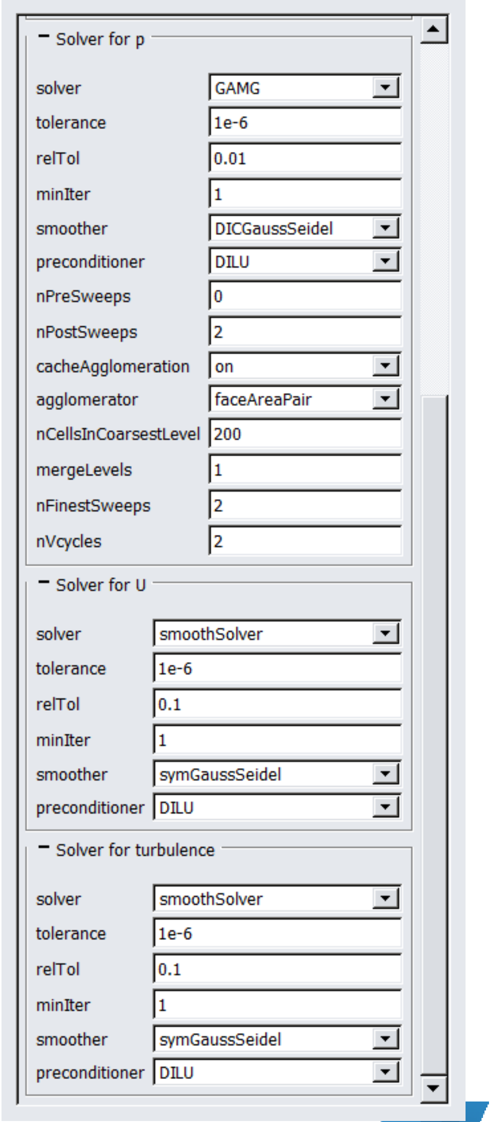
Schemes Settings



fvScheme

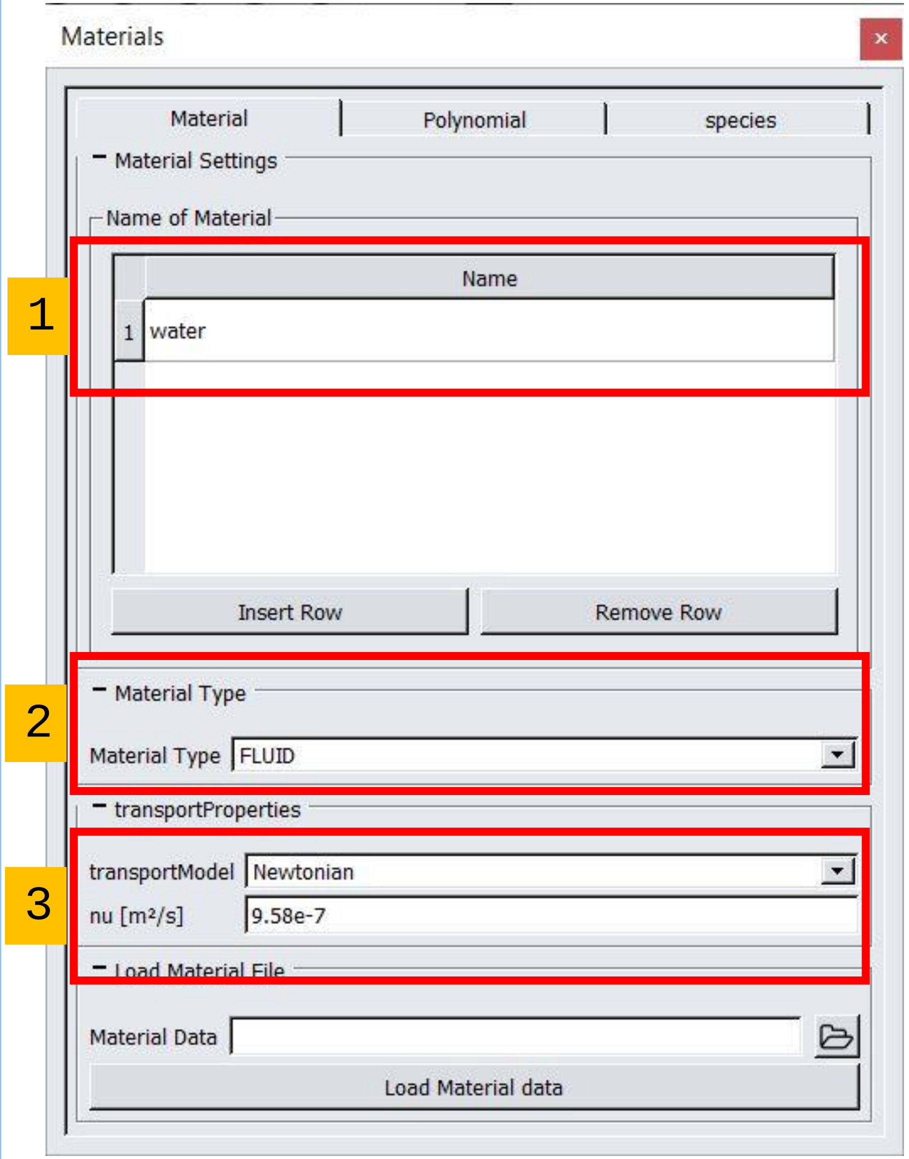
- fvScheme and fvSolution
  - Schemes and simulation convergence tolerance, relaxation factors etc.
- ***Keep it default.***

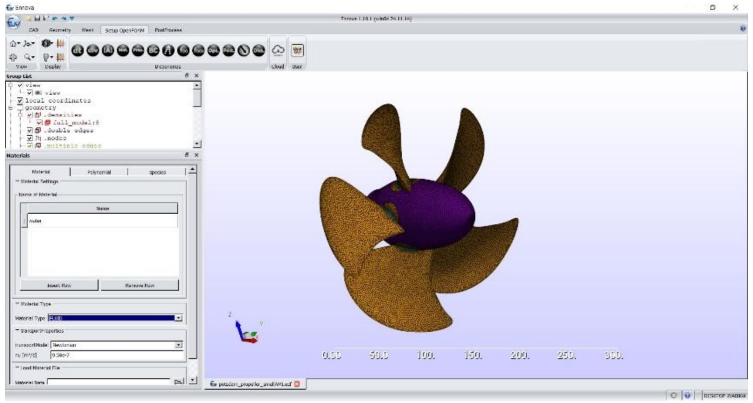




fvSolution

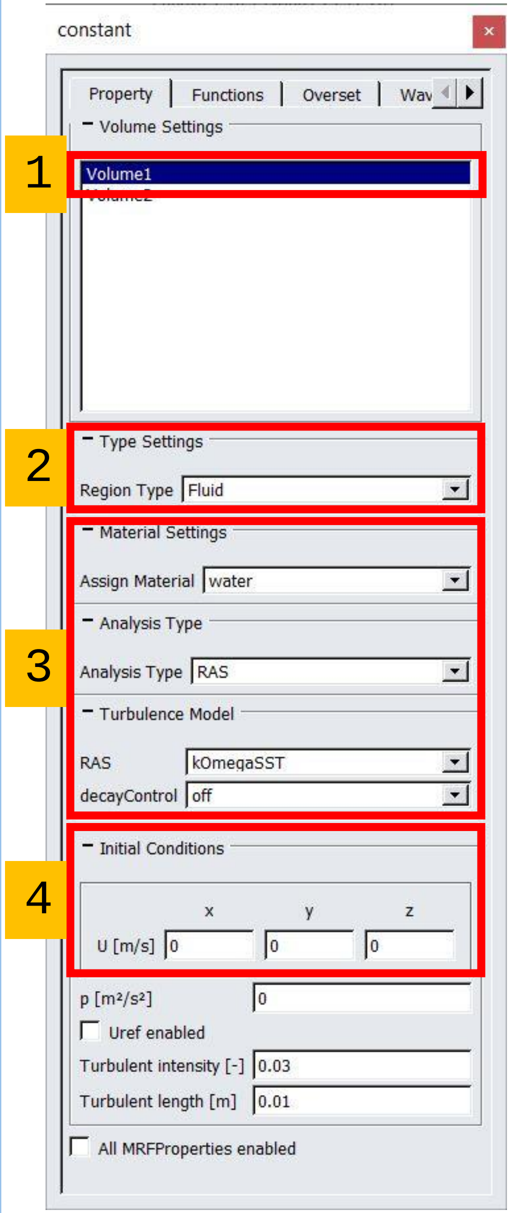
Material Settings





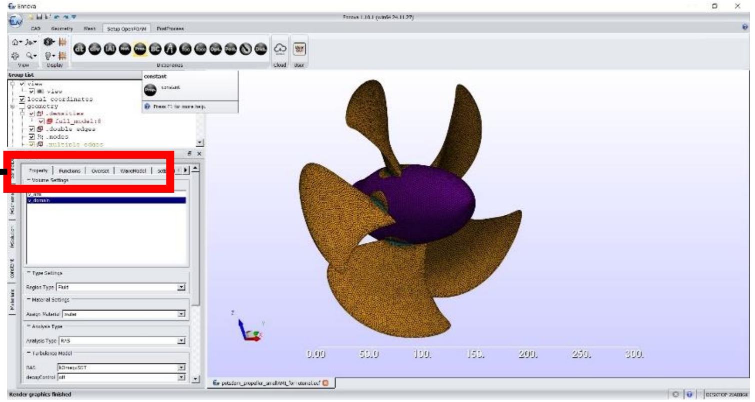
1. Material Name
   - Default is air
   - Double click the Air to update the name
2. Material
   - Material Type : Fluid (Default)
3. transPortProperties
   - transportModel : Newtonian (Default)
   - Nu (Kinematic Viscosity) : $9.58e^{-7} (m^2/s)$ updated value for water

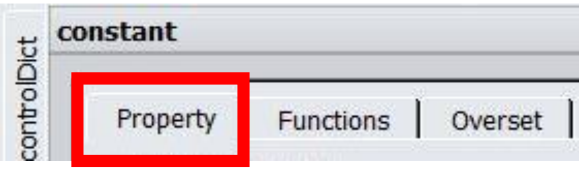
Property Settings

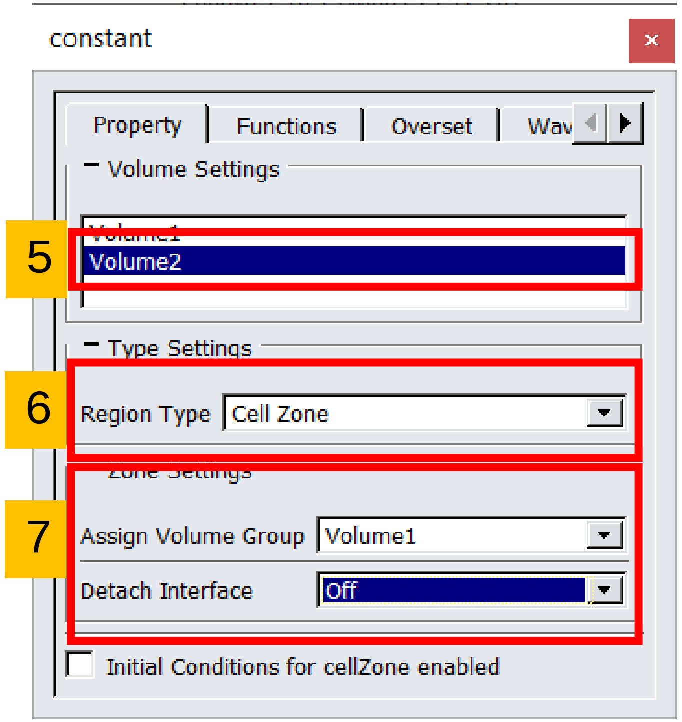


Volume Settings - Property

- Primary Zone : fluid domain
  1. Domain : Volume1
  2. Type Settings
     - Region Type : Fluid (Default)
  3. Material Settings
     - Keep it default
  4. Initial Conditions
     - U (m/s) : 4, 0, 0
     - Supply inlet velocity value



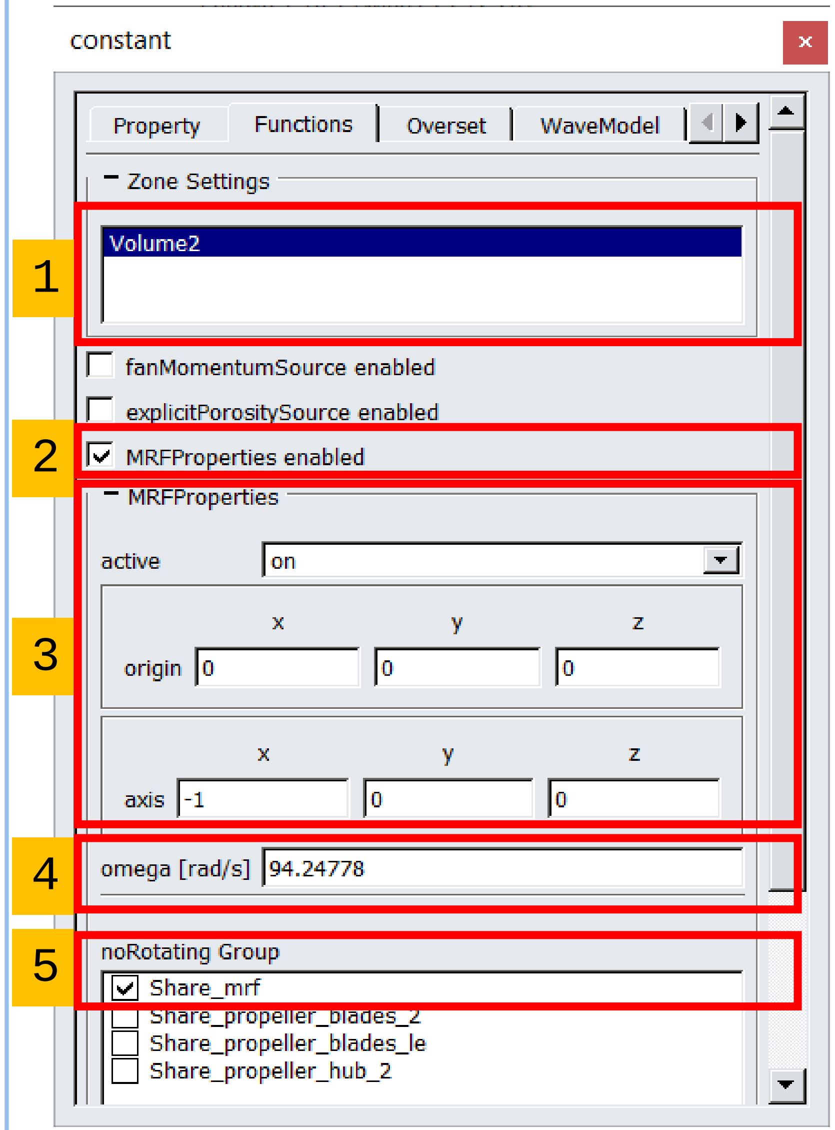


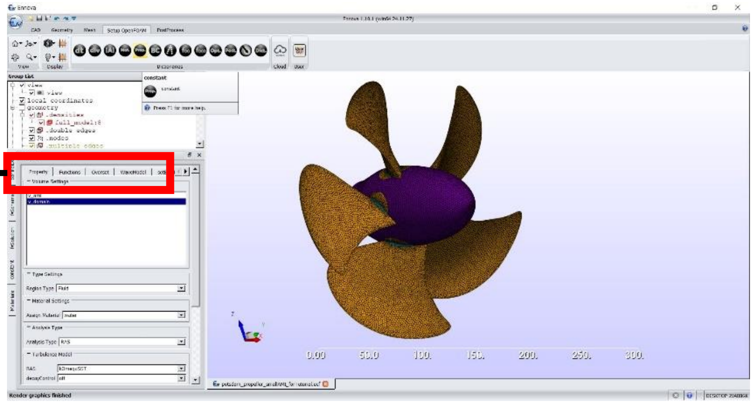


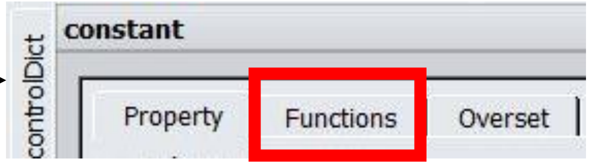
Volume Settings - Property

- Secondary Zone: MRF
  5. Domain : Volume2
  6. Type Settings
     - Region Type : Cell Zone
       - Select it from drop down menu
  7. Zone Settings
     - Assign volume Group : Volume1
       - Assign it with the surrounding zone. In this case it is primary fluid domain Volume1
     - Detach Interface : Off
       - Keep detach interface OFF

Property Settings





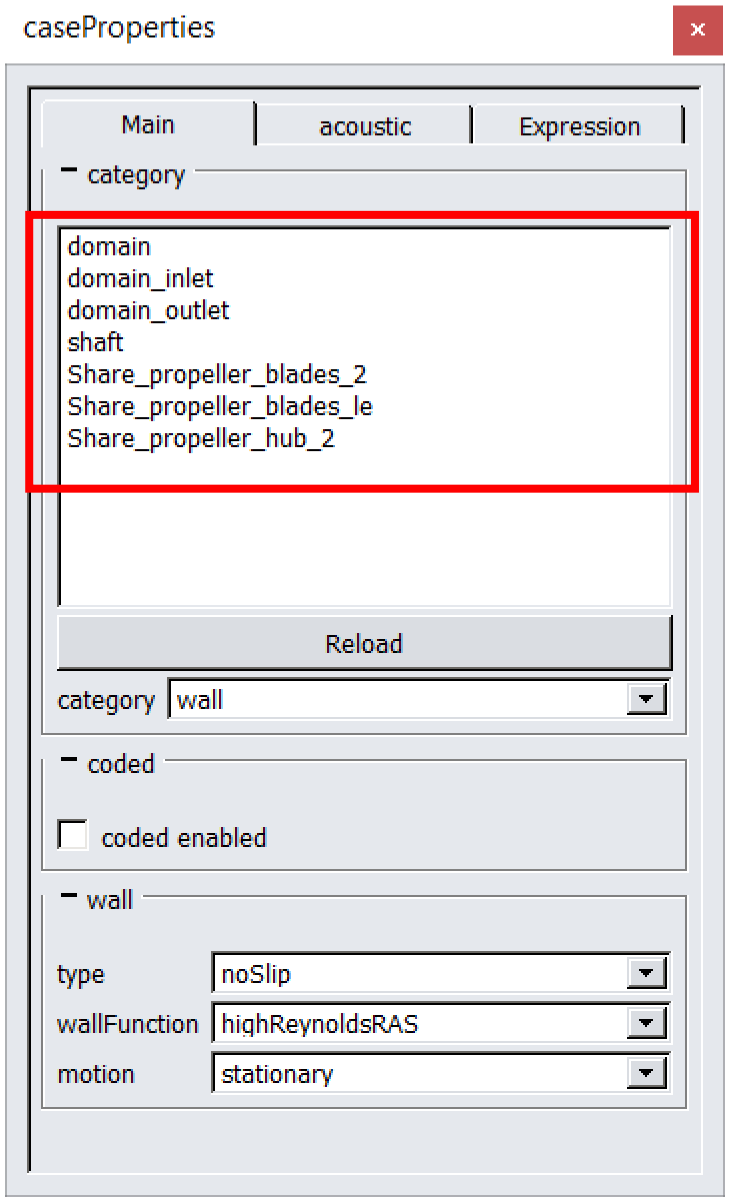


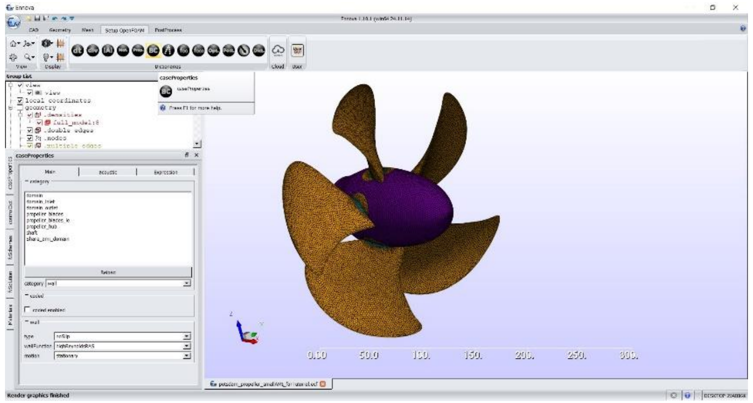
Volume Settings - Functions

- Volume domain which is considered as a cellZone will be available here
  1. Zone Settings : Volume2
  2. MRFProperties enabled
     - Check this option for MRF
  3. MRFProperties
     - Active : On
     - Origin : 0, 0, 0
     - Axis : -1, 0, 0
       - Select rotation axis according to right hand thumb rule
  4. Propeller revolution in rad/s
     - Omega : 94.24778 (900 rpm)
  5. noRotating Group
     - All parts inside Volume2 will be visible here
     - Check Share_mrf – It is a boundary of MRF zone and it is non rotating.

ennova-cfd.com

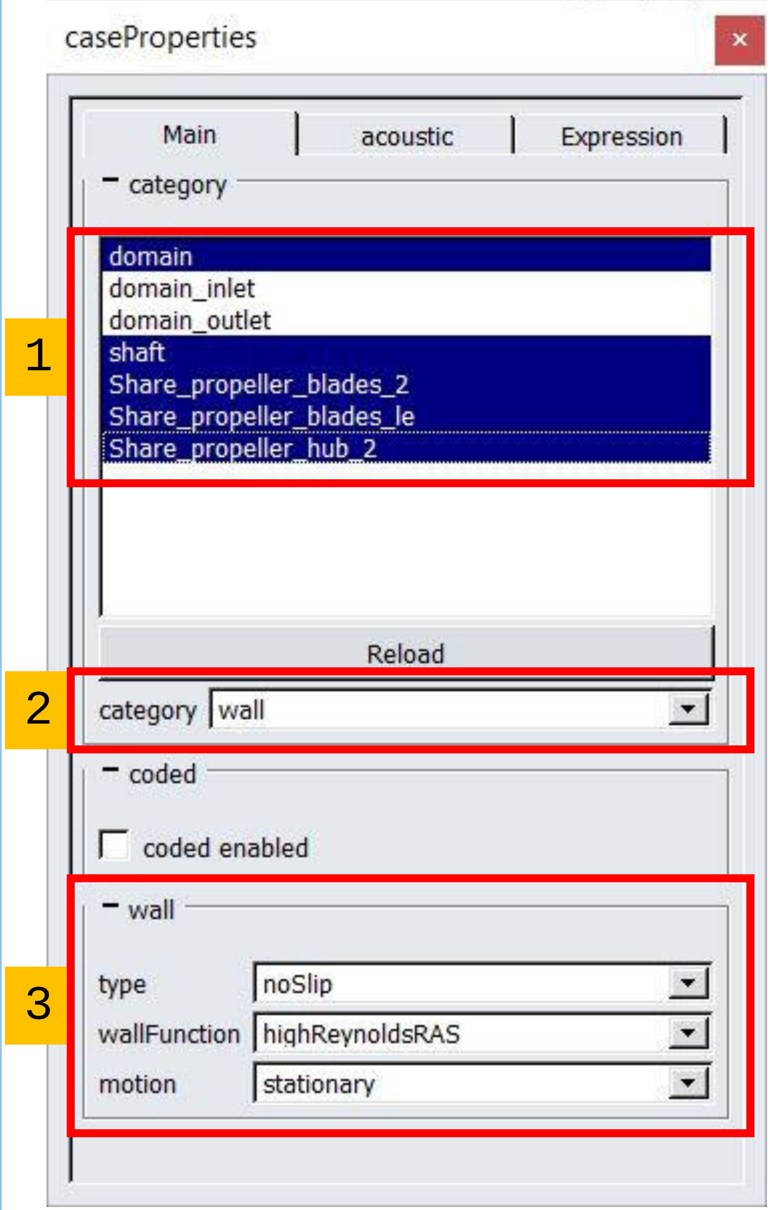
Boundary Conditions

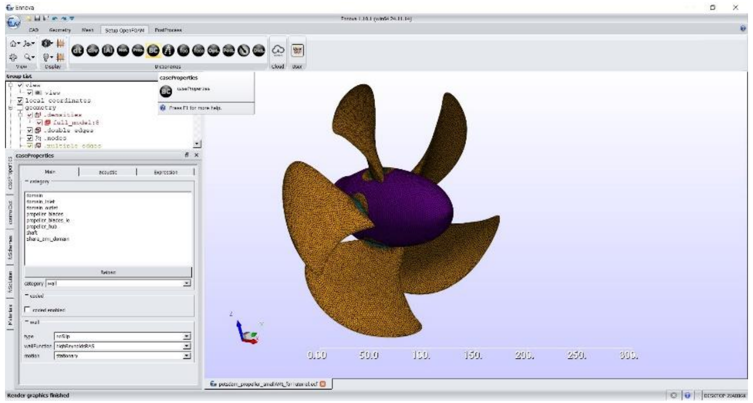




- To make sure that MRF setting is correct, Share_mrf group/part name should not be available in the BC section
- Here, Share_mrf is not available in the BC section. Hence, we can proceed further for BC setup

Boundary Conditions



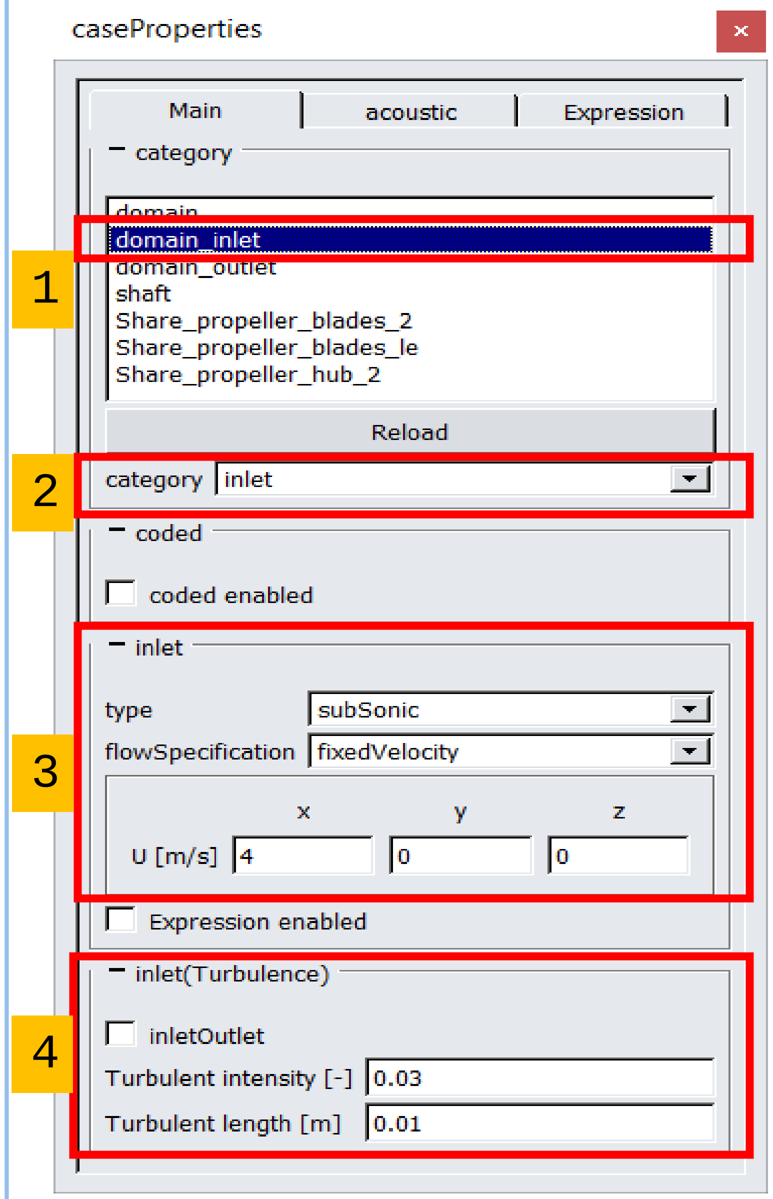


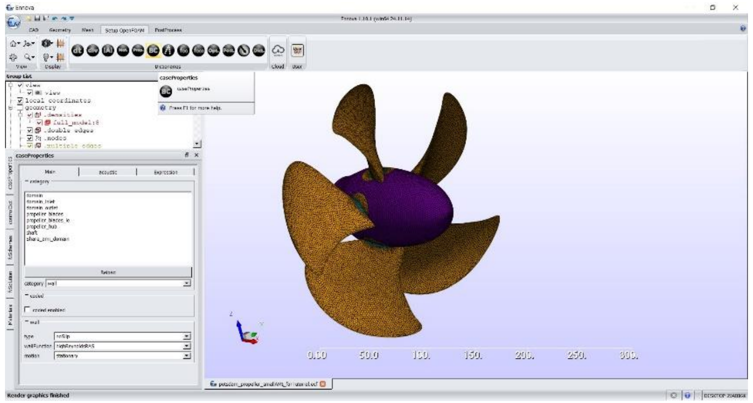
Boundary Conditions - Walls

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

This is MRF steady state case. Therefore propeller is considered as a stationary wall

Boundary Conditions

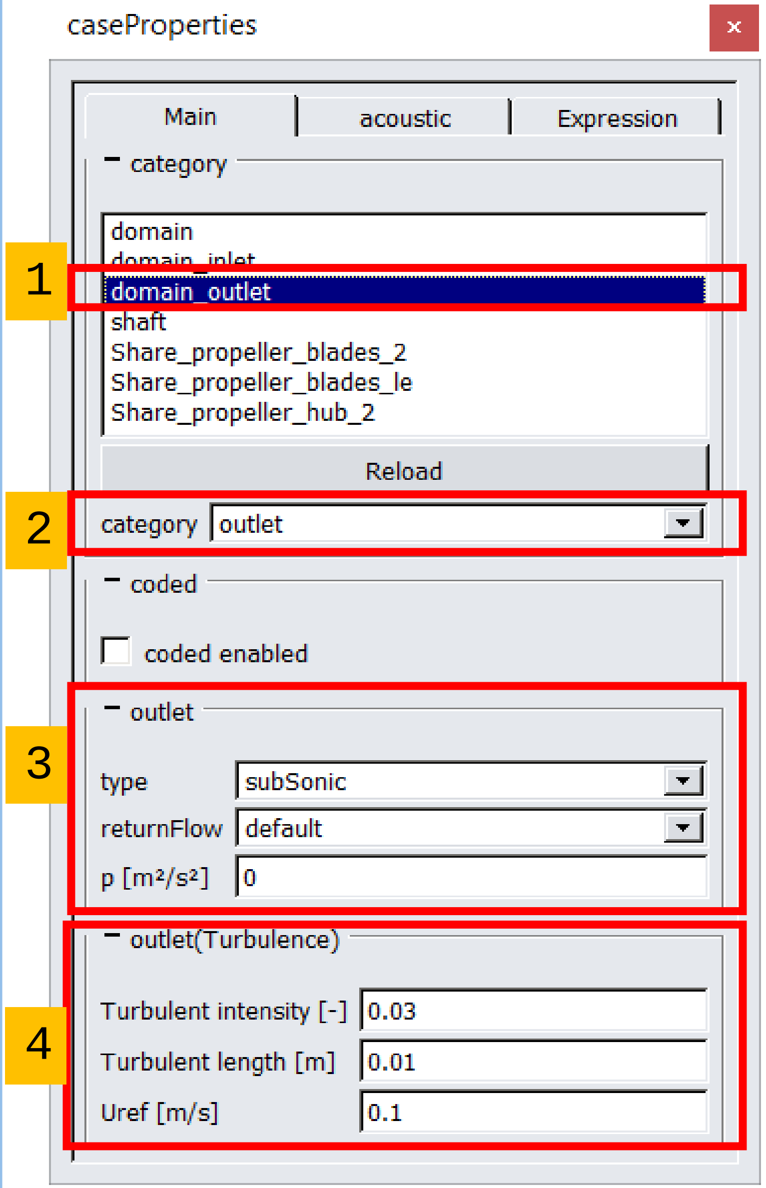


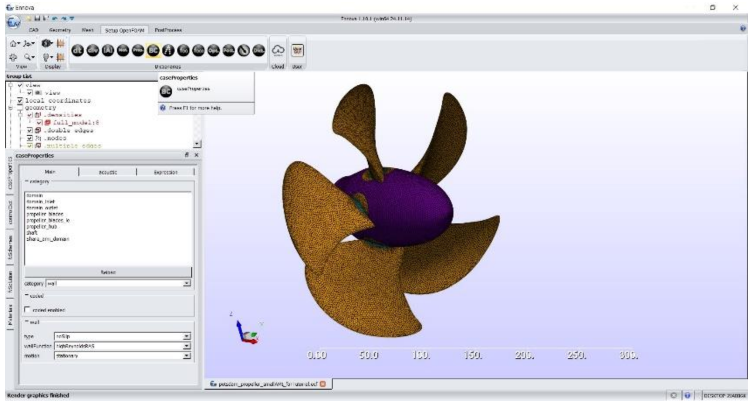


Boundary Conditions - inlet

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

Boundary Conditions

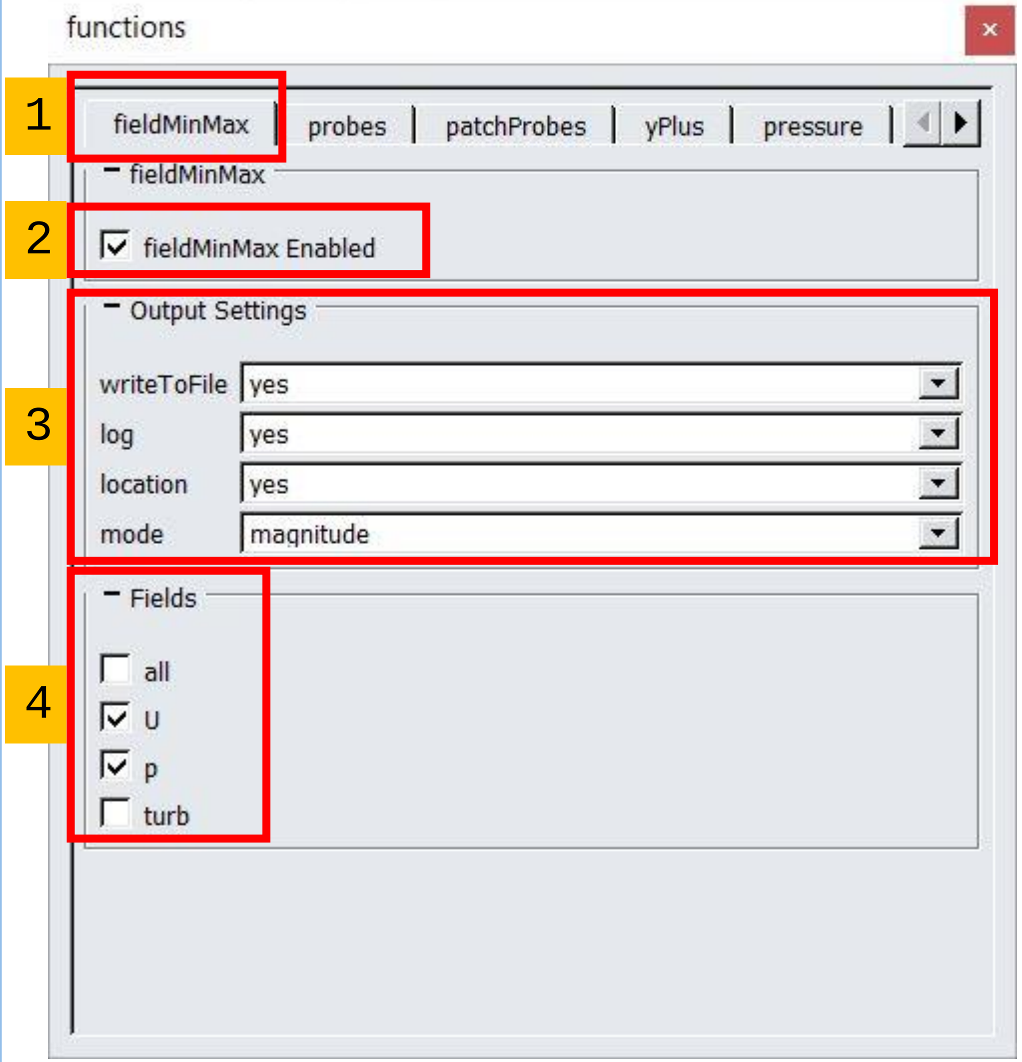


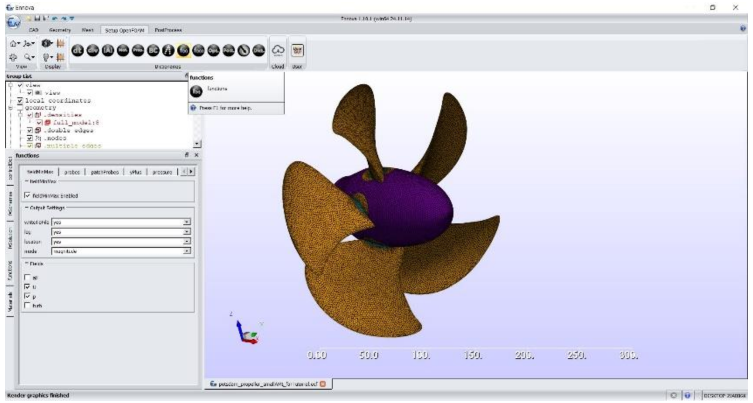


Boundary Conditions - outlet

1. Select part which is outlet to the domain
   - Here it is domain_outlet
2. Category : outlet
   - Select BC type as inlet for domain_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

Post Processing

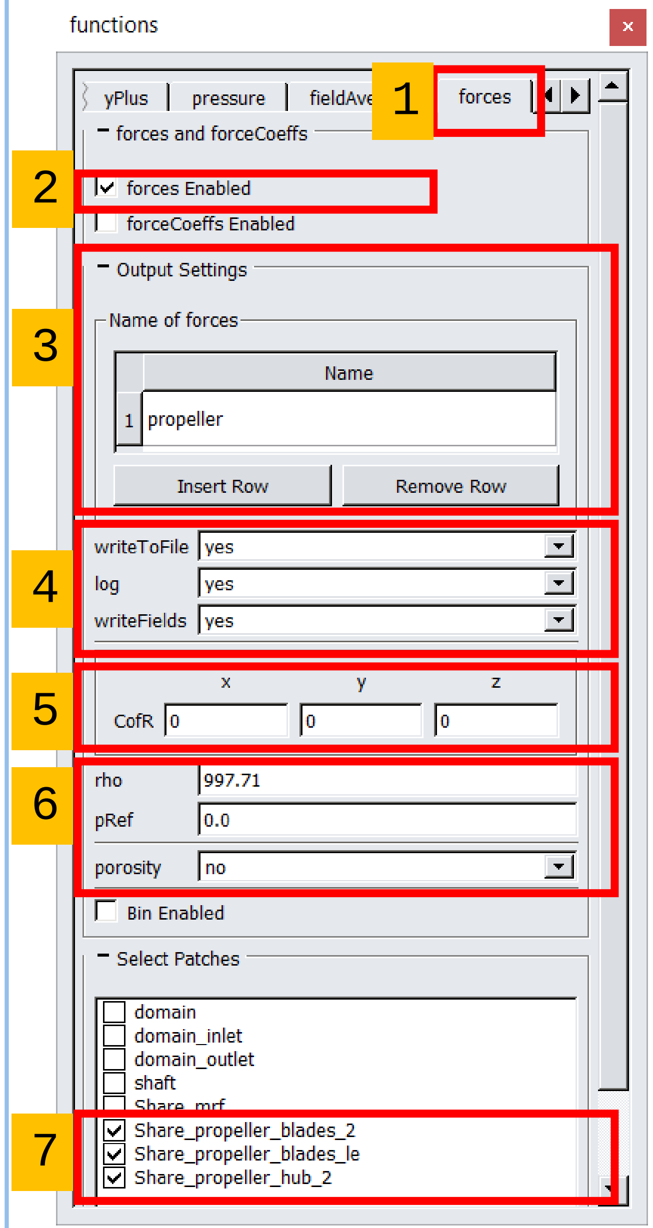


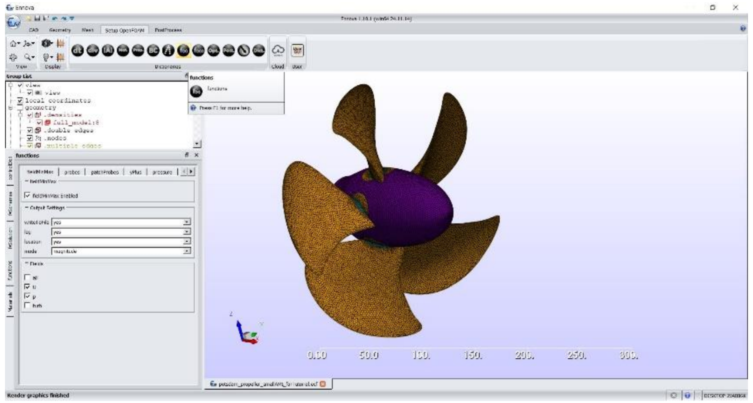


Post Processing

1. Function : fieldMinMax
    - This utility prints the min and max values of selected fields
2. Check box to output the field min and max values
3. Output settings
    - writeToFile : Yes
        - To print all min max values in text file
    - Log : Yes
        - To print values in solver log file
    - Location : Yes
        - Gives co-ordinates of min max values
    - Mode : magnitude (default)
4. Check boxes against fields for which min & max output is needed

Post Processing



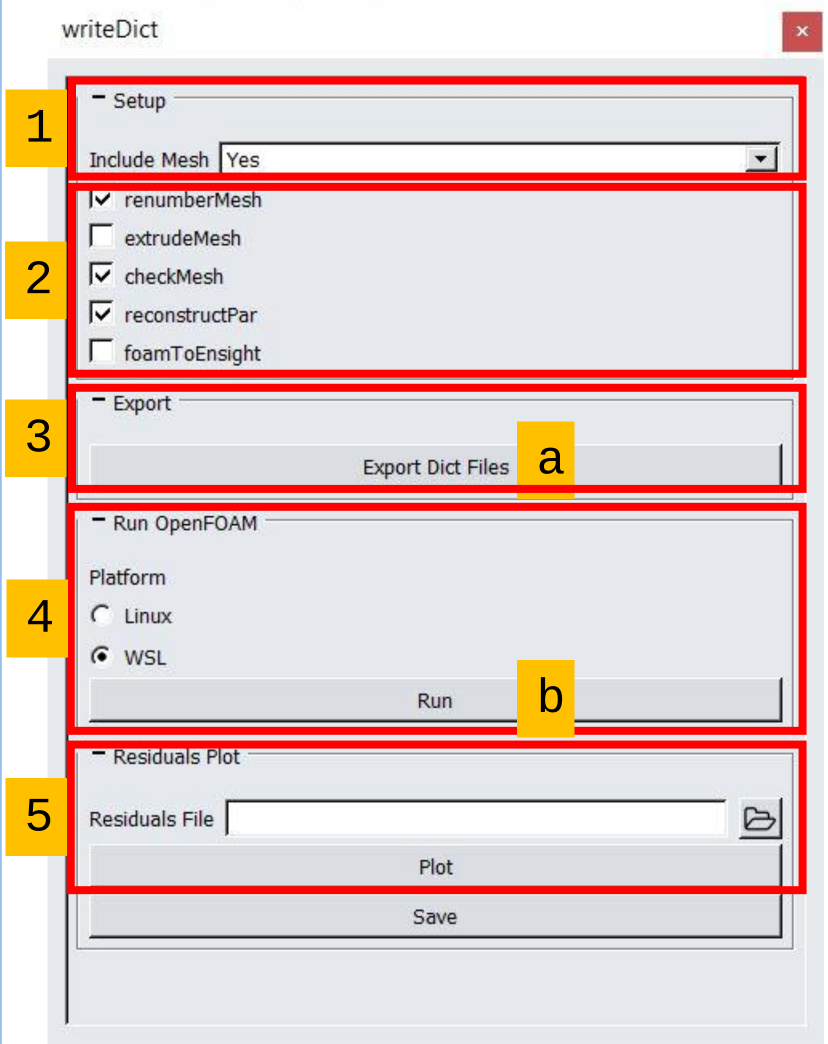


Post Processing

1. Function : forces
   - This utility prints the force and moments on the selected surface patches
   - Force and moment values are needed to find thrust and torque of the propeller
2. Check box to output the force and moments on the selected surface patches
3. Name : text file output name
   - Click on insert rows to add file in which output is required
4. Output settings
   - writeToFile : Yes (To print force values in text file)
   - Log : Yes (To print values in solver log file)
   - writeFiels : Yes
     - To write force and moment data on surface patches
     - Check this box if force and moment contours are needed on surface patches
5. CofR : Give co-ordinates of center of rotation for moment calculation
6. Field properties
   1. Provide density value of primary fluid. Here it is water
7. Check surface patches on which force and moments are to be calculated

ennova-cfd.com

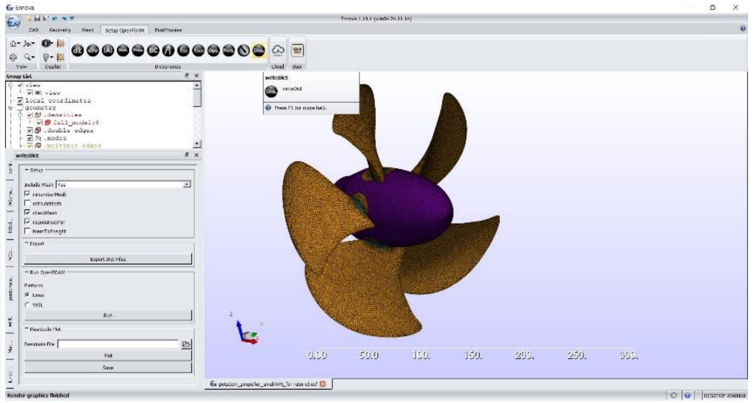
Export OpenFOAM Files



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

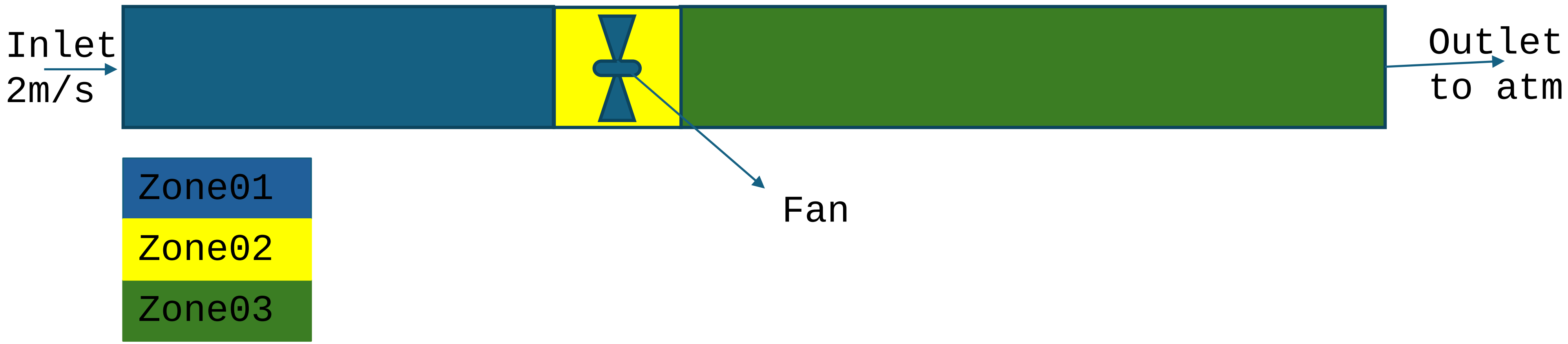


Setup

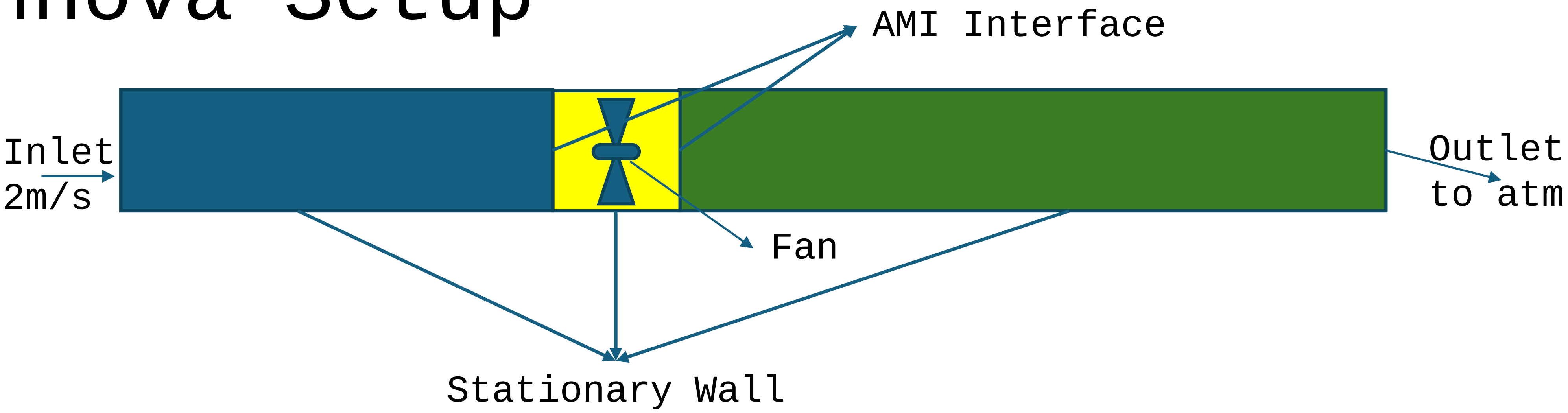
1. Include Mesh : Yes
   1. In Ennova we have generated polyhedral volume mesh for OpenFOAM case
2. Optional OpenFOAM operations to be performed before and after the simulation (check the boxes checked in this case)
   - renumberMesh : To reduce memory requirement for each step
   - checkMesh : To check 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

Case

Fan with Very small gap between blade tip and the wall of the domain



Ennova Setup



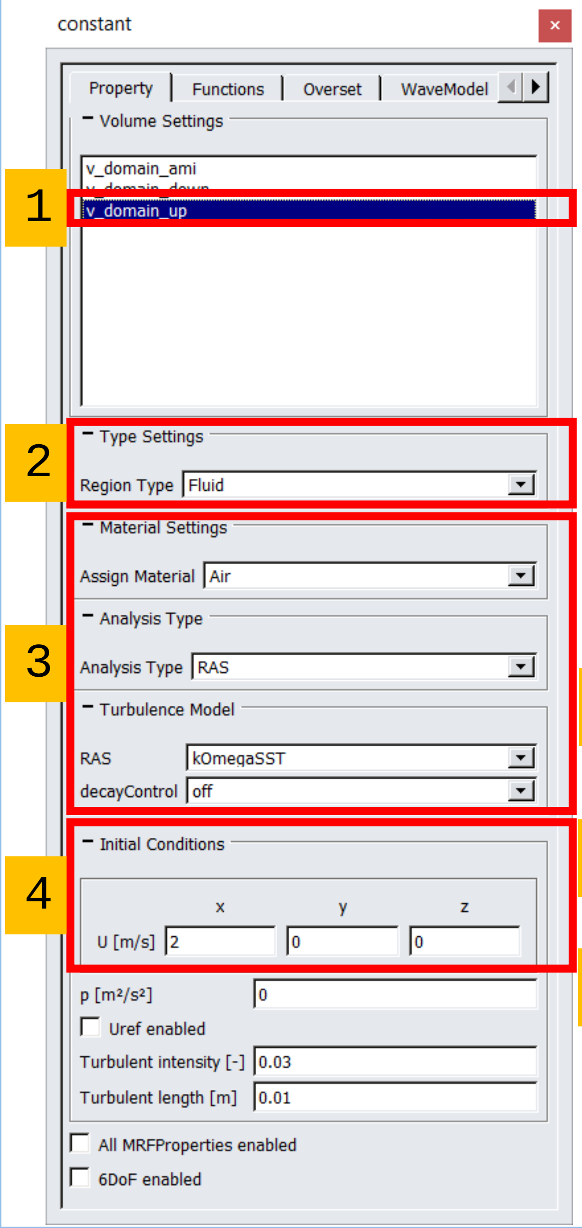
Zone01 – v_domain_up
Zone02 – v_domain_ami
Zone03 – v_domain_down

Note:
When there is very small gap between the rotating part (fan) and outer domain wall, consider entire zone (in yellow) as AMI

**

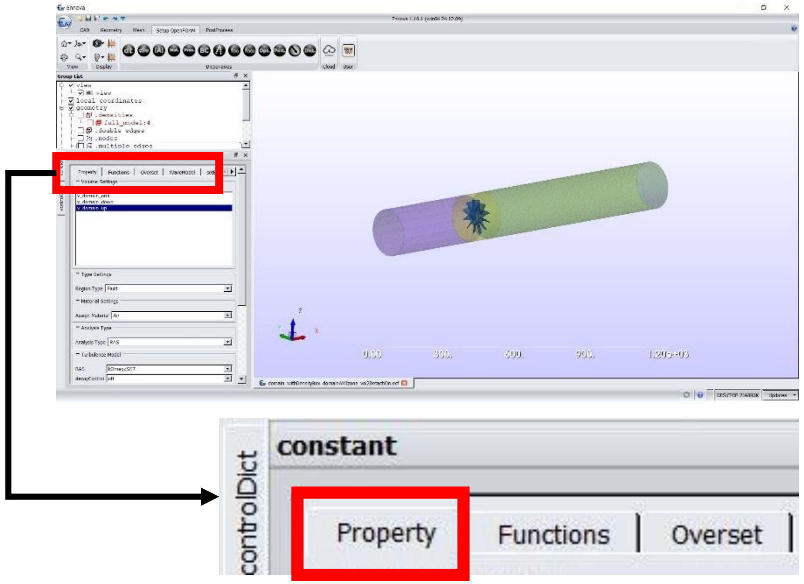
Steps of silde# 1-6 and 14-16 are same in current and previous case

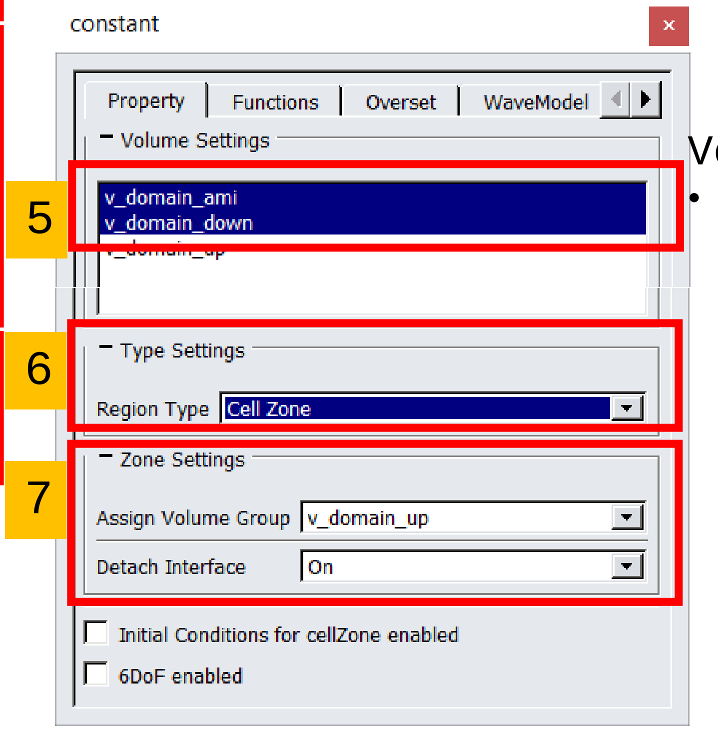
Property Settings



Volume Settings - Property

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

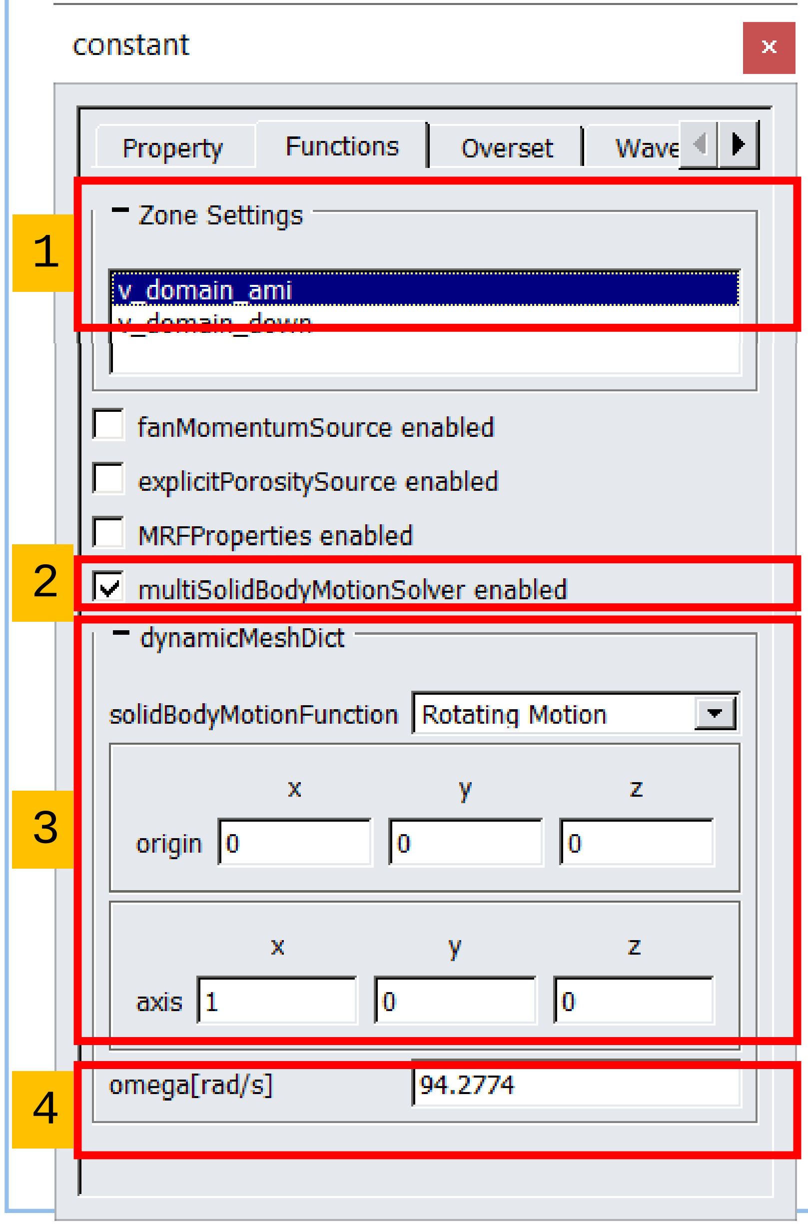


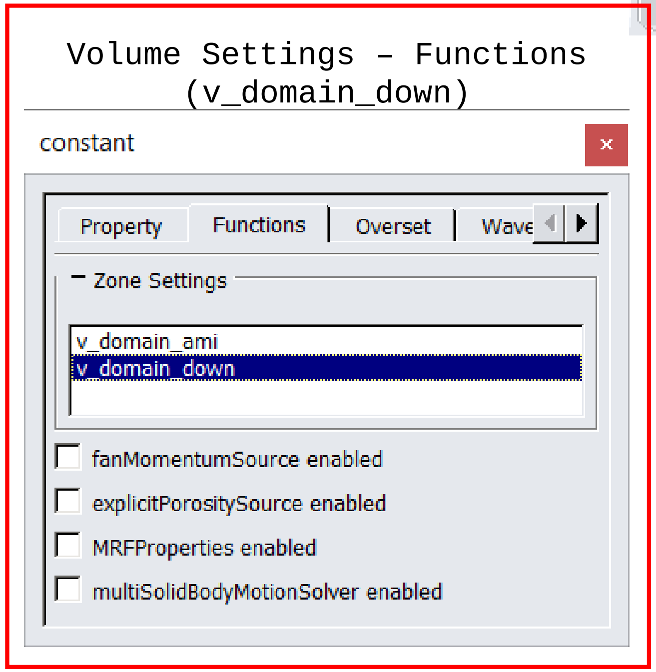


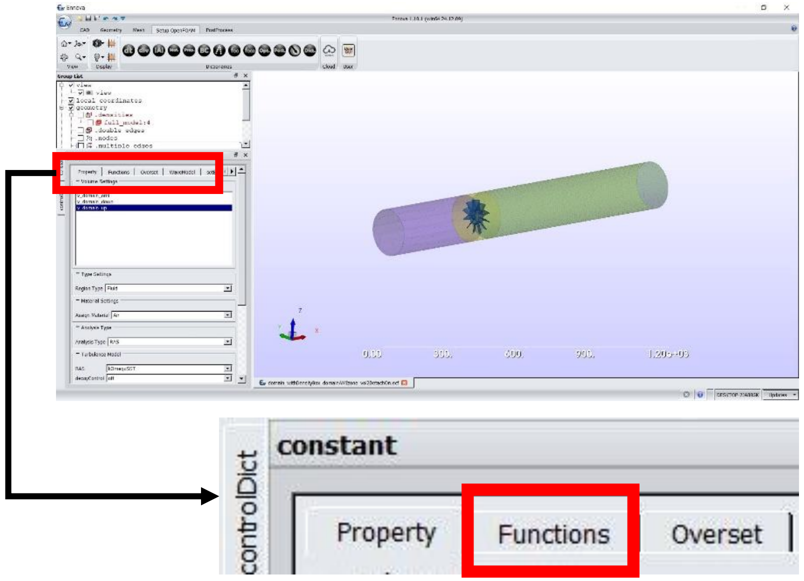
Volume Settings - Property

- Secondary Zone: cellZone
  5. Domain : v_domain_ami & v_domain_down
  6. Type Settings
     - Region Type : Cell Zone
       - Select it from drop down menu
  7. Zone Settings
     - Assign volume Group : v_domain_up
       - Assign it with the primary zone. In this case it is primary fluid domain v_domain_up
     - Detach Interface : On
       - Keep detach interface ON because this cellZone is rotating

Property Settings



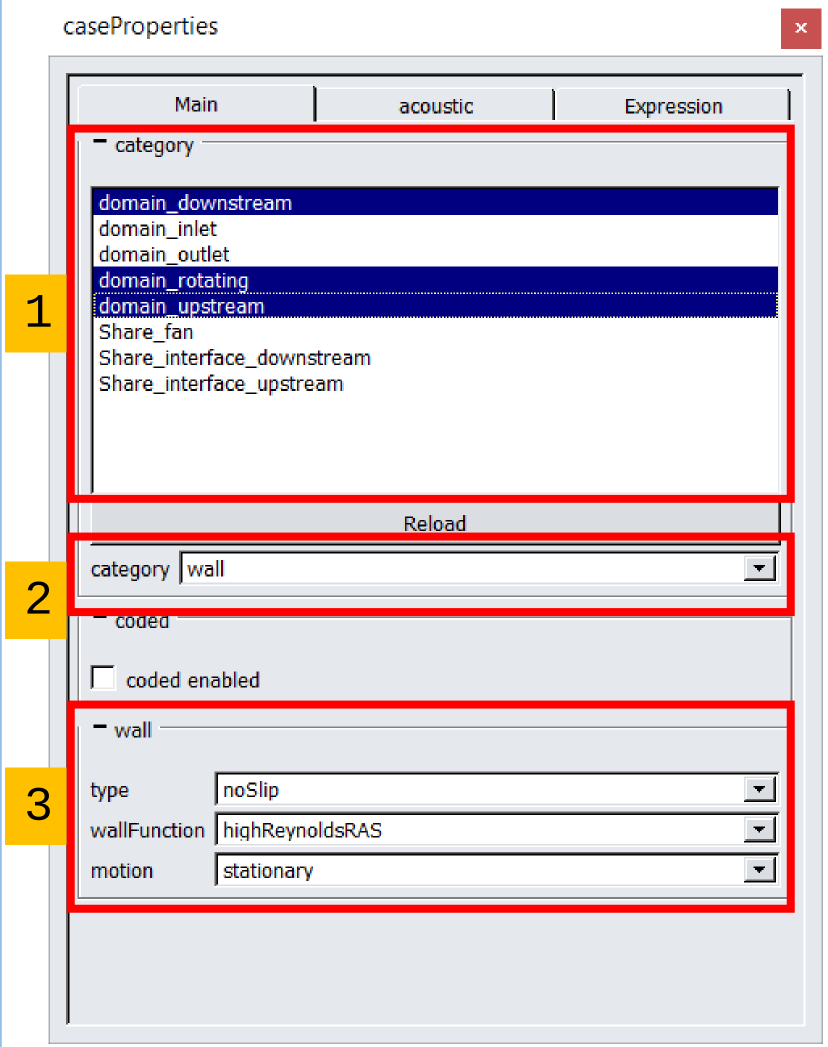


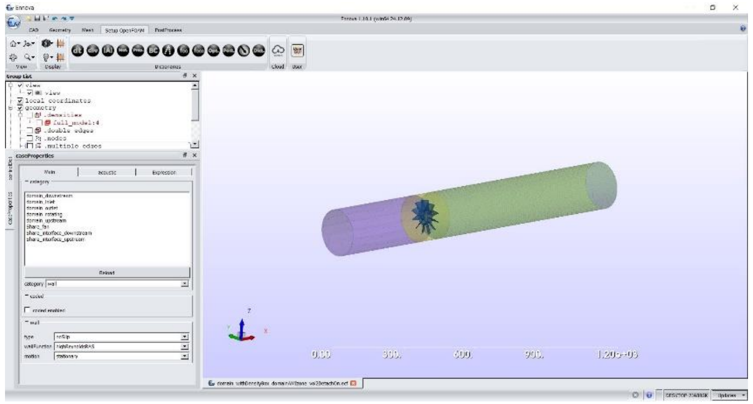


Volume Settings – Functions (v_domain_ami)

- Volume domain which is considered as a cellZone will be available here
  1. Zone Settings : v_ami
  2. Motion Solver
     - Check multiSolidBodyMotionSolver enabled
     - For mesh motion of v_ami
  3. Motion
     - solidBodyMotionFunction : Rotating Motion
     - Origin : 0, 0, 0
     - Axis : -1, 0, 0
       - Select rotation axis according to right hand thumb rule
  4. Propeller revolution in rad/s
     - Omega : 94.24778 (900 rpm)

Boundary Conditions

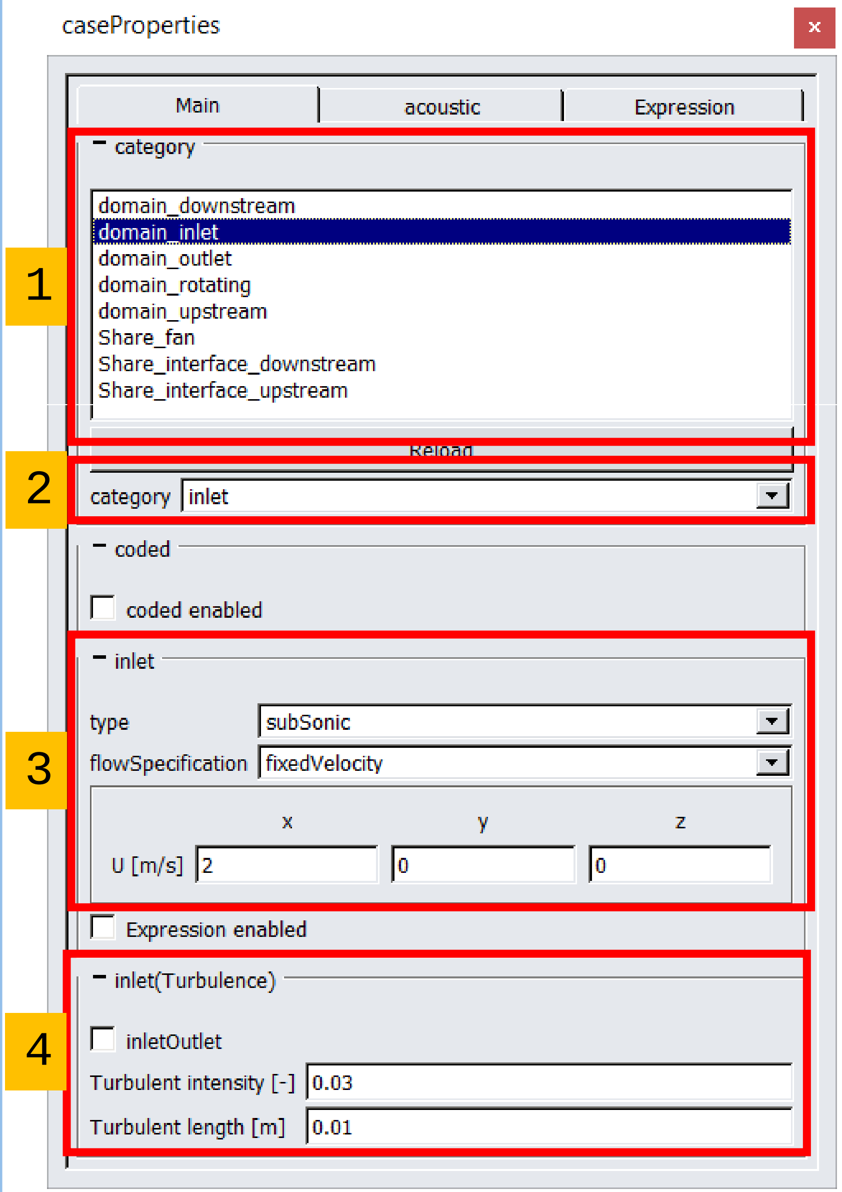


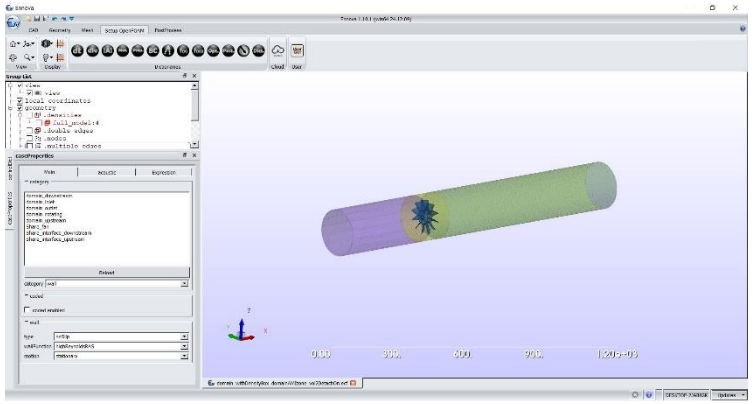


Boundary Conditions - Walls

1. Select all parts which are stationary walls
   - Make sure to select ***outer surface of zone02*** with other wall parts
2. Category : wall
   - Select BC type as wall for domain and shaft
3. Settings for wall type BC
   - Keep defaults
   - Make sure that motion is selected as stationary

Boundary Conditions

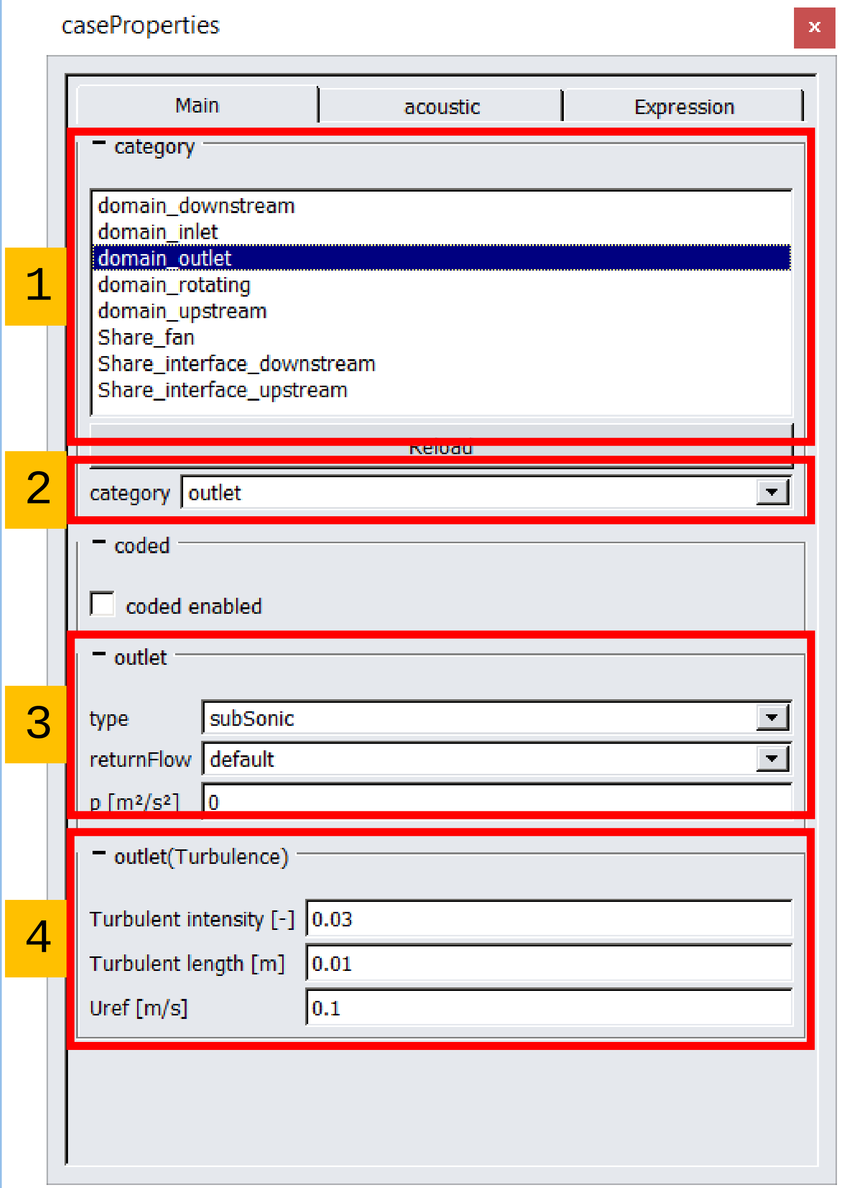


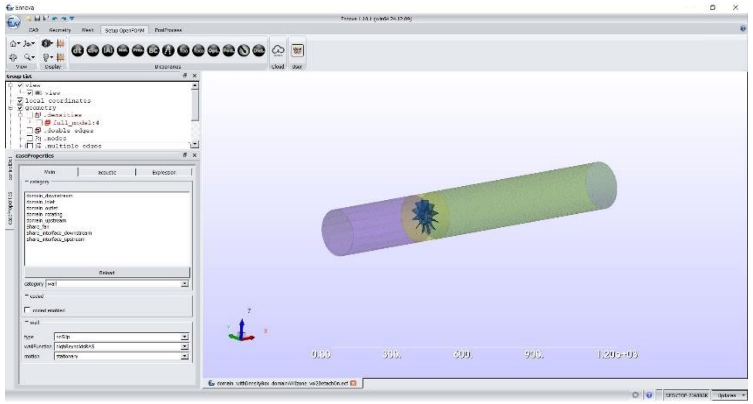


Boundary Conditions - inlet

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

Boundary Conditions

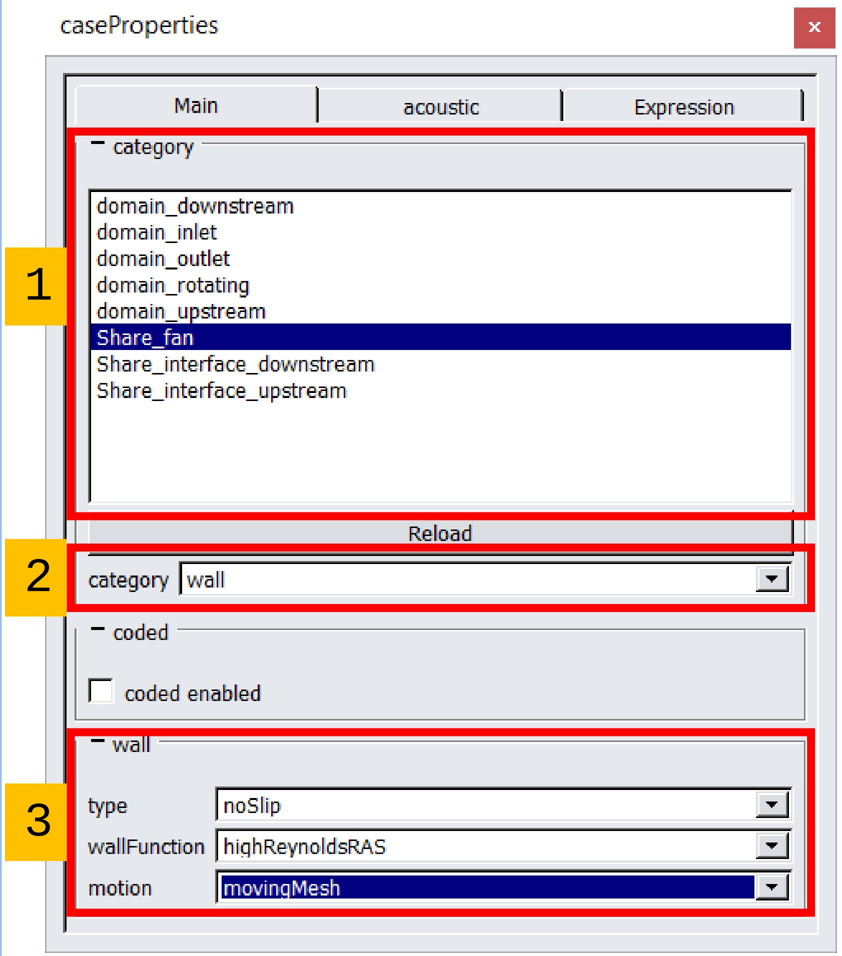


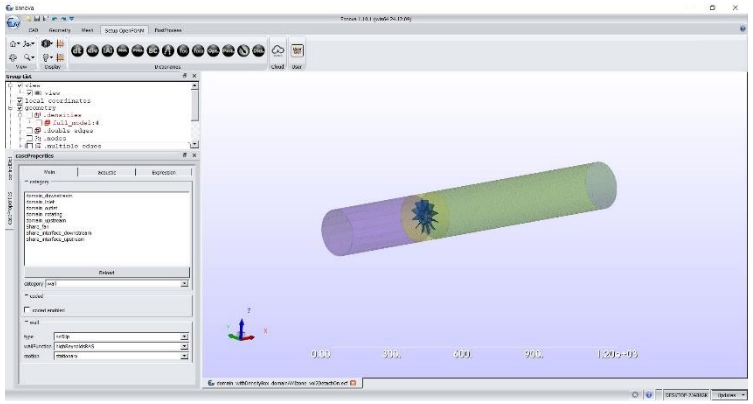


Boundary Conditions - outlet

1. Select part which is outlet to the domain
   - Here it is domain_outlet
2. Category : outlet
   - Select BC type as inlet for domain_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

Boundary Conditions

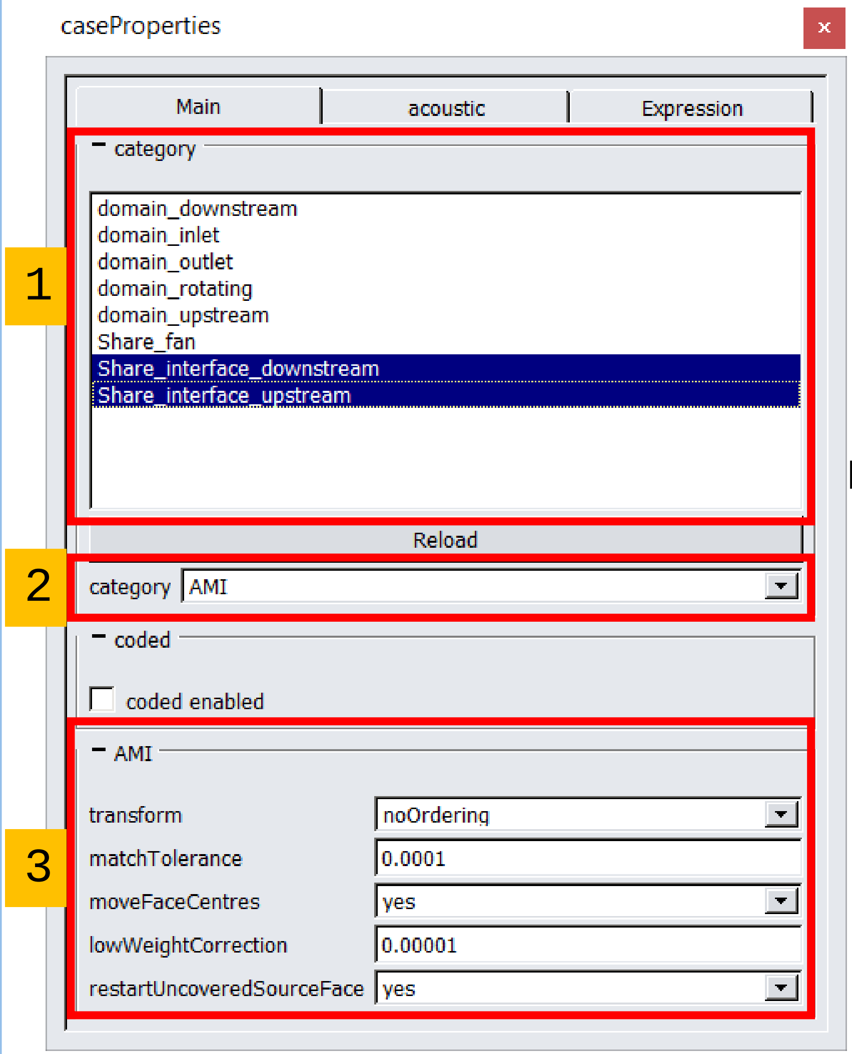


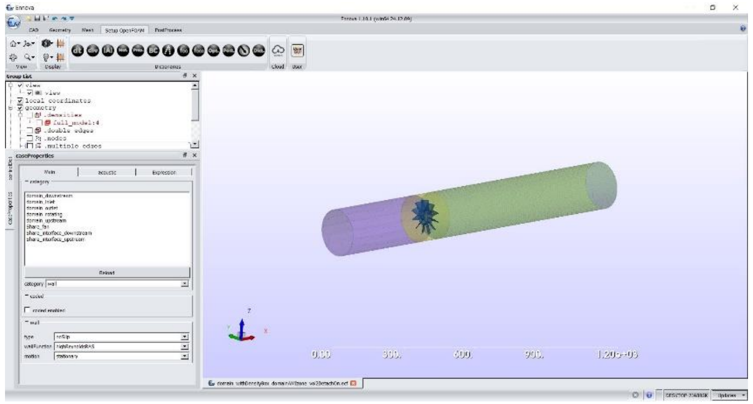


Boundary Conditions - Moving Wall

1. Select parts which are moving
   - Here it is fan
2. Category : wall
   - Select BC type as wall for all propeller parts
3. Settings for moving wall type BC
   - Type : noSlip (default)
   - wallFunction : highReynoldsRAS (default)
   - Motion : ***movingMesh***
     - Fan should be completely inside AMI domain
     - Hence, this should be given as moving mesh motion

Boundary Conditions





Boundary Conditions - AMI

1. Select parts which are interface for AMI and fluid domain
    - Here we have,
        - share_interface_downstream
        - Share_interface_upstream
        - Ennova by defaults gives prefix to parts which share 2 volumes
2. Category : AMI
    - Select BC type as AMI for share_interface_downstream and share_interface_upstream parts
3. Settings for AMI type BC
    - Keep everything as defaults

Thank You