



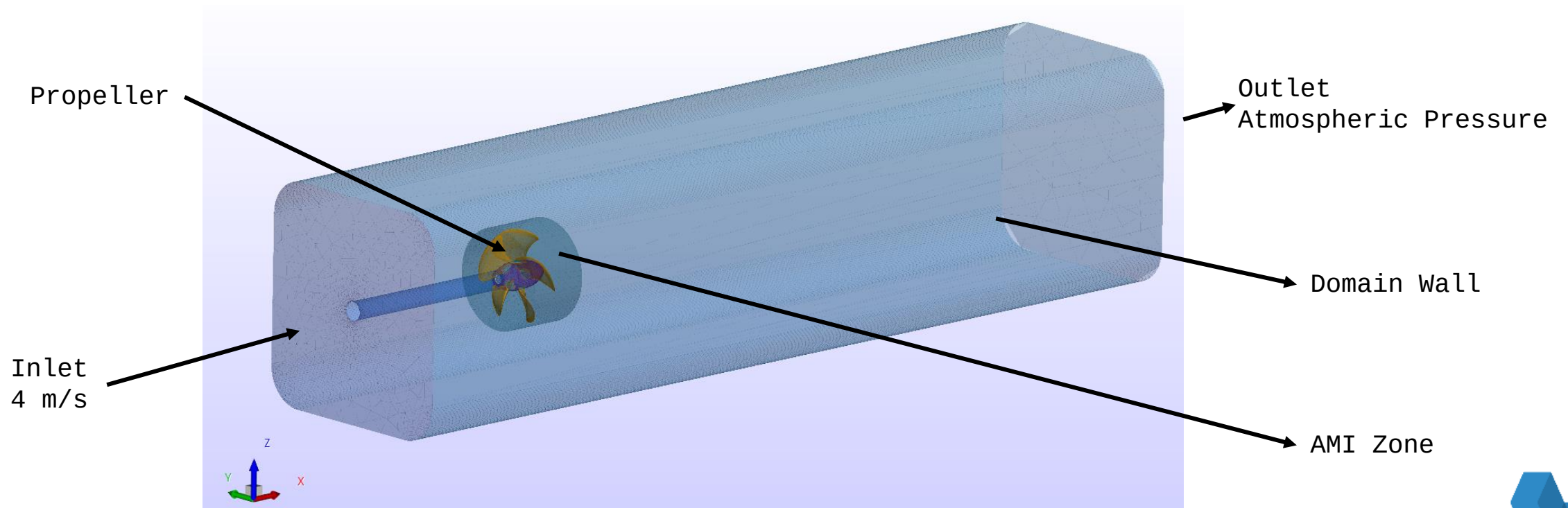
Ennova Technologies

Potsdam Propeller Test Case

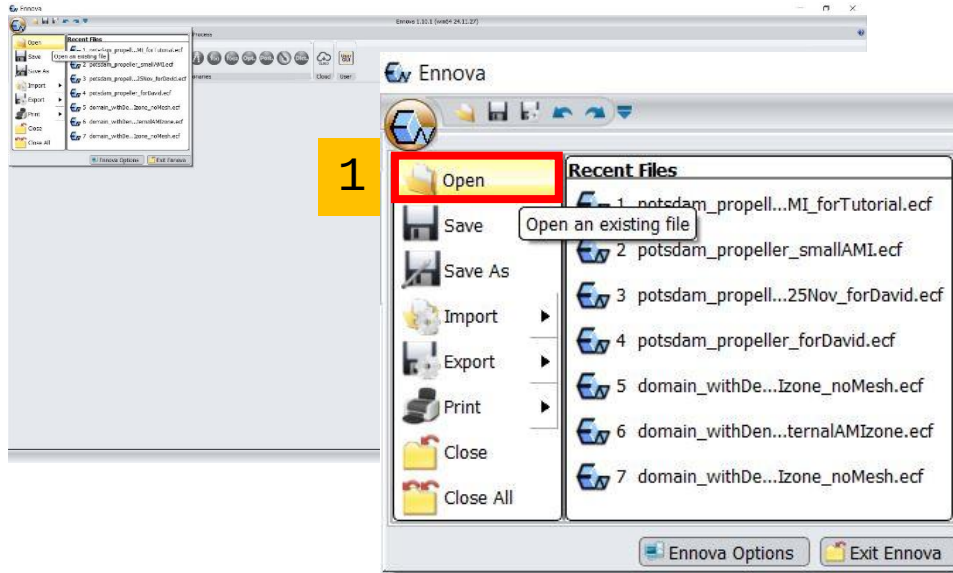
OpenFOAM case setup with Ennova

Boundary Conditions

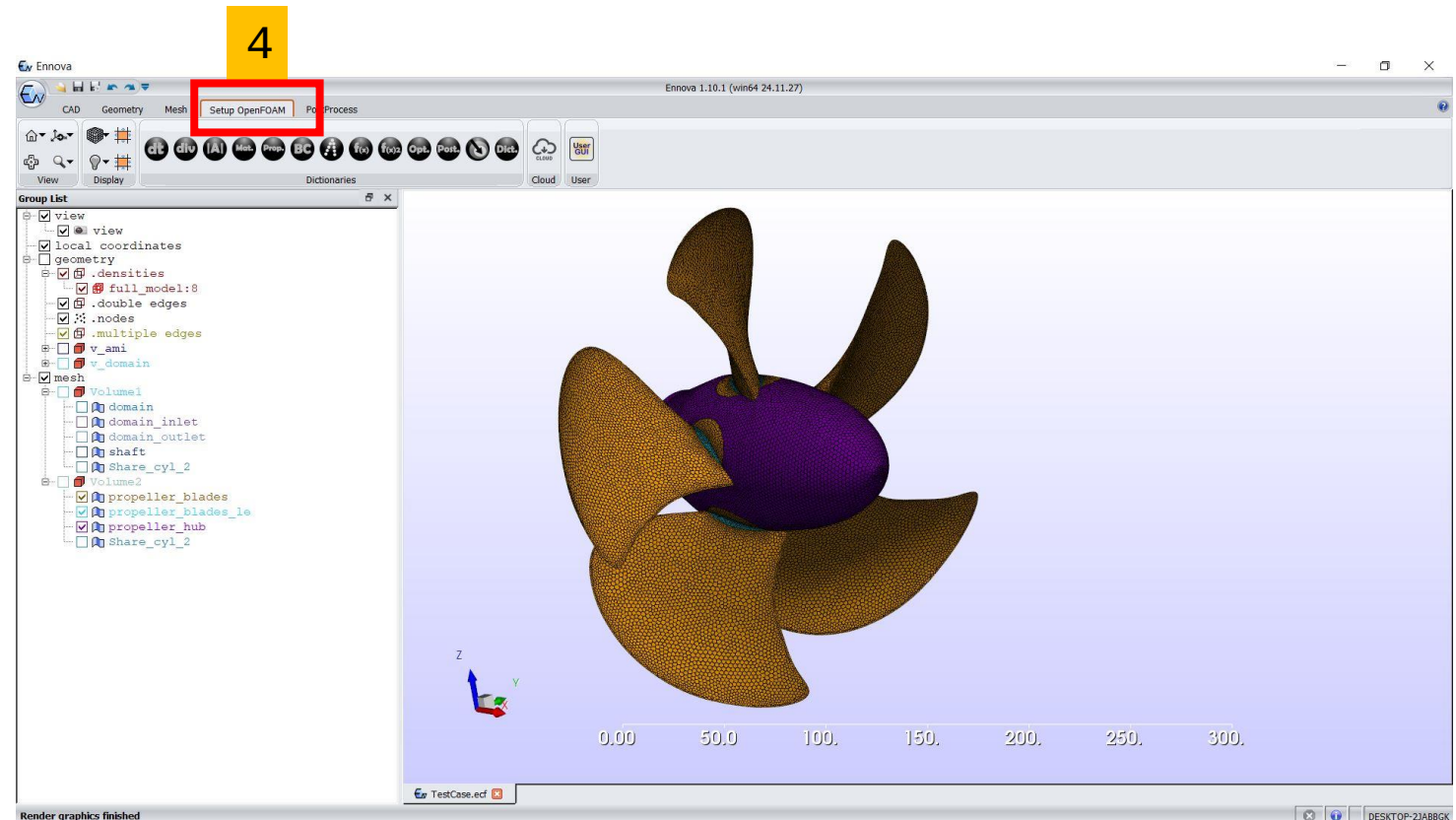
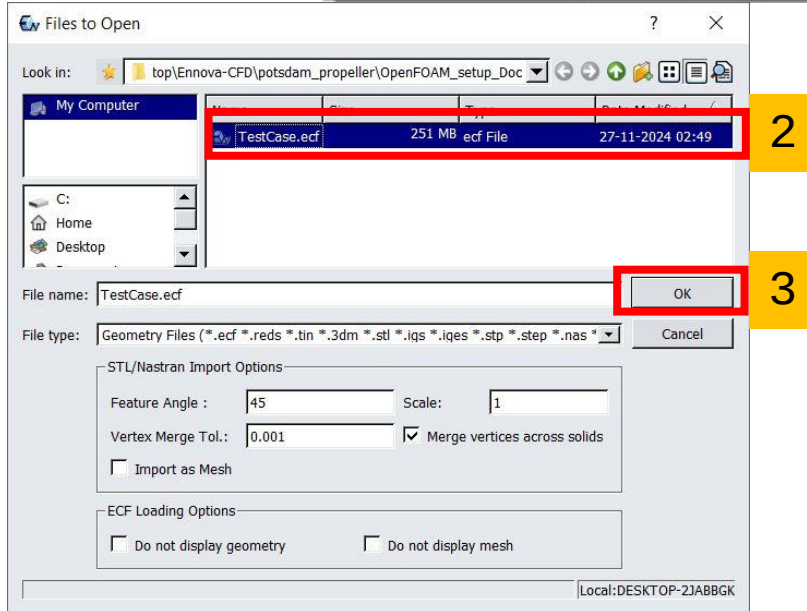
- | | | | |
|----------------------|------------------|------------------------|-------------------------------|
| • Mean flow speed | : 4 m/s | • Reference density | : 997.71 kg/m ³ |
| • Rotation speed | : 900 RPM | • Dynamic viscosity | : 9.559×10^{-4} Pa·s |
| • Flow model | : incompressible | • Turbulence model | : k ω SST |
| • Mesh size | : 1.53M cells | • Simulation type | : Propeller |
| • Simulation point | : 1 (J = 1.0667) | • Time management | : Transient |
| • Fluid | : Water | • Rotational Direction | : (-1, 0, 0) |
| • Reference pressure | : 1 atm | | |



Reading ECF (Volume Mesh File)



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



Solver Settings



controlDict

Main ← Modules

1 ☐ Old version Enabled

2 Version v2312

2 ☐ Overset Mesh Enabled

Solver Type incompressible

Solver pimpleDyMFOam

☐ energyTransport Enabled

☐ scalarTransport Enabled

3 decomposeParDict

method scotch

numberOfSubdomains 8

4 controlDict

startFrom latestTime

startTime 0

stopAt endTime

endTime 2

deltaT 1e-4

writeControl adjustableRunTime

writeInterval 0.1

purgeWrite 0

runTimeModifiable on

adjustTimeStep off

maxCo 0.5

maxAlphaCo 1.0

maxDeltaT 1.0

controlDict Options

writeFormat ascii

writePrecision 12

writeCompression on

timeFormat general

timePrecision 12

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

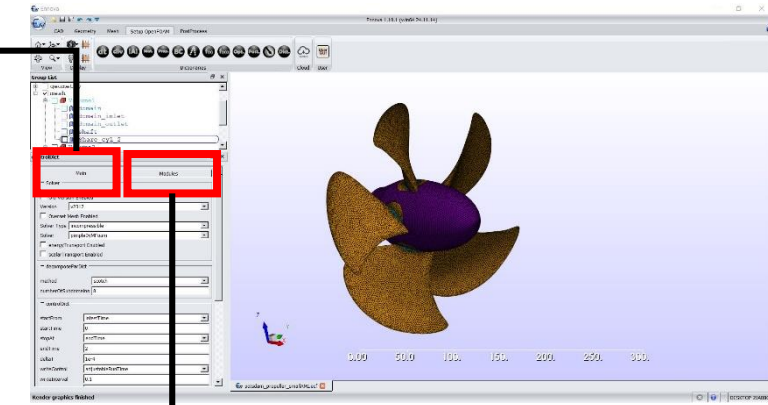
- Method : Scotch
 - For equal number of cell distribution across the processors
- Cores : 8
 - No. of cores available to run case in parallel
 - Optimum is 100K cells/core

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



controlDict

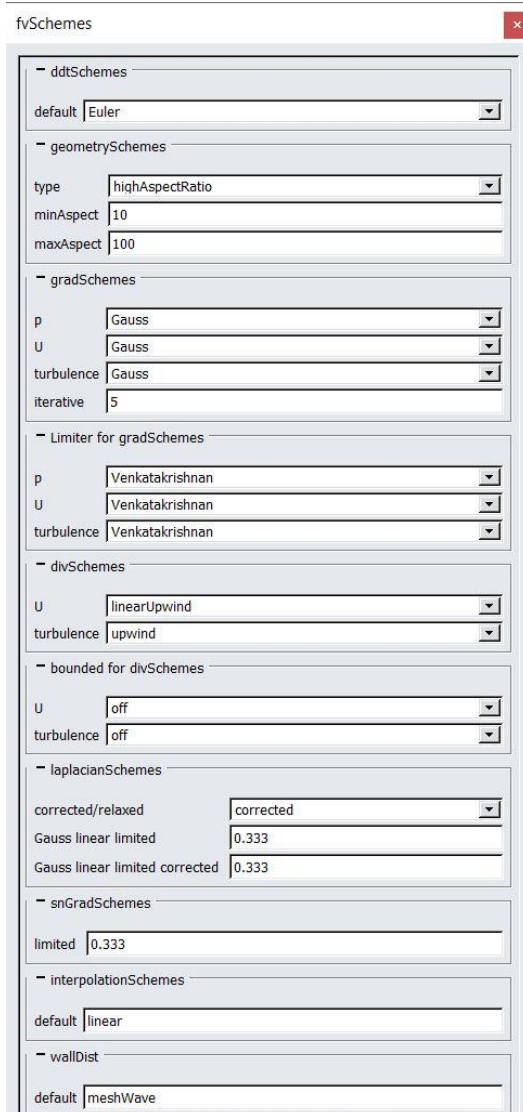
Main ← Modules

5

	Version	FOAM_INST_DIR
1	v2312	/opt/OpenFOAM/OpenFOAM-v2312
2	v2306	/opt/OpenFOAM/OpenFOAM-v2306
3	v2212	/opt/OpenFOAM/OpenFOAM-v2212
4	v2206	/opt/OpenFOAM/OpenFOAM-v2206
		/opt/OpenFOAM/OpenFOAM-

Insert Row Remove Row

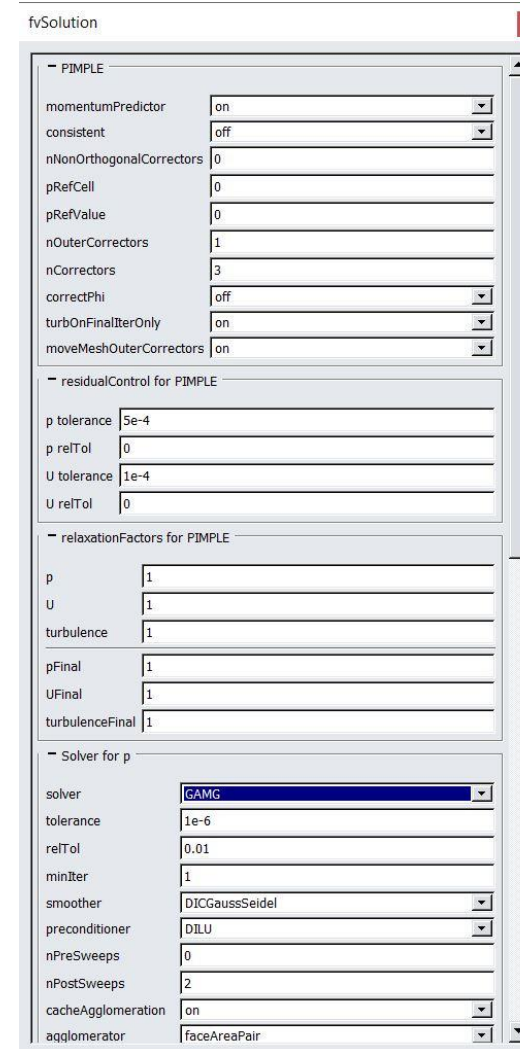
Schemes Settings



fvScheme settings window. The window is titled 'fvScheme' and contains several sections for configuring numerical schemes. The 'default' dropdown is set to 'Euler'. The 'geometrySchemes' section has 'type' set to 'highAspectRatio', 'minAspect' set to '10', and 'maxAspect' set to '100'. The 'gradSchemes' section has 'p' set to 'Gauss', 'U' set to 'Gauss', 'turbulence' set to 'Gauss', and 'iterative' set to '5'. The 'limiter for gradSchemes' section has 'p' set to 'Venkatakrishnan', 'U' set to 'Venkatakrishnan', and 'turbulence' set to 'Venkatakrishnan'. The 'divSchemes' section has 'U' set to 'linearUpwind' and 'turbulence' set to 'upwind'. The 'bounded for divSchemes' section has 'U' set to 'off' and 'turbulence' set to 'off'. The 'laplacianSchemes' section has 'corrected/relaxed' set to 'corrected', 'Gauss linear limited' set to '0.333', and 'Gauss linear limited corrected' set to '0.333'. The 'snGradSchemes' section has 'limited' set to '0.333'. The 'interpolationSchemes' section has 'default' set to 'linear'. The 'wallDist' section has 'default' set to 'meshWave'.

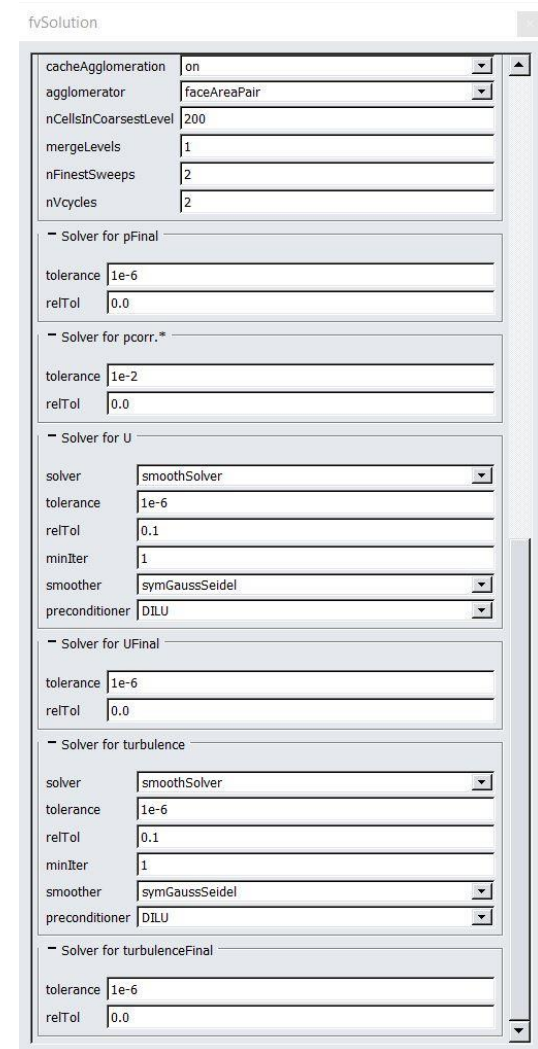
fvScheme

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



fvSolution settings window. The window is titled 'fvSolution' and contains several sections for configuring simulation parameters. The 'PIMPLE' section has 'momentumPredictor' set to 'on', 'consistent' set to 'off', 'nNonOrthogonalCorrectors' set to '0', 'pRefCell' set to '0', 'pRefValue' set to '0', 'nOuterCorrectors' set to '1', 'nCorrectors' set to '3', 'correctPhi' set to 'off', 'turbOnFinalIterOnly' set to 'on', and 'moveMeshOuterCorrectors' set to 'on'. The 'residualControl for PIMPLE' section has 'p tolerance' set to '5e-4', 'p relTol' set to '0', 'U tolerance' set to '1e-4', and 'U relTol' set to '0'. The 'relaxationFactors for PIMPLE' section has 'p' set to '1', 'U' set to '1', and 'turbulence' set to '1'. The 'Solver for p' section has 'pFinal' set to '1', 'UFinal' set to '1', and 'turbulenceFinal' set to '1'. The 'Solver for p' section has 'solver' set to 'GAMG', 'tolerance' set to '1e-6', 'relTol' set to '0.01', 'minIter' set to '1', 'smoother' set to 'DICGaussSeidel', 'preconditioner' set to 'DILU', 'nPreSweeps' set to '0', 'nPostSweeps' set to '2', 'cacheAgglomeration' set to 'on', and 'agglomerator' set to 'faceAreaPair'.

fvSolution



fvSolution settings window. The window is titled 'fvSolution' and contains several sections for configuring simulation parameters. The 'cacheAgglomeration' section has 'cacheAgglomeration' set to 'on', 'agglomerator' set to 'faceAreaPair', 'nCellsInCoarsestLevel' set to '200', 'mergeLevels' set to '1', 'nFinestSweeps' set to '2', and 'nVcycles' set to '2'. The 'Solver for pFinal' section has 'tolerance' set to '1e-6' and 'relTol' set to '0.0'. The 'Solver for pcorr.*' section has 'tolerance' set to '1e-2' and 'relTol' set to '0.0'. The 'Solver for U' section has 'solver' set to 'smoothSolver', 'tolerance' set to '1e-6', 'relTol' set to '0.1', 'minIter' set to '1', 'smoother' set to 'symGaussSeidel', and 'preconditioner' set to 'DILU'. The 'Solver for UFinal' section has 'tolerance' set to '1e-6' and 'relTol' set to '0.0'. The 'Solver for turbulence' section has 'solver' set to 'smoothSolver', 'tolerance' set to '1e-6', 'relTol' set to '0.1', 'minIter' set to '1', 'smoother' set to 'symGaussSeidel', and 'preconditioner' set to 'DILU'. The 'Solver for turbulenceFinal' section has 'tolerance' set to '1e-6' and 'relTol' set to '0.0'.

Material Settings

Materials

Material | Polynomial | species

Material Settings

Name of Material

1 water

Insert Row Remove Row

Material Type

Material Type FLUID

transportProperties

transportModel Newtonian

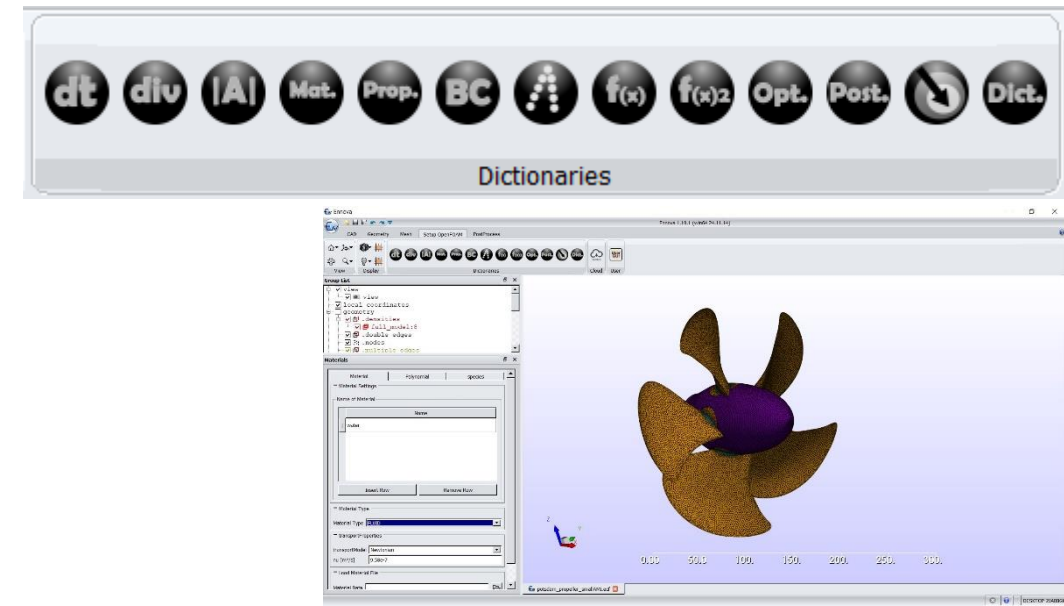
nu [m²/s] 9.58e-7

Load Material File

Material Data

Load Material data

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) : $9.58e^{-7}(\text{m}^2/\text{s})$ updated value for water



Property Settings

constant

Property | Functions | Oversight | Wave

Volume Settings

1. **v_domain**

2. Type Settings

Region Type Fluid

3. Material Settings

Assign Material water

Analysis Type

Analysis Type RAS

Turbulence Model

RAS kOmegaSST

decayControl off

4. Initial Conditions

x y z

U [m/s] 4 0 0

p [m²/s²] 0

☐ Uref enabled

Turbulent intensity [-] 0.03

Turbulent length [m] 0.01

☐ All MRFPProperties enabled

☐ 6DoF enabled

Volume Settings - Property

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

constant

Property | Functions | Oversight | Wave

Volume Settings

5. **v_ami**

6. Type Settings

Region Type Cell Zone

7. Zone Settings

Assign Volume Group v_domain

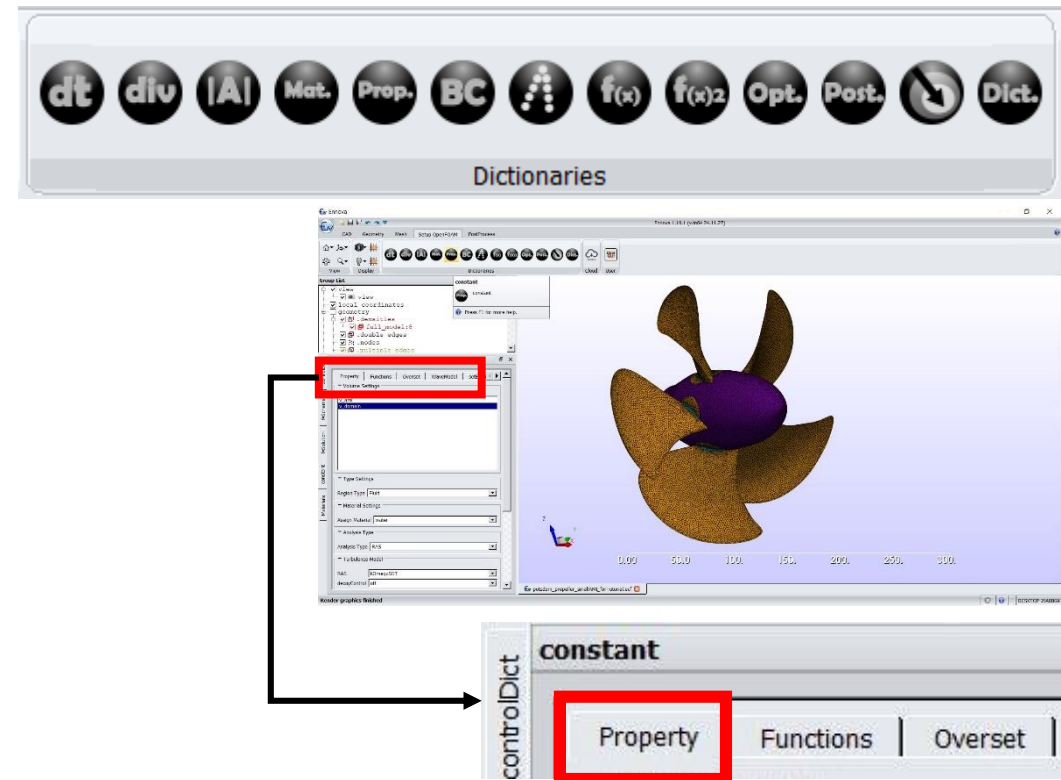
Detach Interface On

☐ Initial Conditions for cellZone enabled

☐ 6DoF enabled

Volume Settings - Property

- Secondary Zone: AMI
 - Domain : v_ami
 - Type Settings
 - Region Type : Cell Zone
 - Select it from drop down menu
 - 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

constant

Property Functions Oversight Wave

1 **Zone Settings**

v_ami

☐ fanMomentumSource enabled

☐ explicitPorositySource enabled

☐ MRFPProperties enabled

2 ☒ multiSolidBodyMotionSolver enabled

dynamicMeshDict

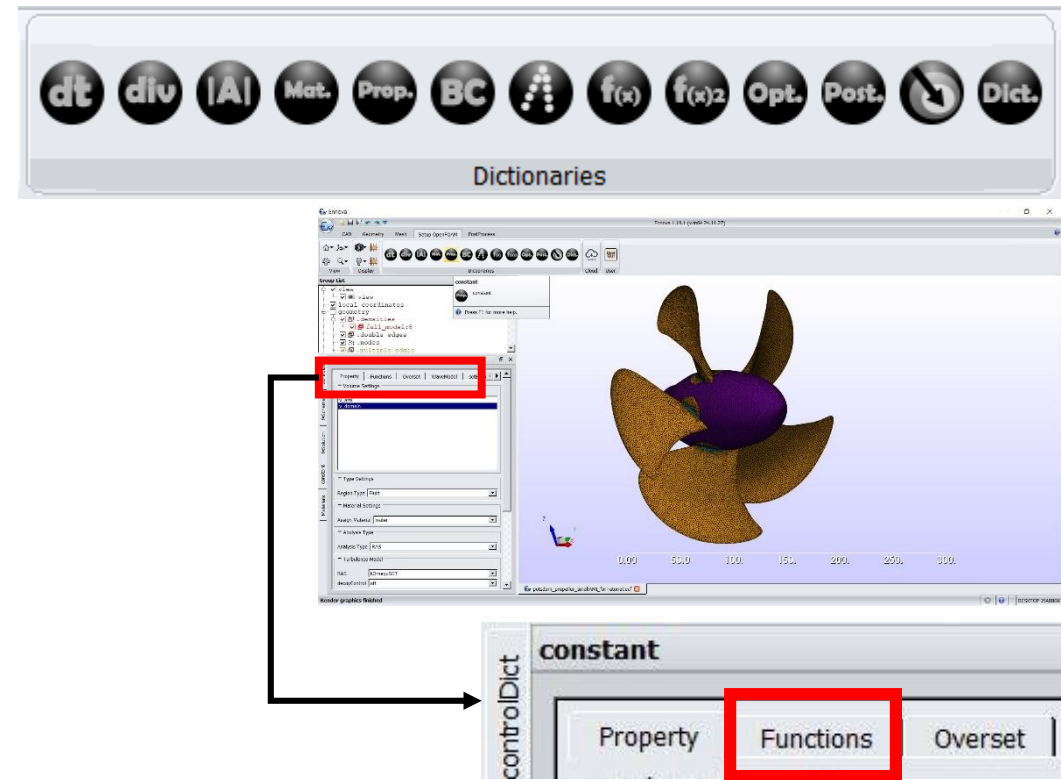
solidBodyMotionFunction Rotating Motion

3

	x	y	z
origin	0	0	0
axis	-1	0	0

4

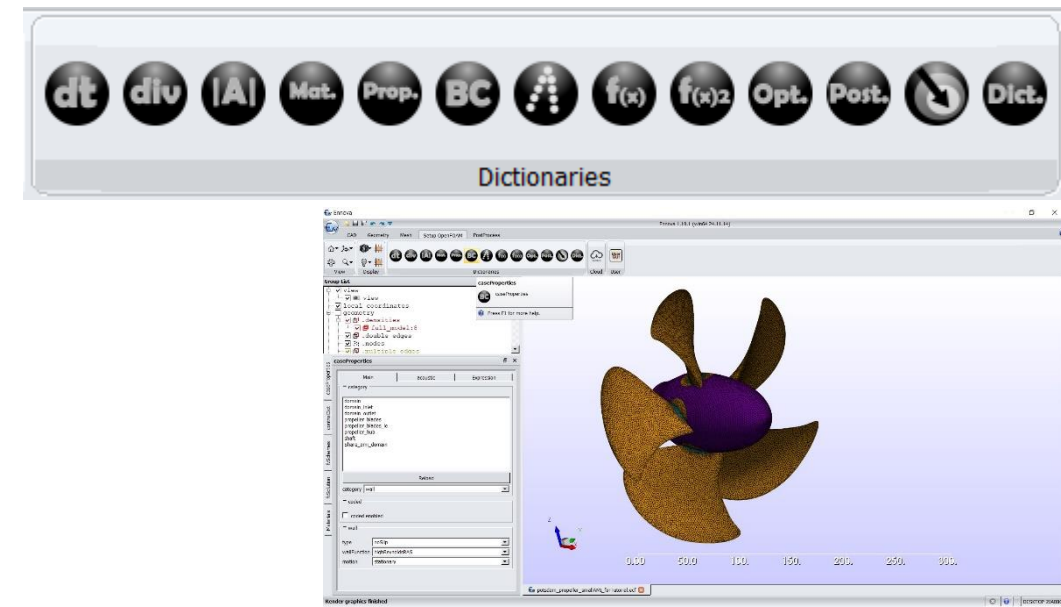
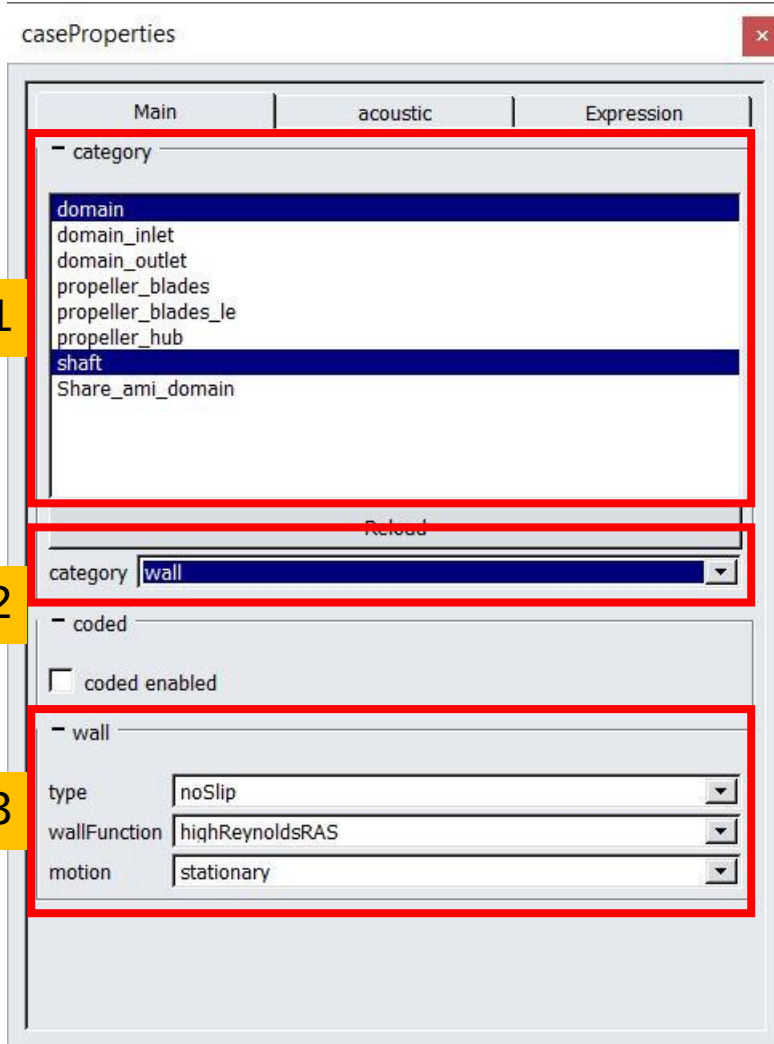
omega[rad/s] 94.24778



Volume Settings - Functions

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

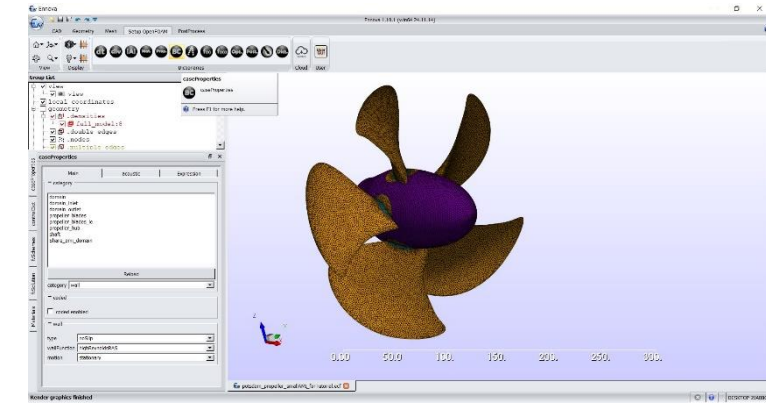
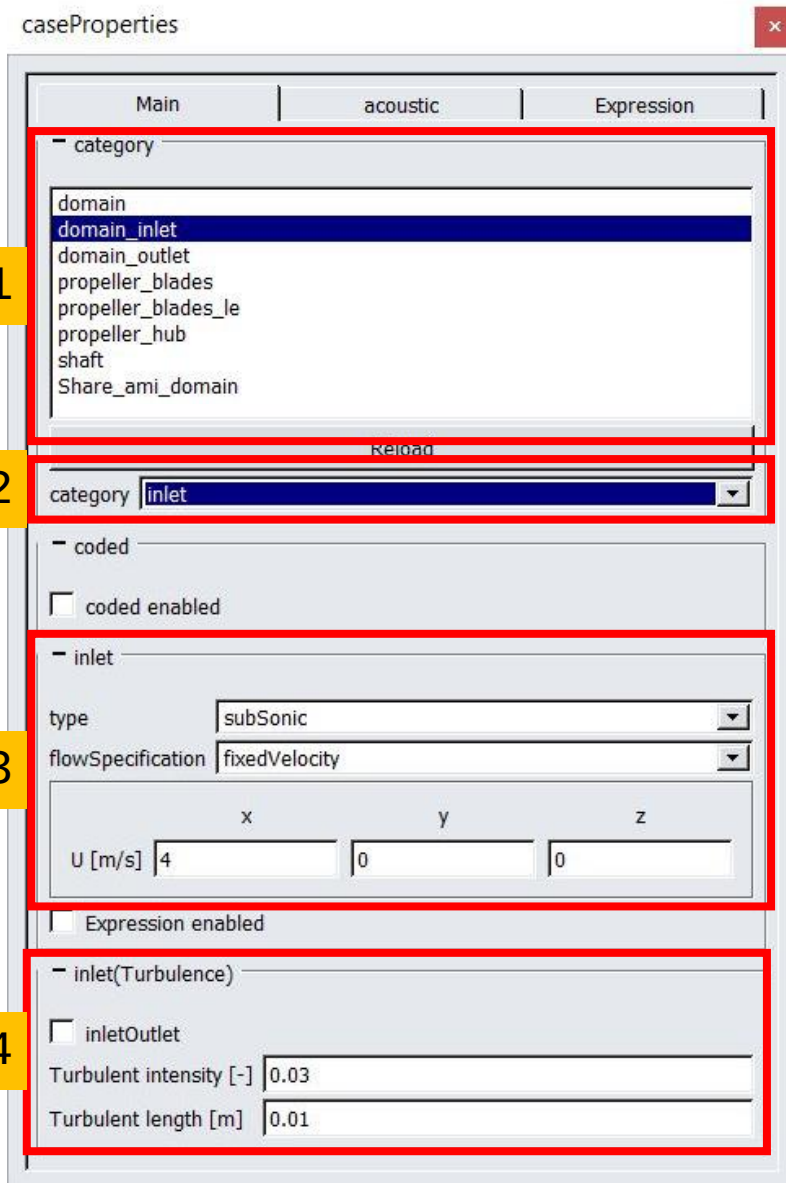
Boundary Conditions



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

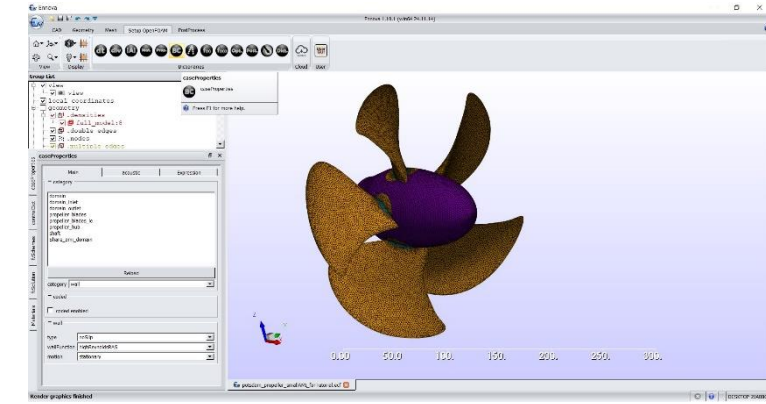
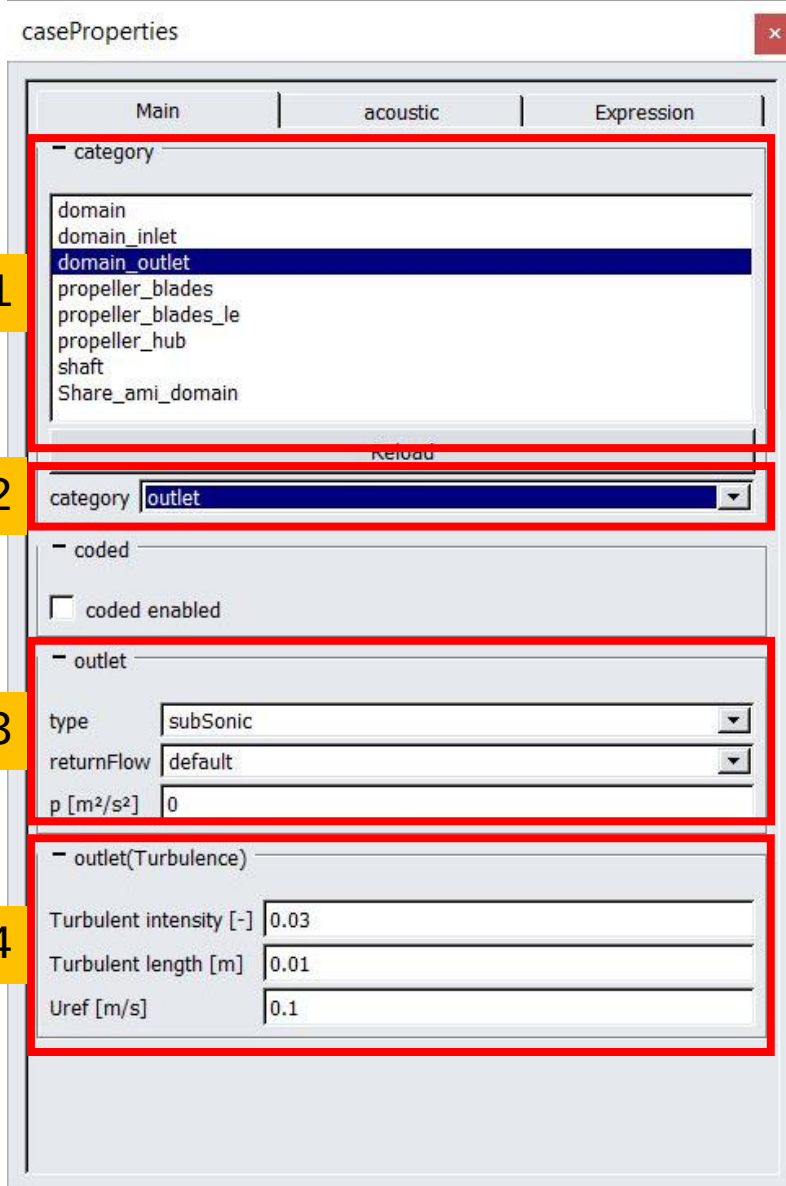
Boundary Conditions



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 fixedVelocity 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

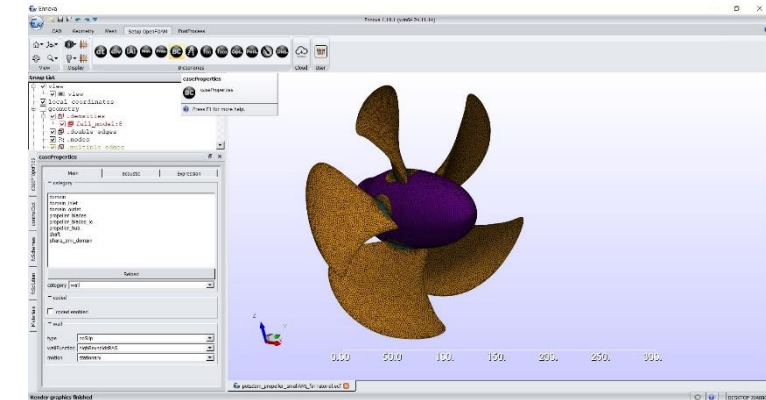
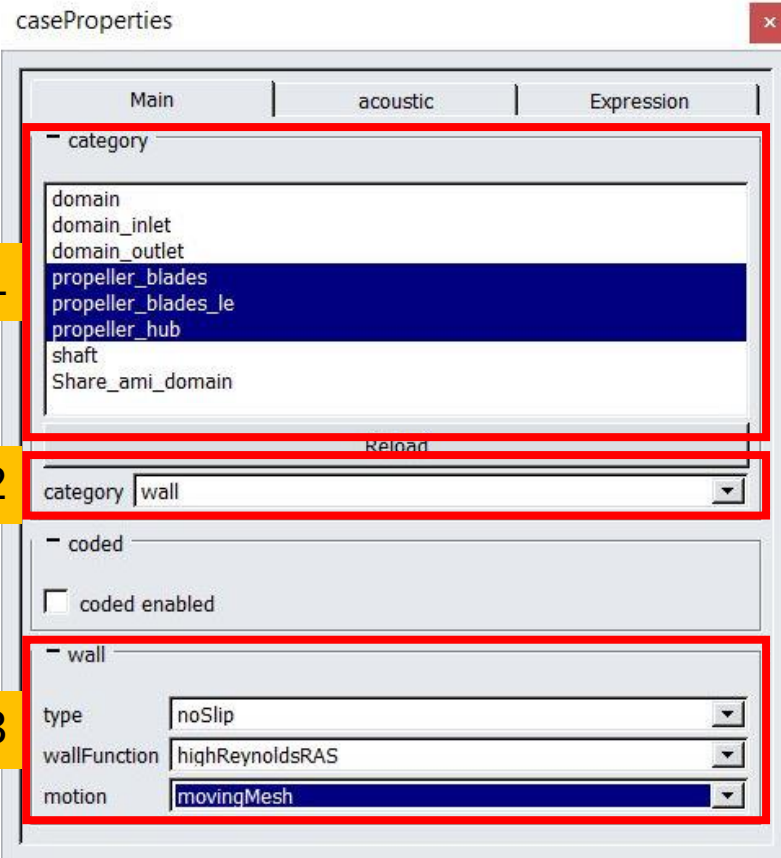
3. Settings for outlet type BC

- 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

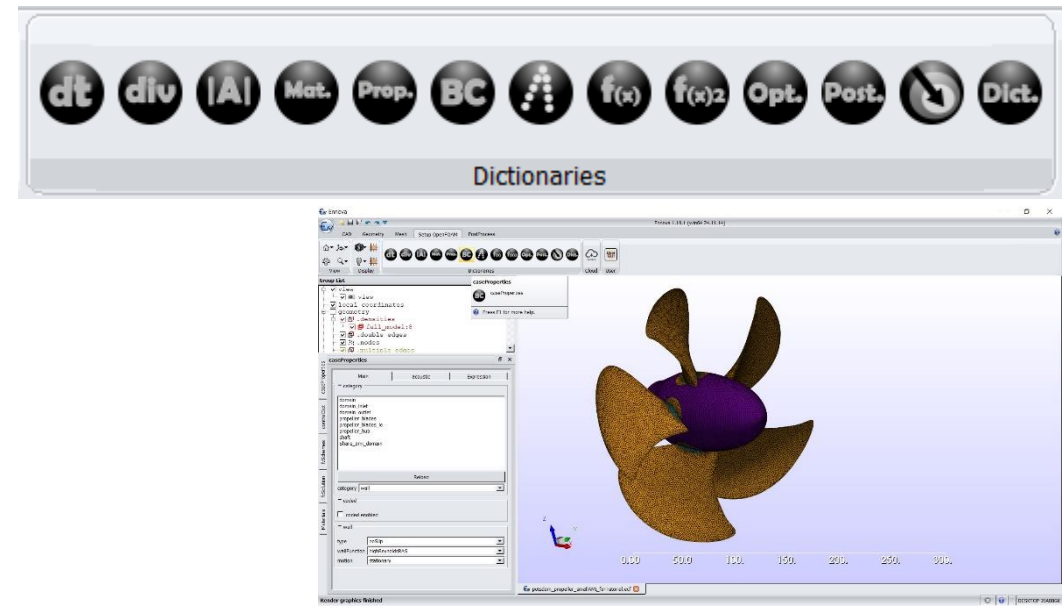
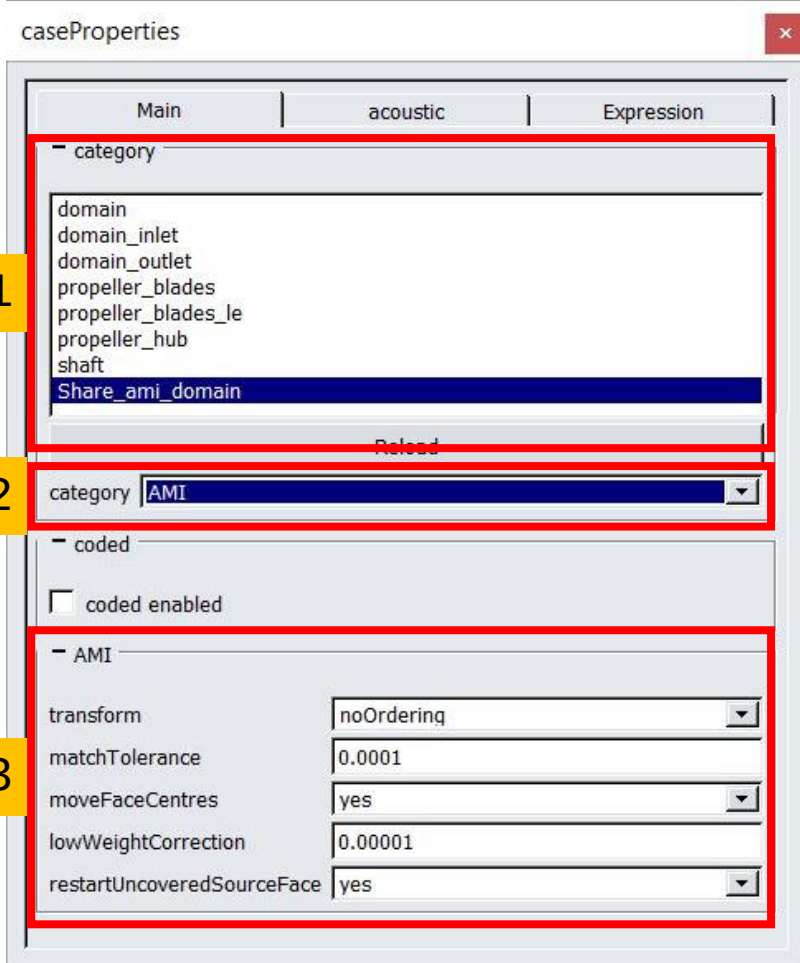
Boundary Conditions



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

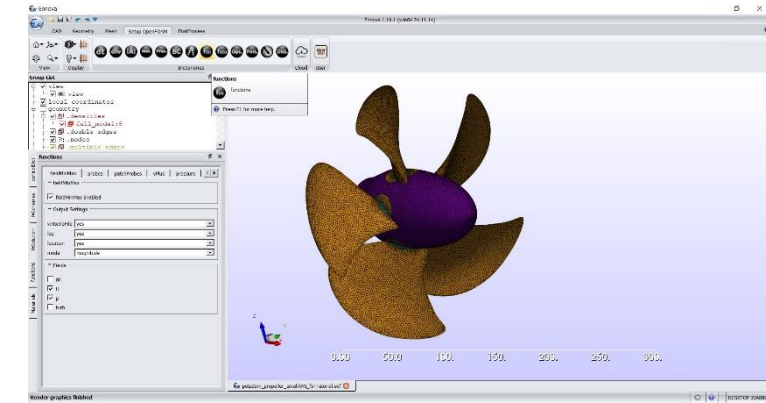
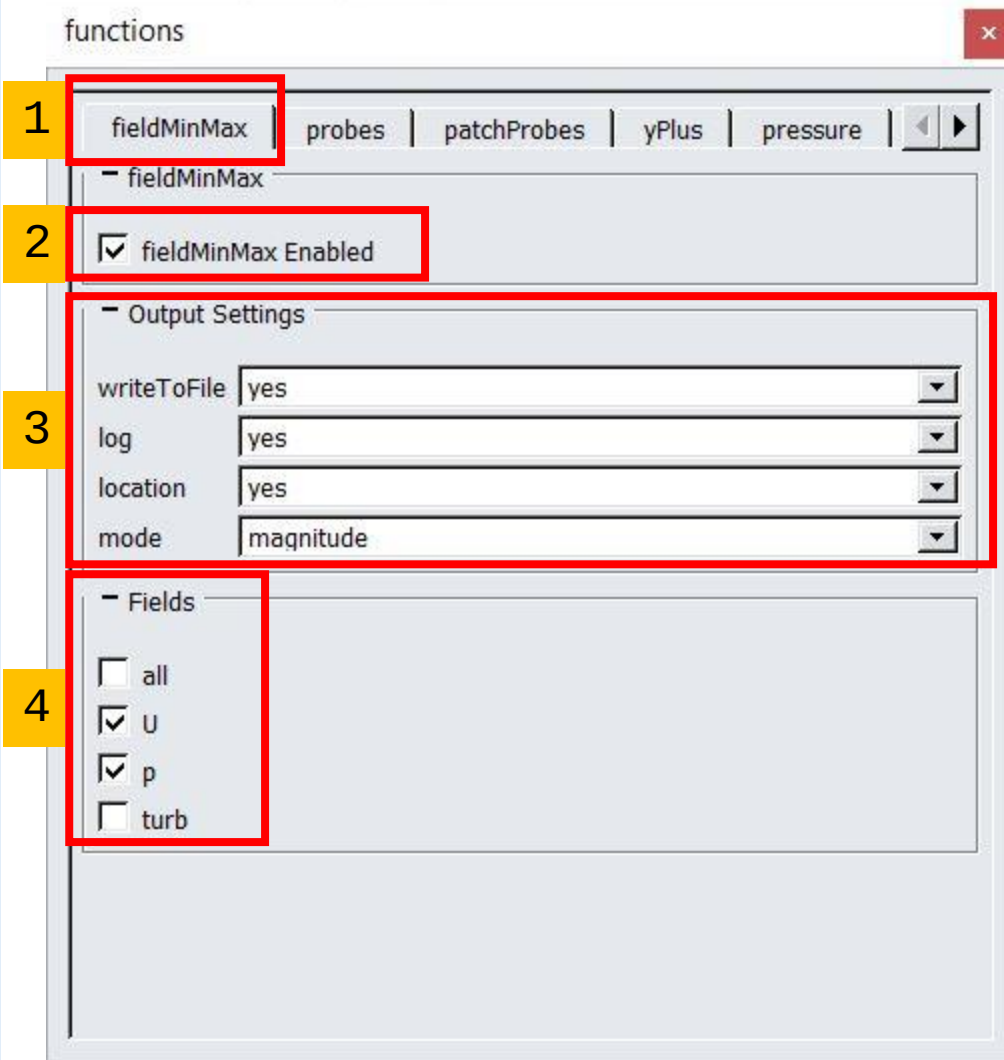
Boundary Conditions



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



functions

patchProbes | yPlus | pressure | fieldA **1** forces

forces and forceCoeffs

2 ☒ forces Enabled
☐ forceCoeffs Enabled

3 Output Settings

Name of forces

	Name
1	propeller

Insert Row Remove Row

4 writeToFile yes
log yes
writeFields yes

5 CofR

6 rho 997.71
pRef 0.0
porosity no

☐ Bin Enabled

7 Select Patches

- ☐ domain
- ☐ domain_inlet
- ☐ domain_outlet
- ☒ propeller_blades
- ☒ propeller_blades_le
- ☒ propeller_hub
- ☐ shaft
- ☐ Share_ami_domain

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

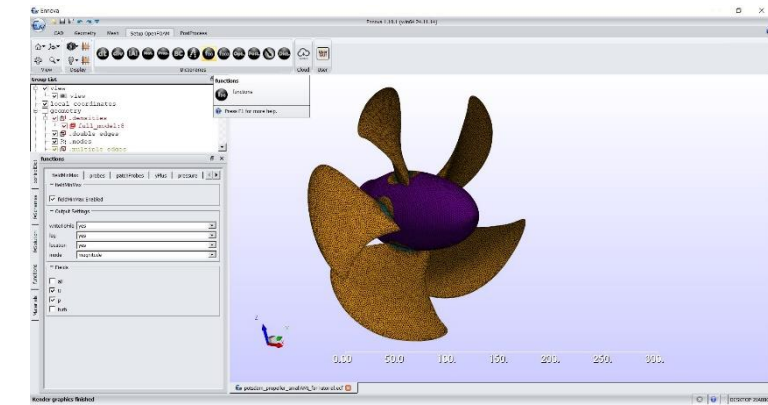
- writeToFile : Yes (To print all min max values in text file)
- Log : Yes (To print values in solver log file)
- writeFields : Yes
 - To write force and moment data on surface patches
 - Check this box if force and moment contours are needed on surface patches

5. CofR : Give co-ordinates of center of roation for moment calculation

6. Field properties

- 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



writeDict

1 Setup

Include Mesh Yes

2

3

4

5

Export Dict Files a

Run OpenFOAM

Platform

Linux

WSL b

Run

Residuals Plot

Residuals File

Plot

Save

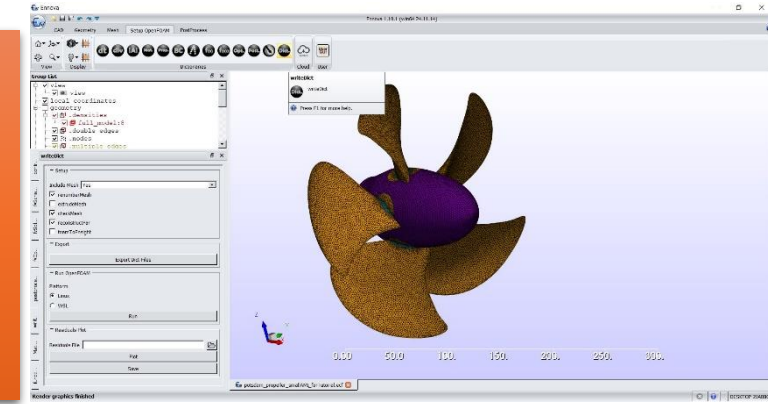
There are two ways of running OpenFOAM case with Ennova,

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

Thank You