Marine Propeller - CFD Simulation

CFD Software Tutorial
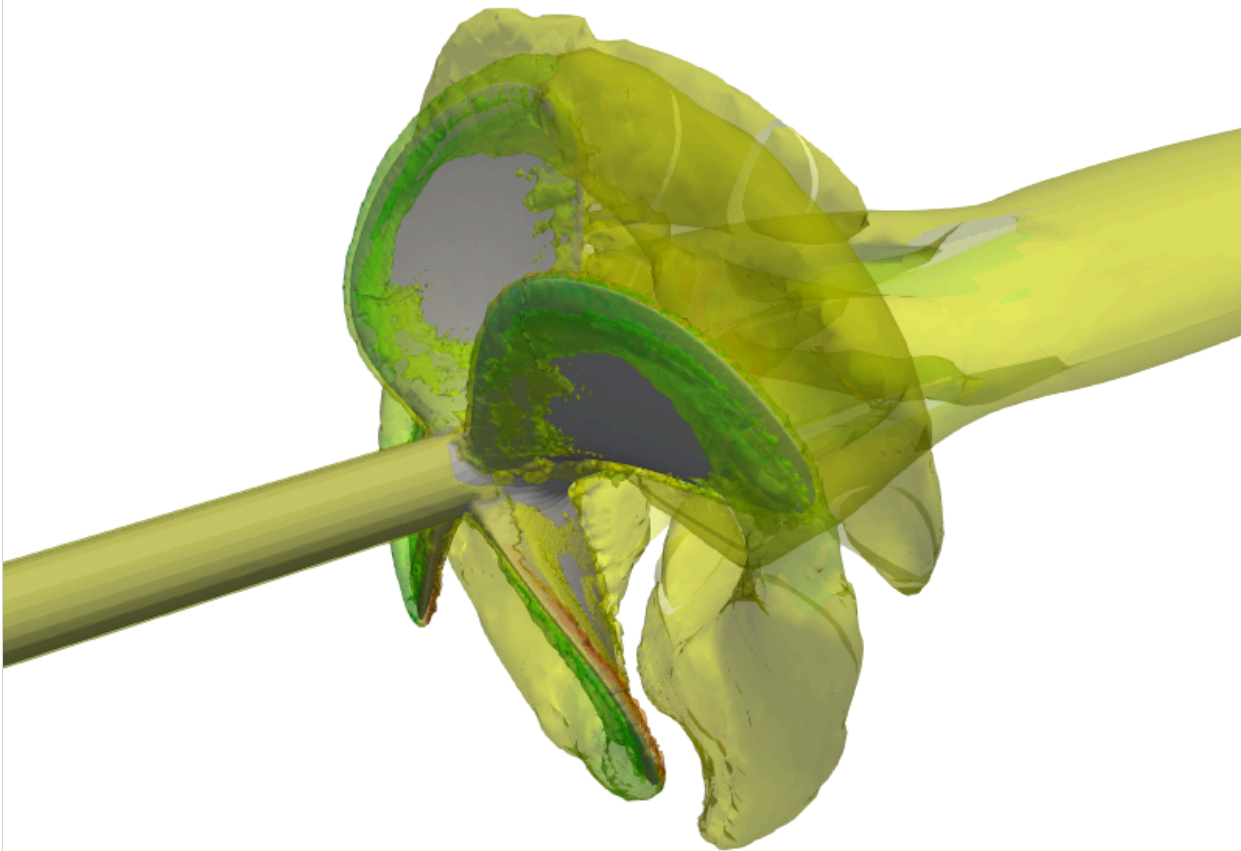
1. Introduction

In this tutorial, we will be modeling a quarter of the propeller due to the double symmetry condition. You will also learn how to create a C̲ ub-volume in the fluid zone) - this is to mark the location of the rotating zone. Using the MRF model to introduce a pseudo

rotation of the propeller - the propeller will be solved in a rotating reference frame and by this, we introduce artificial motion. It is less accurate than resolving motion directly but is more efficient and allows for steady-state simulation of rotating parts. In advanced post-processing using ParaView we show you how to create an iso-surface of a Q criterion.

2. Download SimFlow

SimFlow is a general purpose CFD Software

To follow this tutorial, you will need SimFlow free version, you may download it via the following link:
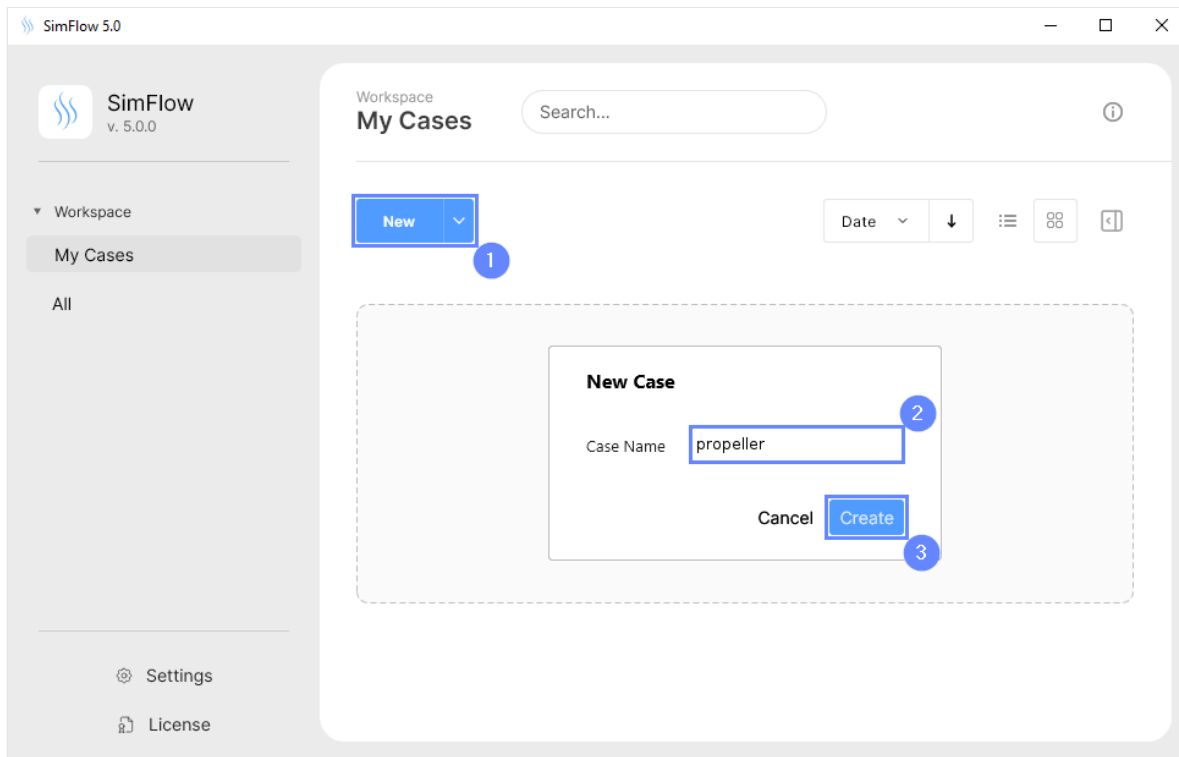
Download SimFlow

3. Create Case

Open SimFlow and create a new case named *propeller*

1   Click   New

2   Provide name   propeller

3   Click   Create   to open a new case
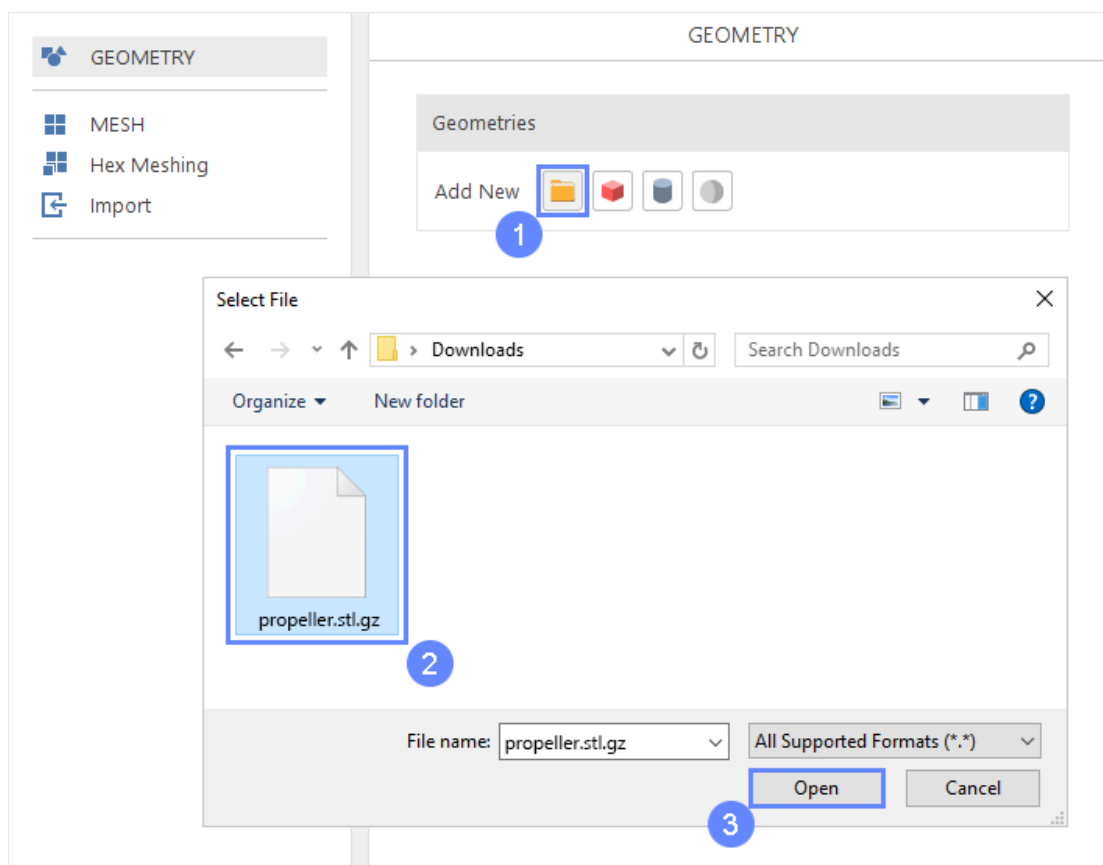
SimFlow 5.0

SimFlow
v. 5.0.0

▾ Workspace

My Cases

All

⚙ Settings

🔒 License

Workspace
My Cases

Search...

New ▾ ①

Date ▾ ↓

New Case

Case Name   propeller ②

Cancel   Create ③

---

4. Import Geometry - Propeller

After creating case Download Geometry | Propeller ▉

1 Click `Import Geometry`

2 Select geometry file `propeller.stl.gz`

3 Click `Open`

GEOMETRY

Geometries

Add New ▢ ▣ ◼ ◯

① ② ③

Select File ✕

← → ⌄ ↑ ▢ > Downloads ⌄ ↻ | Search Downloads 🔍

Organize ▼ New folder ▣ ▼ ▢ ❓

propeller.stl.gz

File name: propeller.stl.gz ⌄ | All Supported Formats (*.*) ⌄

Open Cancel

---

5. Imported Geometry Units

The STL format does not contain the unit information which are defined during the geometry export. If we do not know the exported unit, we can estimate it based on the total size of the model. It is displayed next to *Geometry size* label. In our case, the default unit meter is correct.

1   To confirm default unit `meter`, press OK

Imported Geometry Units

Choose units in which the geometry was created
(This operation is required only for geometry in STL format)

Geometry created in	mm	cm	m	in	ft

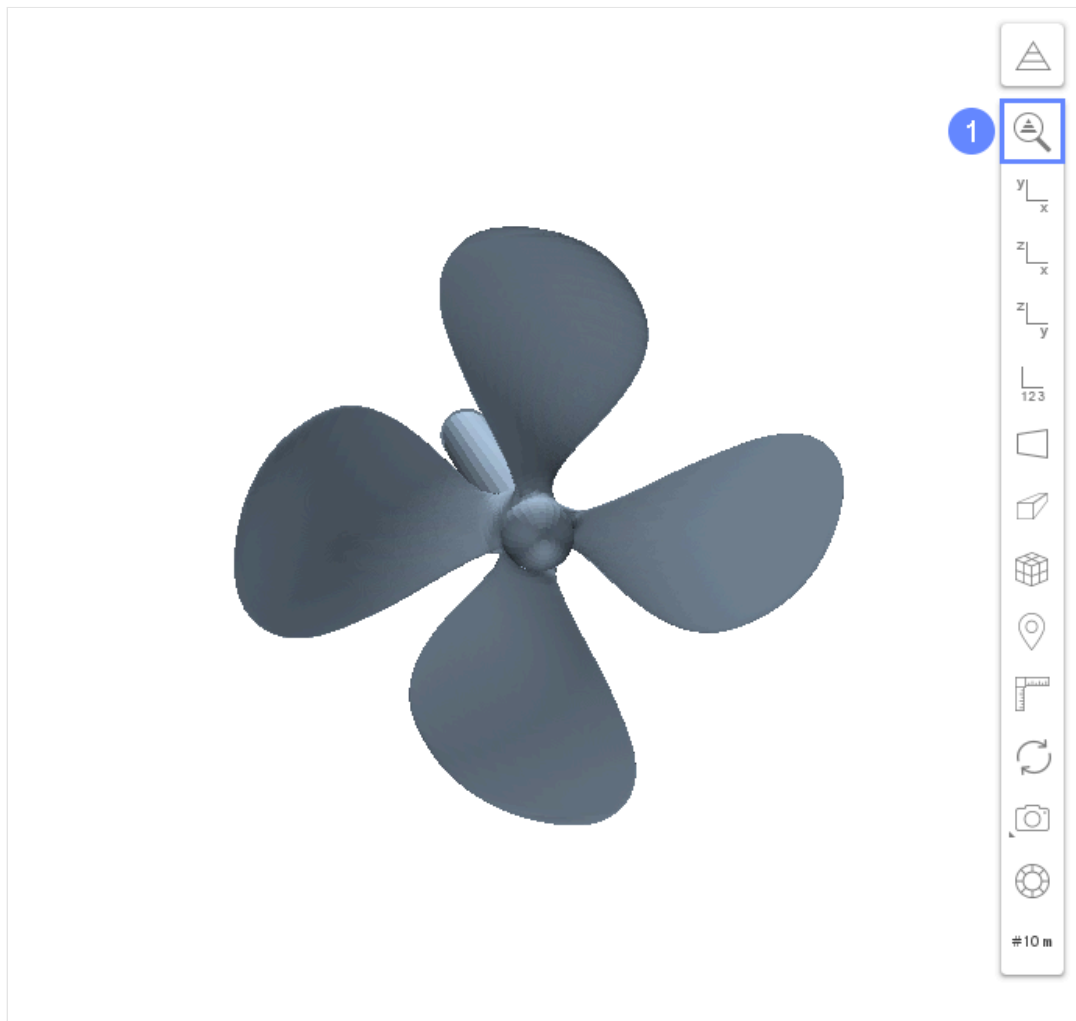| Geometry size	X	0.76 [m]	Y	0.37301 [m]	Z	0.37301 [m]

1

OK

---

6. Propeller Geometry

After importing geometry, it will appear in the 3D window.

1 Click `Fit View` to zoom the geometry

7. Domain Boundary

We need to create a cylindrical boundary for the domain. For this purpose, we will create a cylindrical geometry for later use in the meshing process.
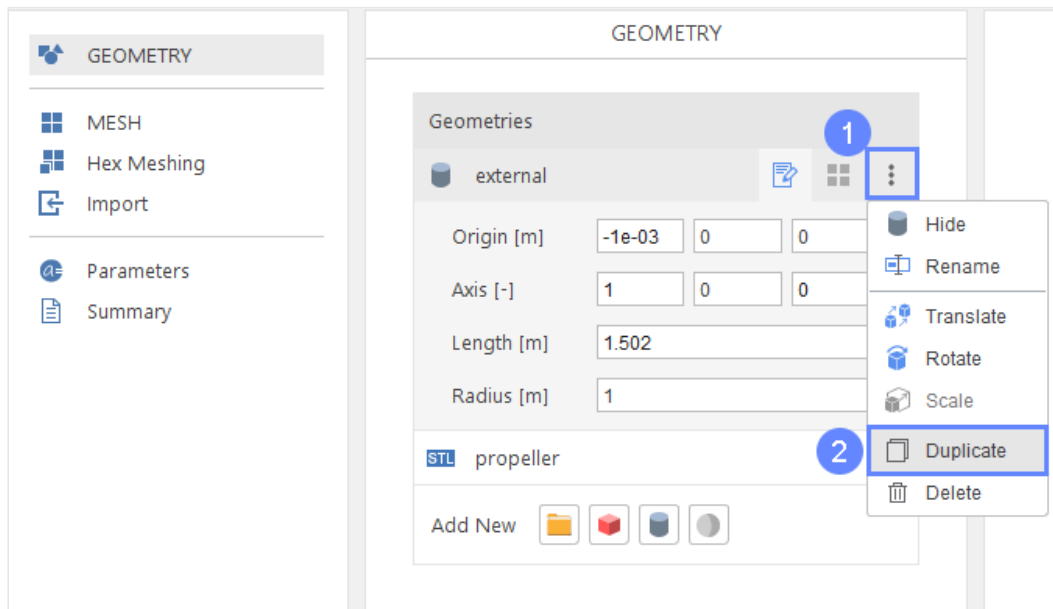
1. Click `Create Cylinder`

2. Rename `cylinder_1` to `external`
   (double click on the name to rename it, press enter to apply)

3. Click `Properties` button if properties panel is not displayed

4   Set origin, axis, and length of the cylinder accordingly

     Origin [m]   -1e-03   0   0

     Axis [−]   1   0   0

     Length [m]   1.502

8. Refinement Area (I)

To better resolve the flow near the propeller, we will create an area with a higher mesh resolution. To do this, we will create a new geometry by copying settings from external geometry.

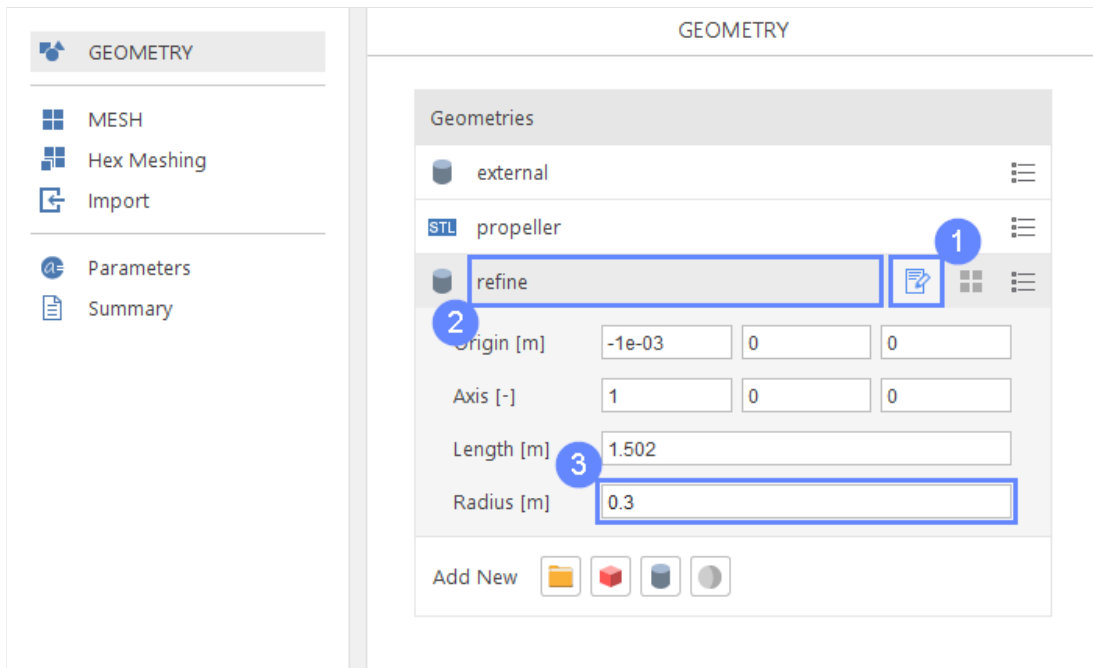1   Click `Options` of the `external` geometry

2   Select `Duplicate` option

9. Refinement Area (II)

1   Click `Properties` if they are not displayed

2   Rename `external_1` to `refine`

3   Set radius of the refinement geometry

`Radius [m]`   `0.3`

GEOMETRY

GEOMETRY

MESH

Hex Meshing

Import

Parameters

Summary

Geometries

external

STL propeller

refine   ① ▦ ☰

② Origin [m]   -1e-03   0   0

Axis [-]   1   0   0

Length [m] ③ 1.502

Radius [m]   0.3

Add New 📁 🟥 🗄 ◐

10. Display Geometries

In order to see all geometries, we will decrease the opacity of the external and refine.

1   Select `external`

2   Click `Display Properties`

3   Adjust `Opacity` to `50%`

4   Adjust opacity to 50% for `refine` geometry by repeating previous steps
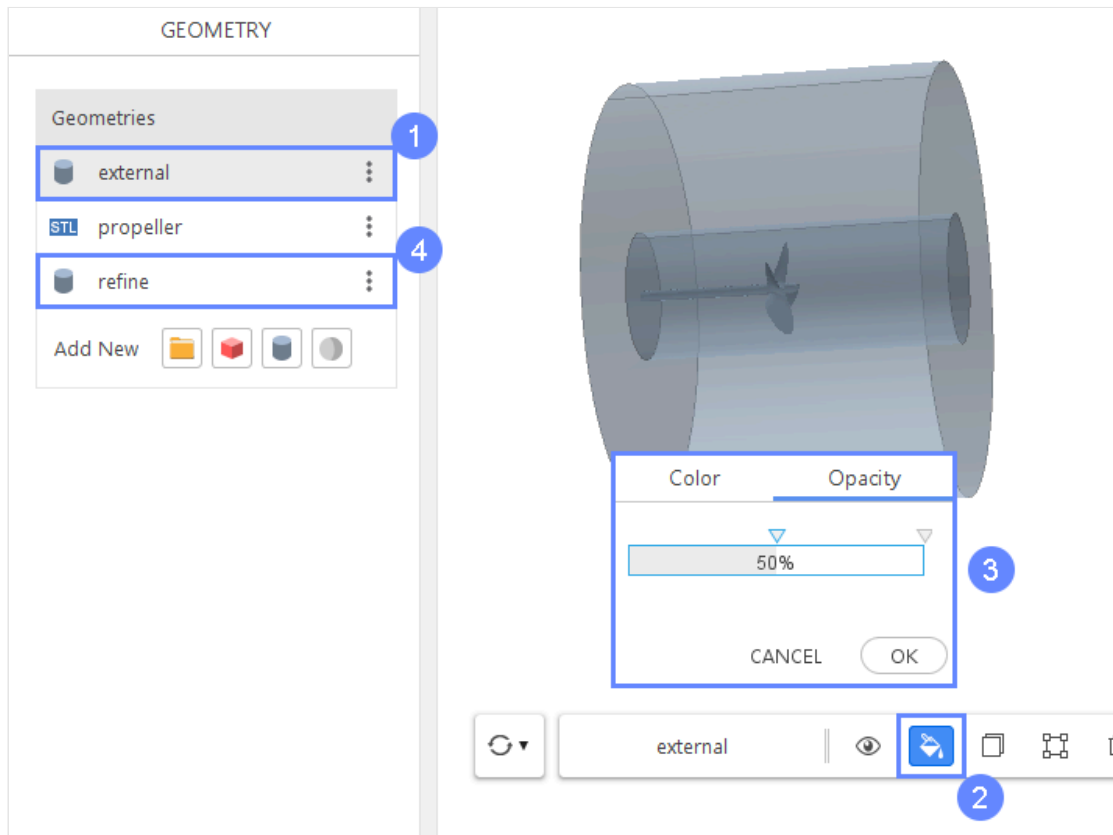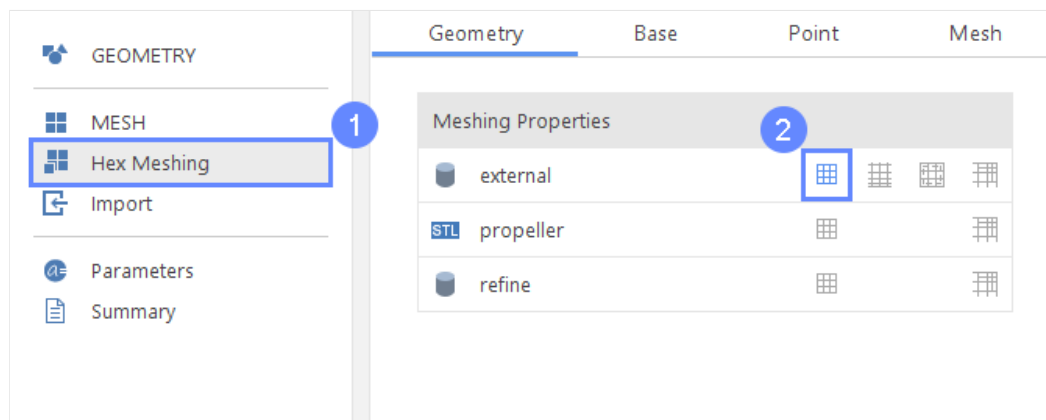
11. Meshing Parameters - External

1   Go to `Hex Meshing` panel

2   Enable `Mesh Geometry` on external geometry
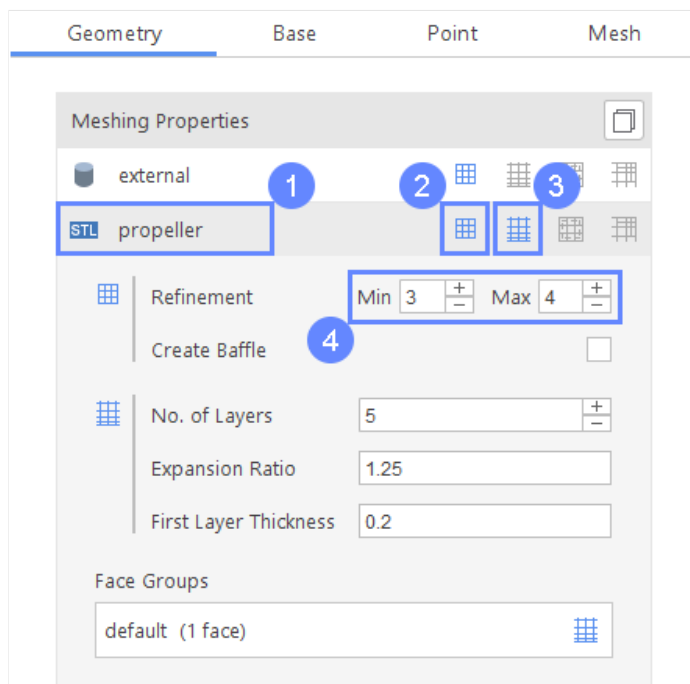
12. Meshing Parameters - Propeller

1   Select `propeller` geometry

2   Enable `Mesh Geometry`

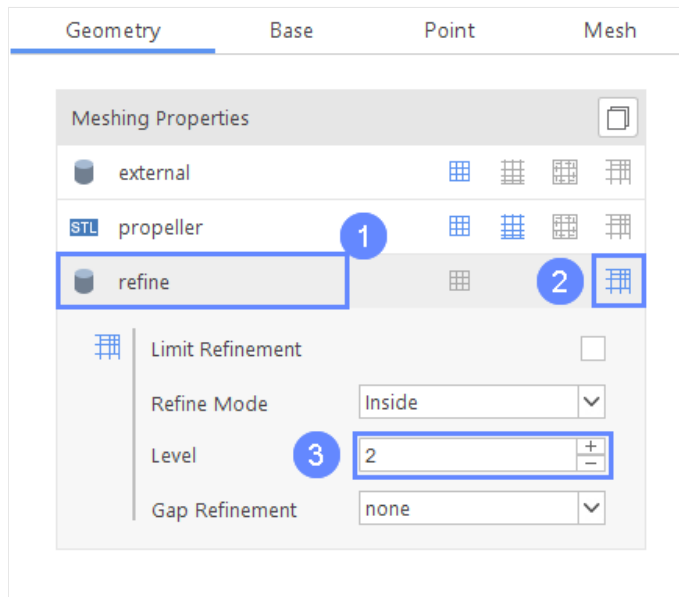3   Enable `Create Boundary Layer Mesh`

4   Set `Refinement` to `Min 3` `Max 4`

13. Meshing Parameters - Refine

We want to create refinement in the area along the propeller induced flow.

1   Select `refine` geometry

2   Enable `Mesh Geometry`

3   Set refinement `Level` to `2`

14. Base Mesh

We want to create a mesh of only one blade of the propeller. For this purpose, we will create a base mesh covering only one-fourth of the geometry.

1. Go to the `Base` tab

2. Define base mesh minimum and maximum extend

   `Min [m]    0    0    0`

   `Max [m]    1.5    1    1`

3. Define Division along each axis

   `Division    30    20    20`

Geometry	Base	Point	Mesh

Base Mesh Type

Box	Cylinder	Plate	Wedge

Geometry

Min [m]	0	0	0
Max [m]	1.5	1	1

Autosize

Mesh

Division	30	20	20
Grading	1	1	1
Cell Size [m]	0.05	0.05	0.05

15. Base Mesh Boundaries

1   2   Define boundary names accordingly

X- inlet

X+ outlet

Y- right

Z- left

Boundaries

X-	inlet ▾		▮ ▾
X+ ①	outlet ▾		▮ ▾
Y-	right ▾		▮ ▾
Y+	boundaries ▾		▮ ▾
Z- ②	left ▾		▮ ▾
Z+	boundaries ▾		▮ ▾

16. Material Point

Now we will define material point outside the propeller geometry.

1　Go to `Point` tab

2　Set location of the material point

　　`Material Point`　`0.5`　`0.5`　`0.5`

Geometry	Base	Point	Mesh
		①	
Multi-Zone Mesh			☐
Material Point	0.5	0.5	0.5 ⊕
②			

17. Start Meshing Process

1　Go to `Mesh` tab

　　: the meshing process with `Mesh` button

Geometry	Base	Point	Mesh

1

CPU (1)	Settings	Notify Me

- ⦿ serial
- ○ parallel

Number of processors [2   ⊞]

Restore Previous Mesh   ⚙ Mesh

2

18. Mesh

After a few minutes of meshing the following mesh should appear.
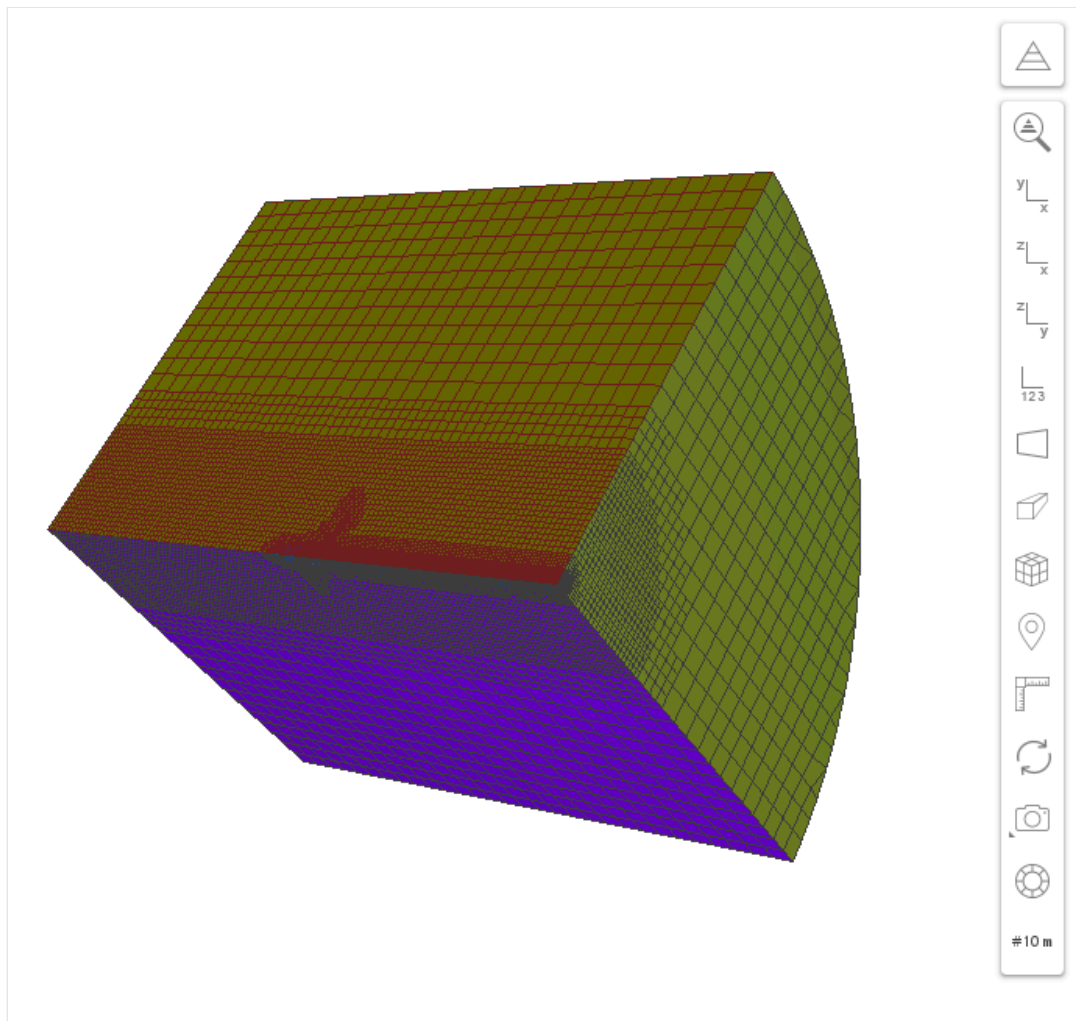
19. Boundary Types

After creating the mesh, we have to set proper boundary types.

1. Go to `Mesh` panel

2. Change the boundary type of the `external` geometry to `patch`

3. Make sure you have appropriate `boundary types` selected
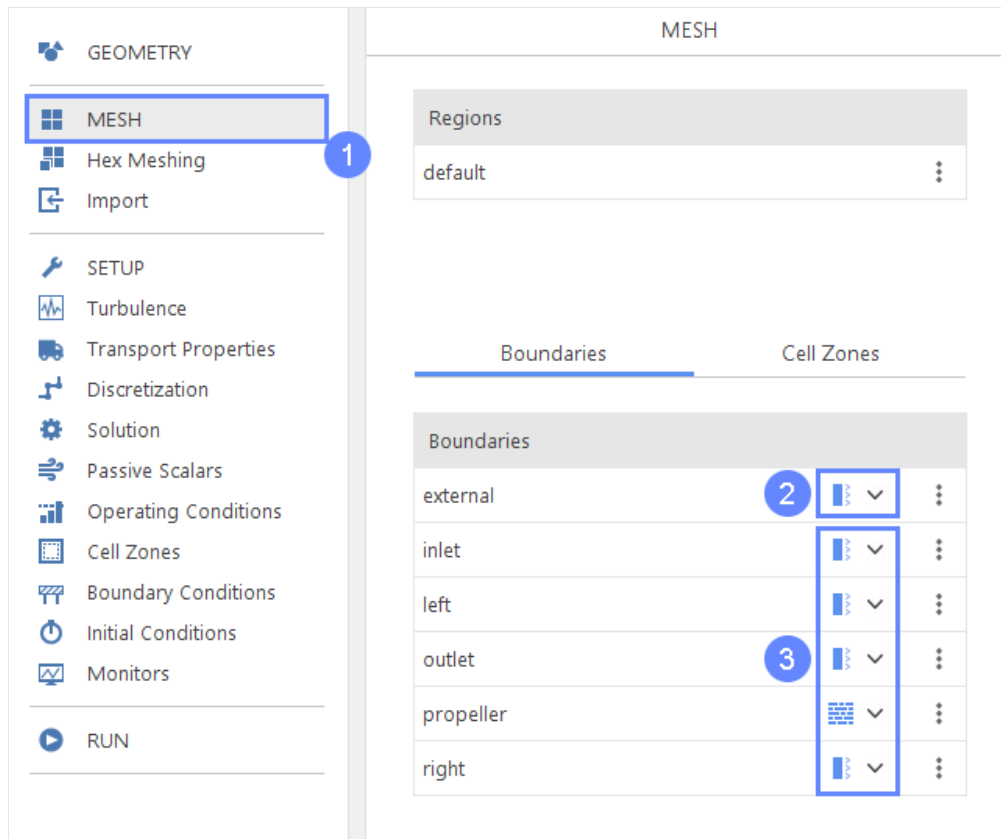
20. Boundary Interface (I)

As we will be simulating only one blade of the propeller we have to create a boundary interface to make the model periodic.

1 Hold `CTRL` key and select `left` and `right` boundary

2 Click `Create Arbitrary Interface` between left and right boundary

MESH

Regions

default ⋮

Boundaries | Cell Zones

Boundaries

external

inlet

left

outlet

propeller

right
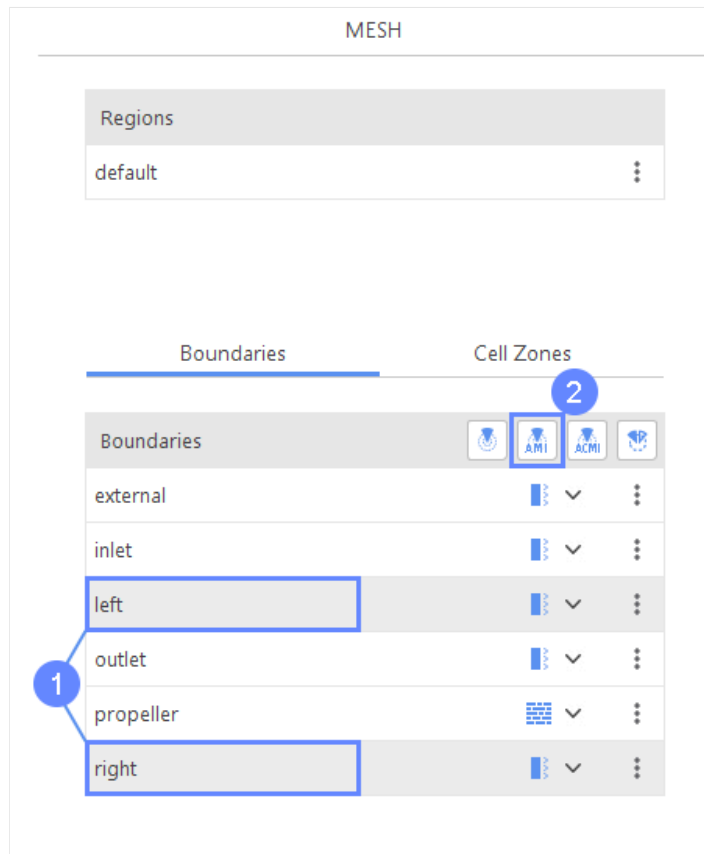
21. Boundary Interface (II)

1 Expand the interface properties

2 Change Transform type to `Rotational`

3 Define Rotation Axis

`Rotation Axis   1   0   0`

22. Select Solver - SIMPLE

For the simulation of a marine propeller, we will use a steady-state incompressible SIMPLE (simpleFoam) solver.

1. Go to `Setup` panel
2. Enable `Steady State` filter
3. Enable `Incompressible` flow filter
4. Pick `SIMPLE` (simpleFoam) solver
5. Click `Select` button to choose the solver

SOLVER SELECTION
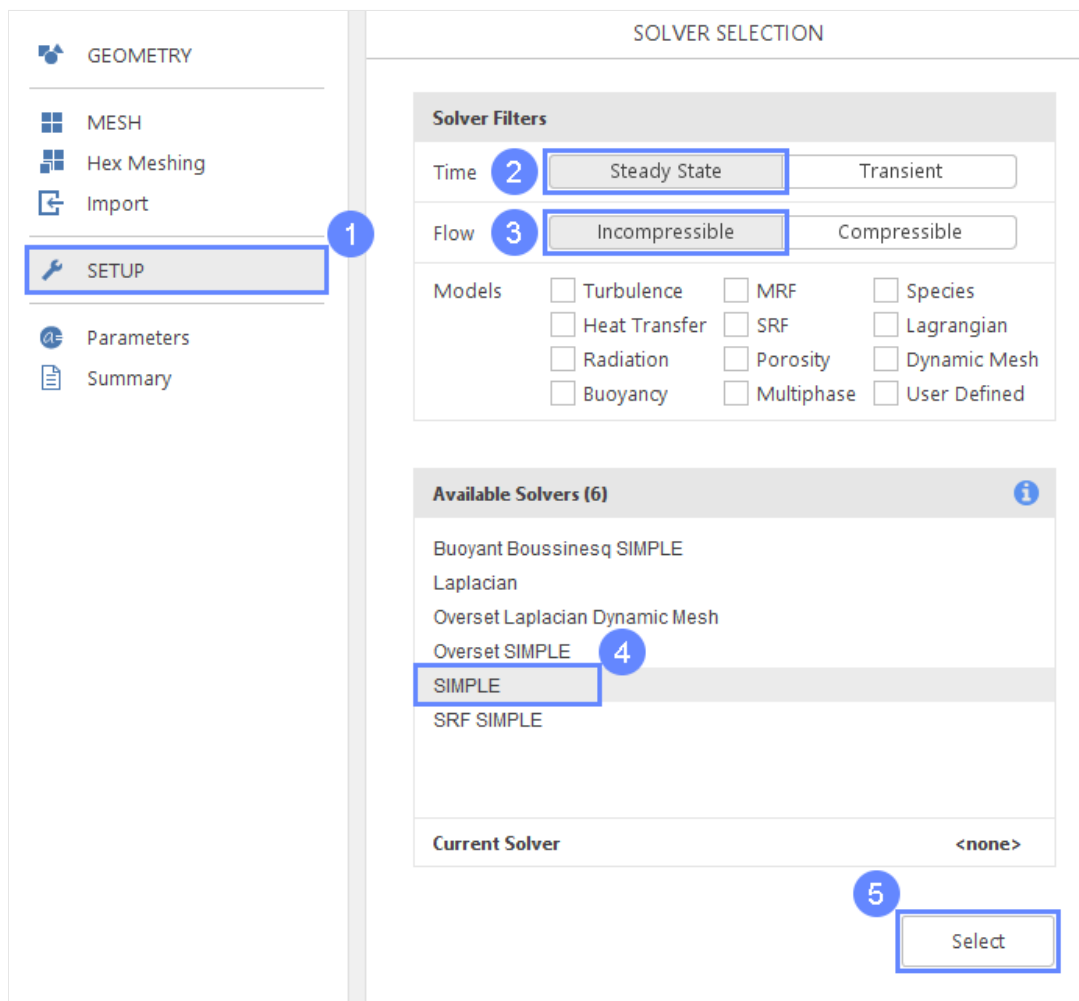
Solver Filters

Time   ②   [Steady State]   [Transient]

Flow   ③   [Incompressible]   [Compressible]

Models
- ☐ Turbulence   ☐ MRF   ☐ Species
- ☐ Heat Transfer   ☐ SRF   ☐ Lagrangian
- ☐ Radiation   ☐ Porosity   ☐ Dynamic Mesh
- ☐ Buoyancy   ☐ Multiphase   ☐ User Defined

Available Solvers (6)　ⓘ

Buoyant Boussinesq SIMPLE
Laplacian
Overset Laplacian Dynamic Mesh
Overset SIMPLE ④
SIMPLE
SRF SIMPLE

Current Solver   <none>

⑤ [Select]

Left panel navigation:
- GEOMETRY
- MESH
- Hex Meshing
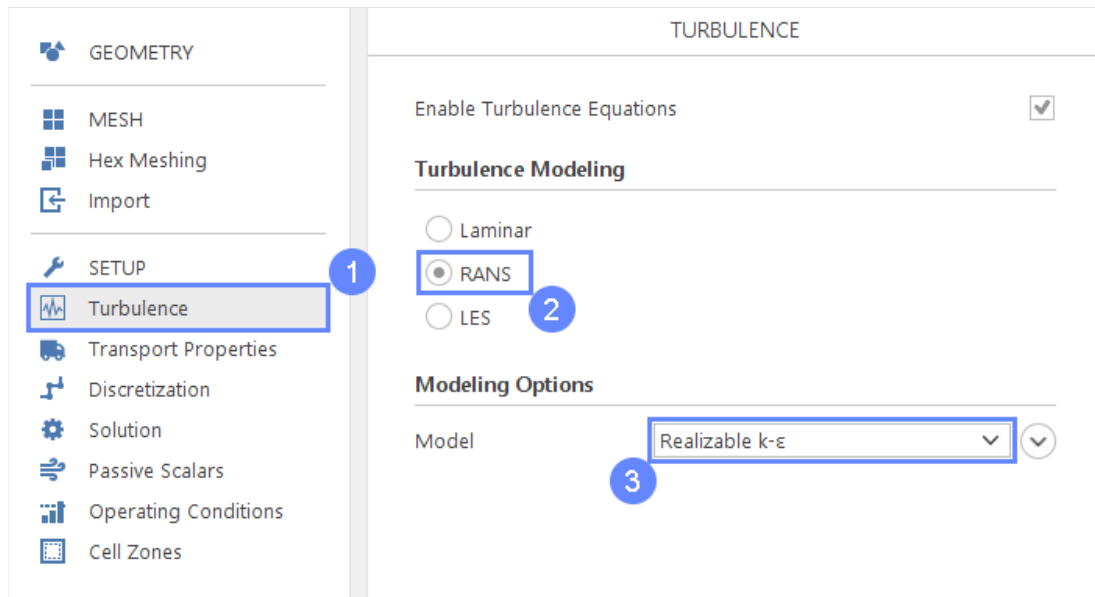- Import
- ① SETUP
- Parameters
- Summary

---

23. Turbulence Model

For the purpose of this tutorial we will simulate the turbulence phenomenon using $Realizable\ k{-}\varepsilon$ model.

1. Go to `Turbulence` panel
2. Select `RANS` modeling
3. Select $Realizable\ k{-}\varepsilon$ turbulence model

TURBULENCE

GEOMETRY

MESH
Hex Meshing
Import

SETUP
1
Turbulence
Transport Properties
Discretization
Solution
Passive Scalars
Operating Conditions
Cell Zones

Enable Turbulence Equations ☑

Turbulence Modeling

○ Laminar
⦿ RANS **2**
○ LES

**Modeling Options**

Model   Realizable k-ε ⌄ ⌄
3

24. Water Properties

1   Go to `Transport Properties` panel

2   Define kinematic viscosity of water

ν [m^2/s]   1e-06

Note that you do not have to define density. The equations that describe single phase incompressible flow operates on a kinematic pressure(pressure divided by reference density). Therefore, the density property does not explicitly appear and you have to remember to multiply resulting pressure and forces by the density value to obtain a physical results.

TRANSPORT PROPERTIES

Transport Model   Newtonian

v [m²/s]   1e-06

     Material Database

①

②

---
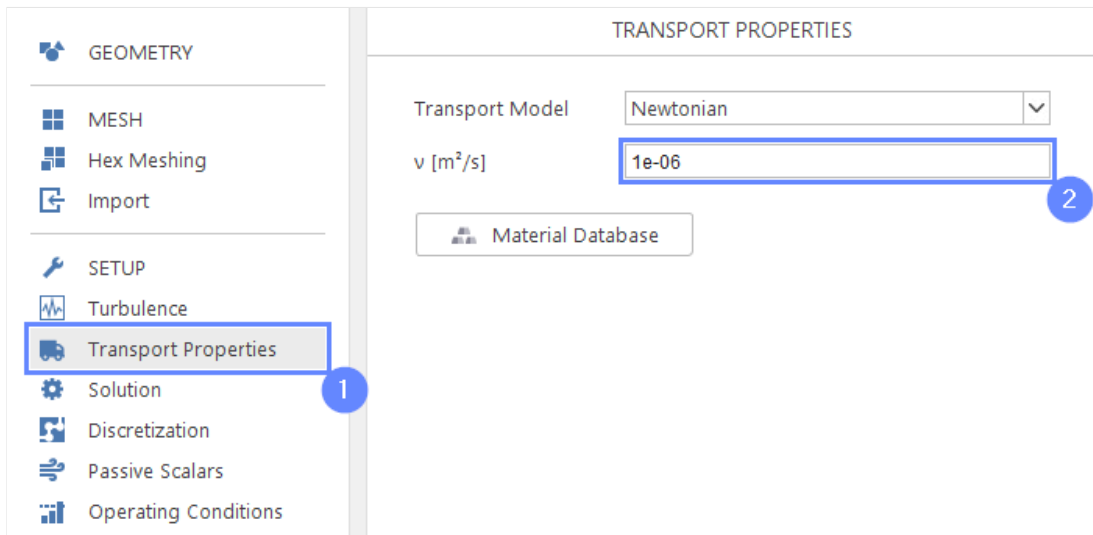
25. Cell Zones for MRF (I)

To be able to model propeller rotation, we will take advantage of a rotating reference frame. This technique will allow modeling the propeller rotation without a need to rotate the mesh. The rotating reference frame can only be applied to a sub-region of the mesh defined by a cell zone object (a list of mesh cells). Therefore, we will first create a cell zone.

1   Go to `Mesh` panel

2   Expand list of options form the `default` region

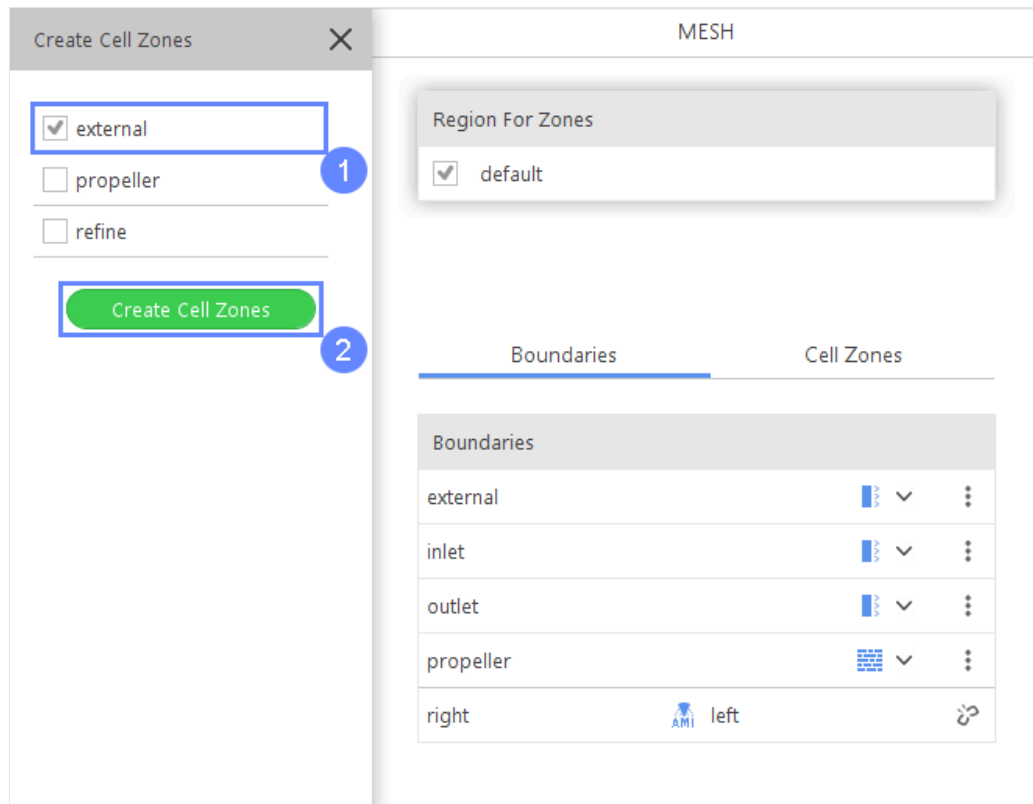3   Select `Add Cell Zones` from the menu
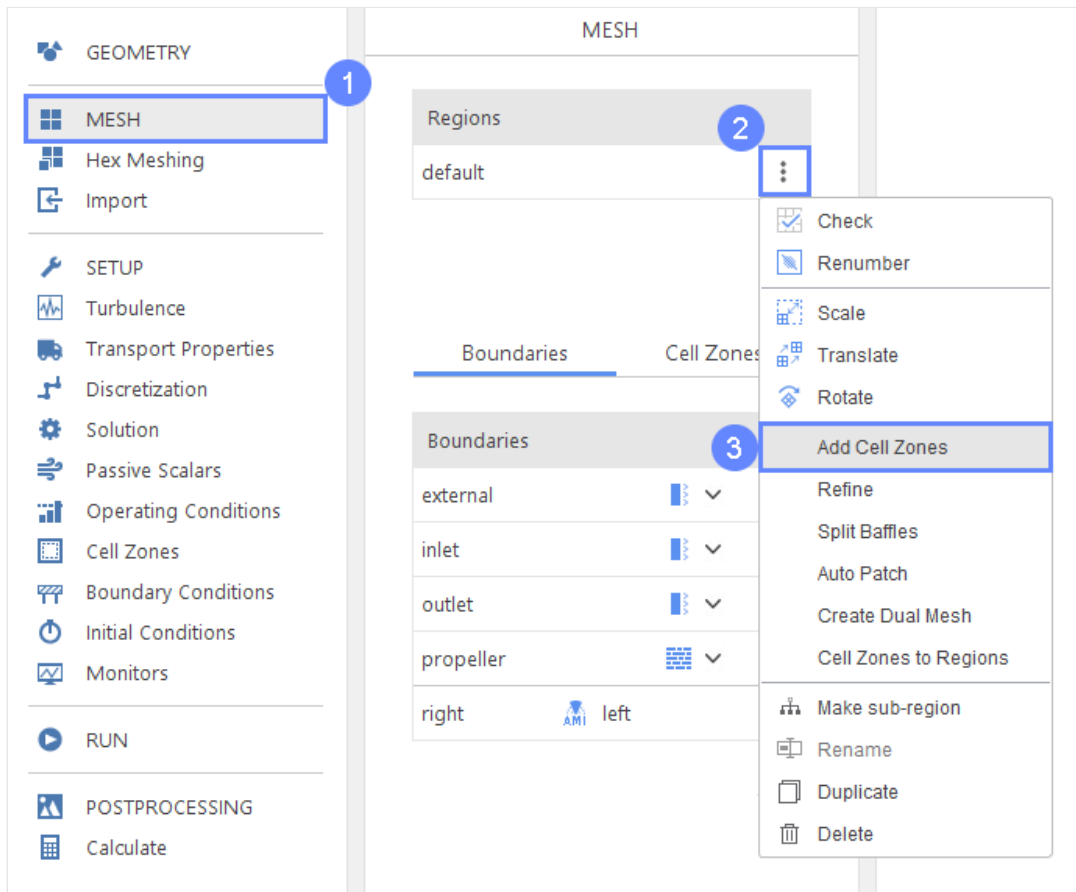
26. Cell Zones for MRF (II)

For the purpose of this tutorial we will use the whole mesh as the rotating zone. To do this we need to create the cell zone inside the external geometry.
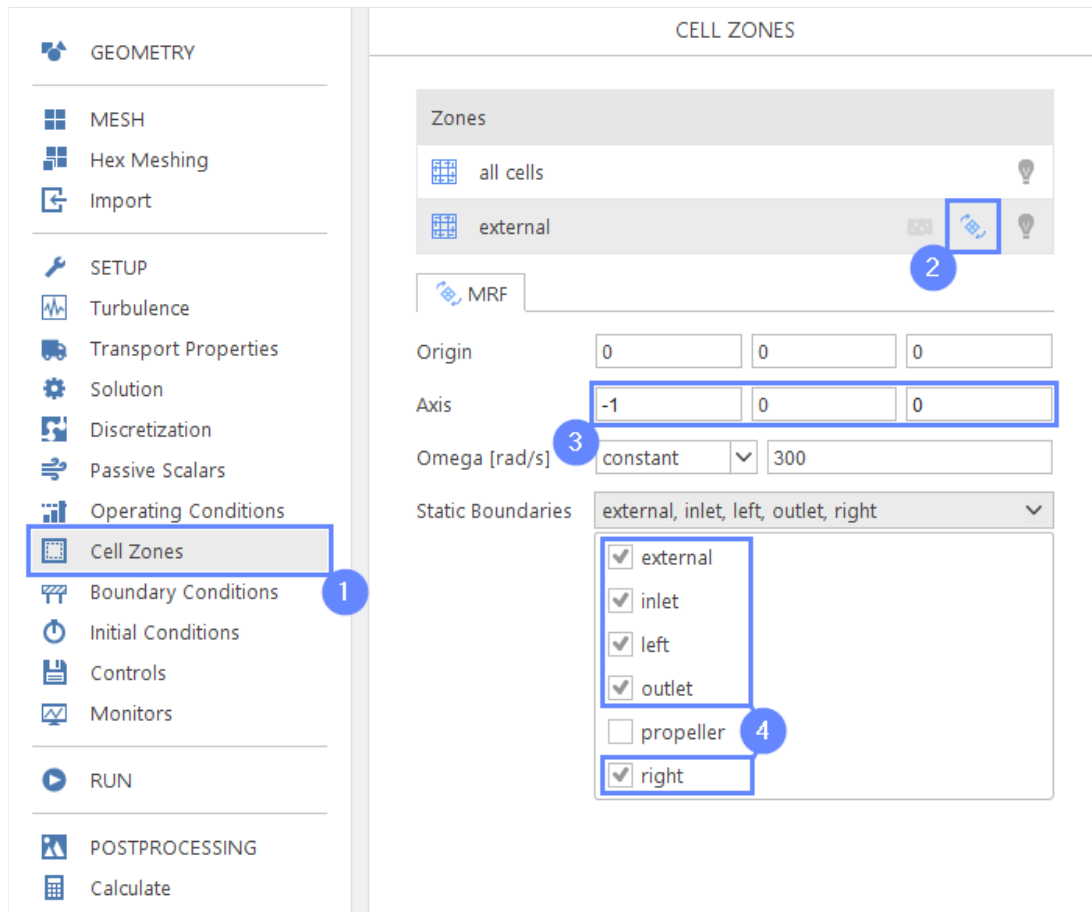
1. Select `external` geometry
2. Click [Create Cell Zones]

27. Rotating Reference Frame

1   Go to `Cell Zones` panel

2   Enable `Rotating Reference Frame` for `external` zone

3   Define Axis of the propeller

      `Axis   -1  0  0`

4   Select boundaries that will not be defined in the rotating frame of reference

      `Static Boundaries` :

      `external`

      `inlet`

      `left`

      `outlet`

      `right`

CELL ZONES

Zones

⊞	all cells		💡
⊞	external	▣ 🔷 ②	💡

🔷 MRF

Origin	0	0	0
Axis	-1	0	0

③

Omega [rad/s] constant ⌄ 300

Static Boundaries external, inlet, left, outlet, right ⌄

☑ external
☑ inlet
☑ left
☑ outlet
☐ propeller ④
☑ right

Navigation menu:
- GEOMETRY
- MESH
- Hex Meshing
- Import
- SETUP
- Turbulence
- Transport Properties
- Solution
- Discretization
- Passive Scalars
- Operating Conditions
- Cell Zones
- Boundary Conditions ①
- Initial Conditions
- Controls
- Monitors
- RUN
- POSTPROCESSING
- Calculate

---

28. Boundary Conditions - External (Turbulence)

1   Go to `Boundary Conditions` panel

2   Select `external` boundary

3   Select `Turbulence` tab

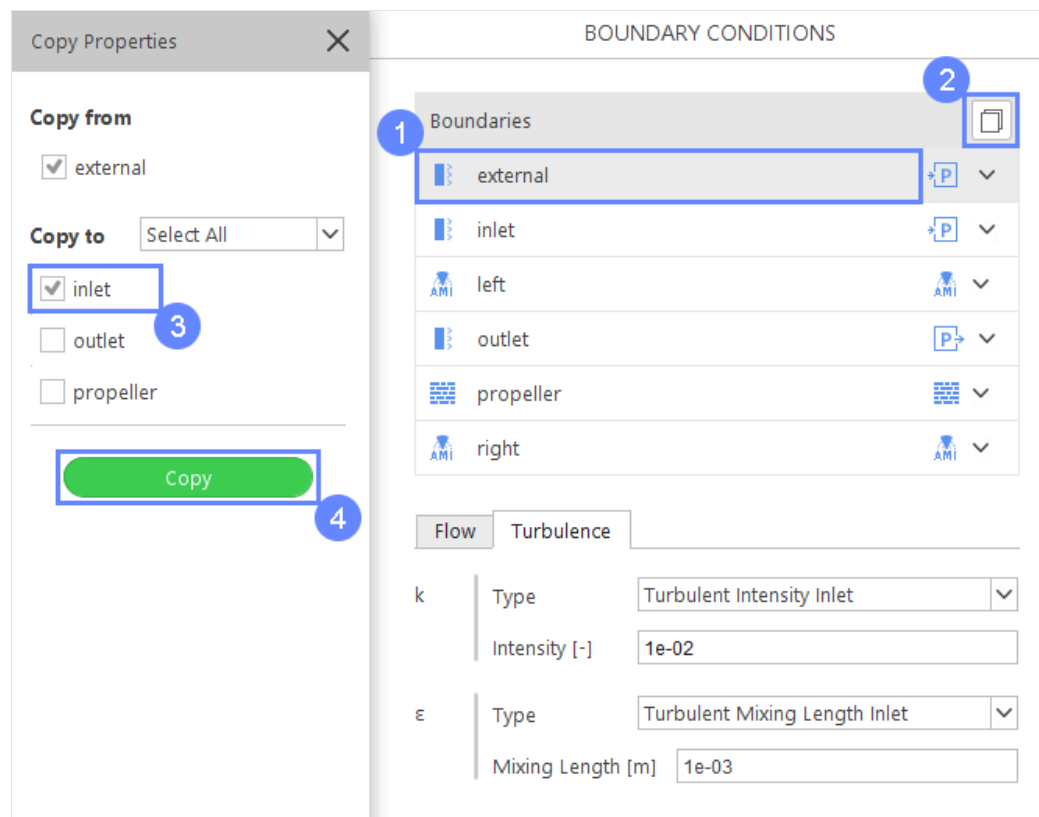4   Set `Turbulence Intensity` to `1e-02`
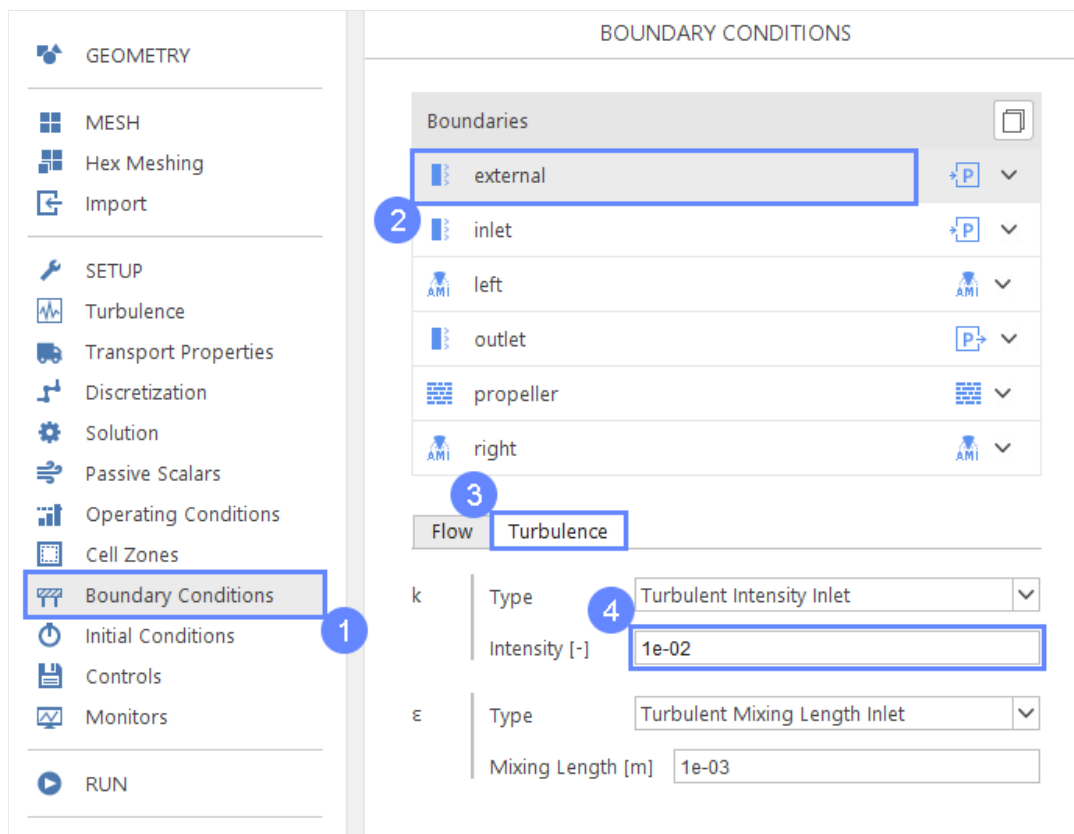
29. Boundary Conditions - Inlet

We will use the same boundary conditions for inlet and external boundaries, to achieve this we can copy settings from the external boundary to inlet.

1. Make sure you have selected the `external` boundary

2. Click `Copy Boundary Conditions`
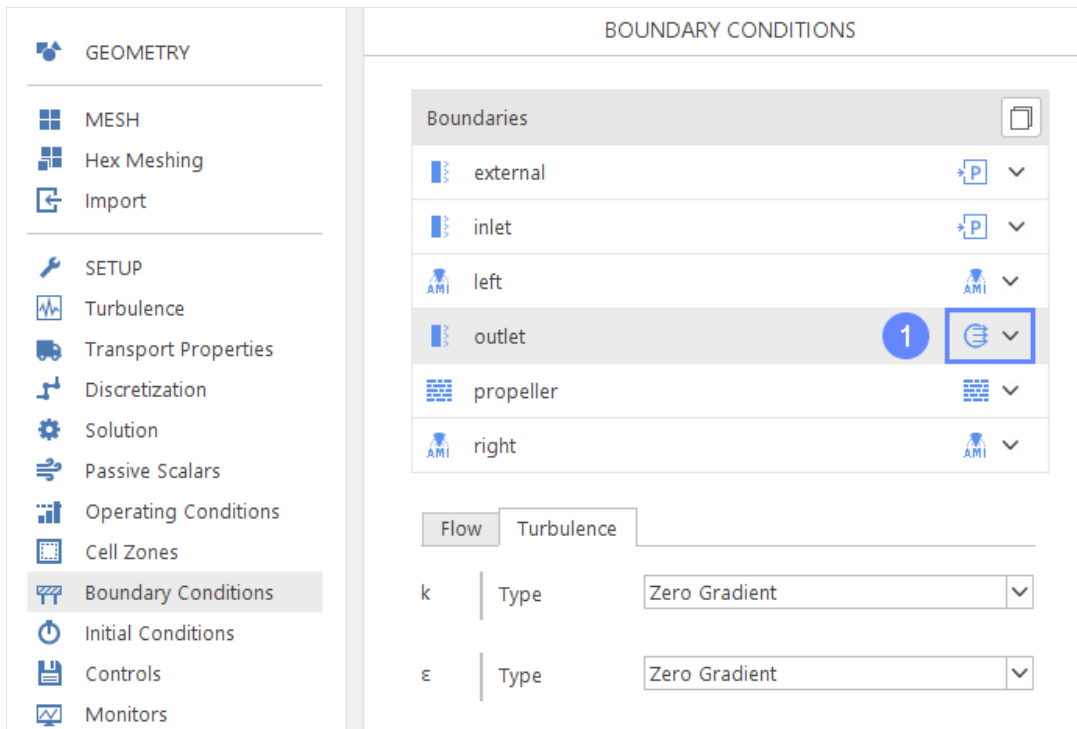
3. Select `inlet` boundary

4. Click `Copy`

30. Boundary Conditions - Outlet

1   Change `outlet` character to `Outflow`

BOUNDARY CONDITIONS

Boundaries		
external		P ⌄
inlet		P ⌄
left		AMI ⌄
outlet	**1**	⌄
propeller		⌄
right		AMI ⌄

Flow	Turbulence

k	Type	Zero Gradient ⌄
ε	Type	Zero Gradient ⌄

GEOMETRY

- MESH
- Hex Meshing
- Import

- SETUP
- Turbulence
- Transport Properties
- Discretization
- Solution
- Passive Scalars
- Operating Conditions
- Cell Zones
- Boundary Conditions
- Initial Conditions
- Controls
- Monitors

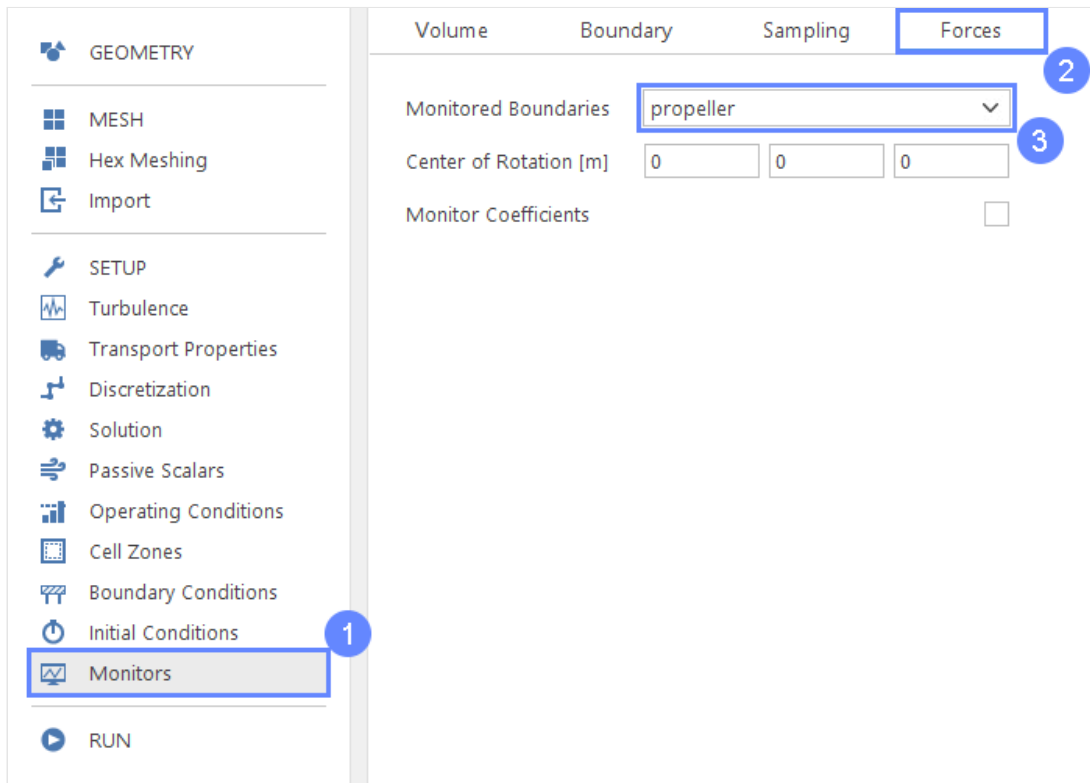# 31. Monitor Forces

We want to monitor solution progress by observing force coefficients.

1. Go to `Monitors` panel
2. Select `Forces` tab
3. Expand `Monitored Boundaries` list
4. Select `propeller` boundary

GEOMETRY

MESH
Hex Meshing
Import

SETUP
Turbulence
Transport Properties
Discretization
Solution
Passive Scalars
Operating Conditions
Cell Zones
Boundary Conditions
Initial Conditions
Monitors   ①

RUN

Volume   Boundary   Sampling   **Forces** ②

Monitored Boundaries   propeller ▾ ③

Center of Rotation [m]   0   0   0

Monitor Coefficients   ☐

---

32. Run - Time Control

1   Go to `Run` panel

2   Set `Number of Iterations` to `1000`
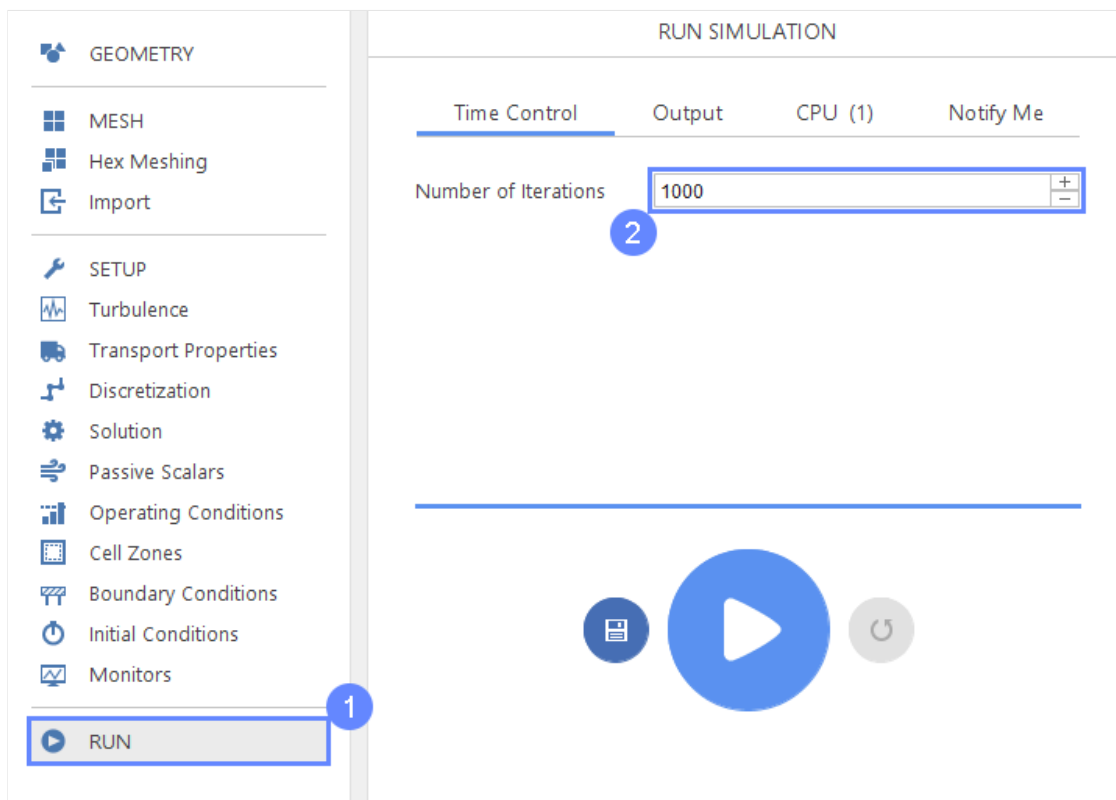
33. Run - CPU

To speed up the calculation process, take advantage of parallel computing and increase the number of CPUs based on your PC's capability. The free version allows you to use only one processor (serial mode). To get the full version, you can use the contact form to

Request 30-day Trial

Estimated computation time for serial mode: 15 minutes
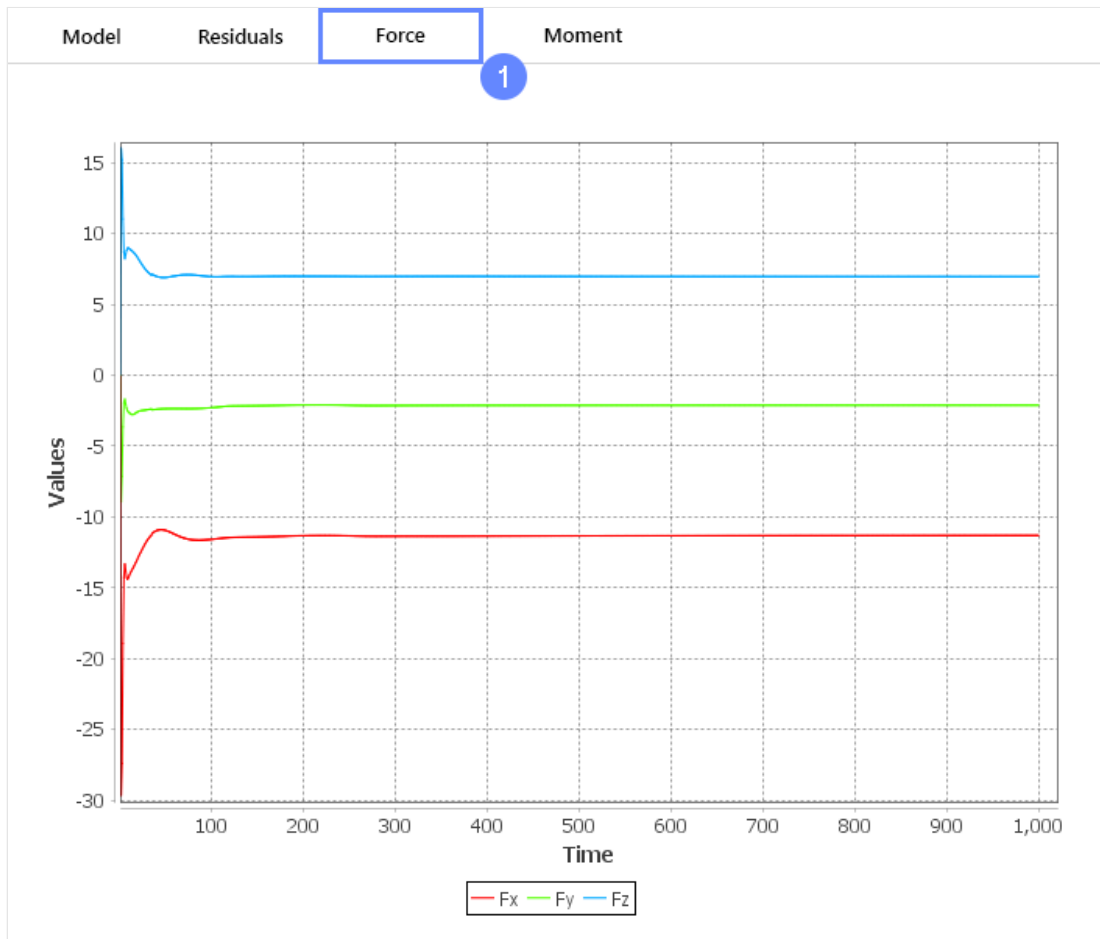
1 Switch to `CPU` tab

2 Click `Run Simulation` button

RUN SIMULATION

Time Control	Output	CPU (1)	Notify Me

①

◉ serial

◯ parallel

Number of processors   6   [+]
                             [−]

②

---
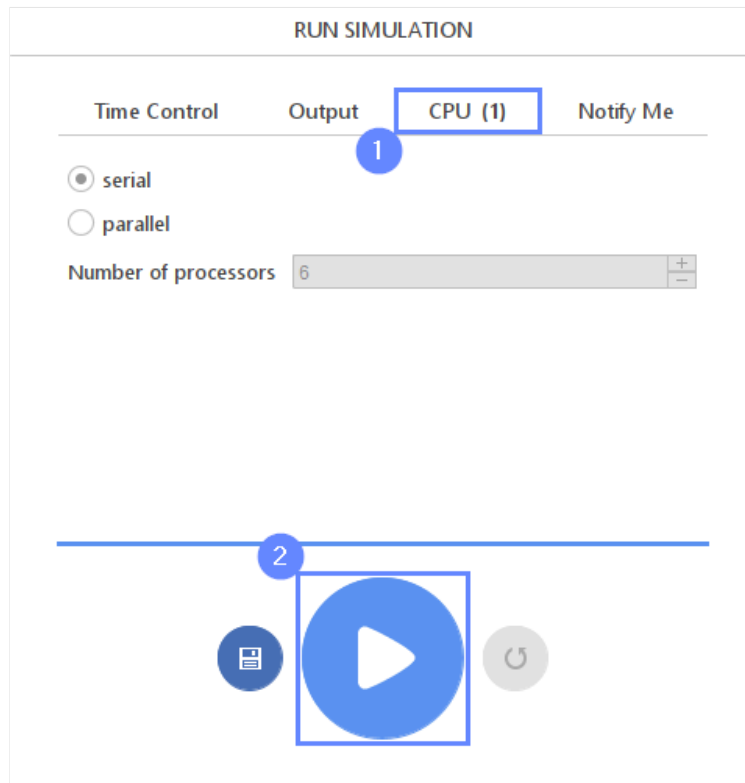
34. Monitor Solution - Force

Check if solution converges by observing stabilization of forces on the propeller boundary.
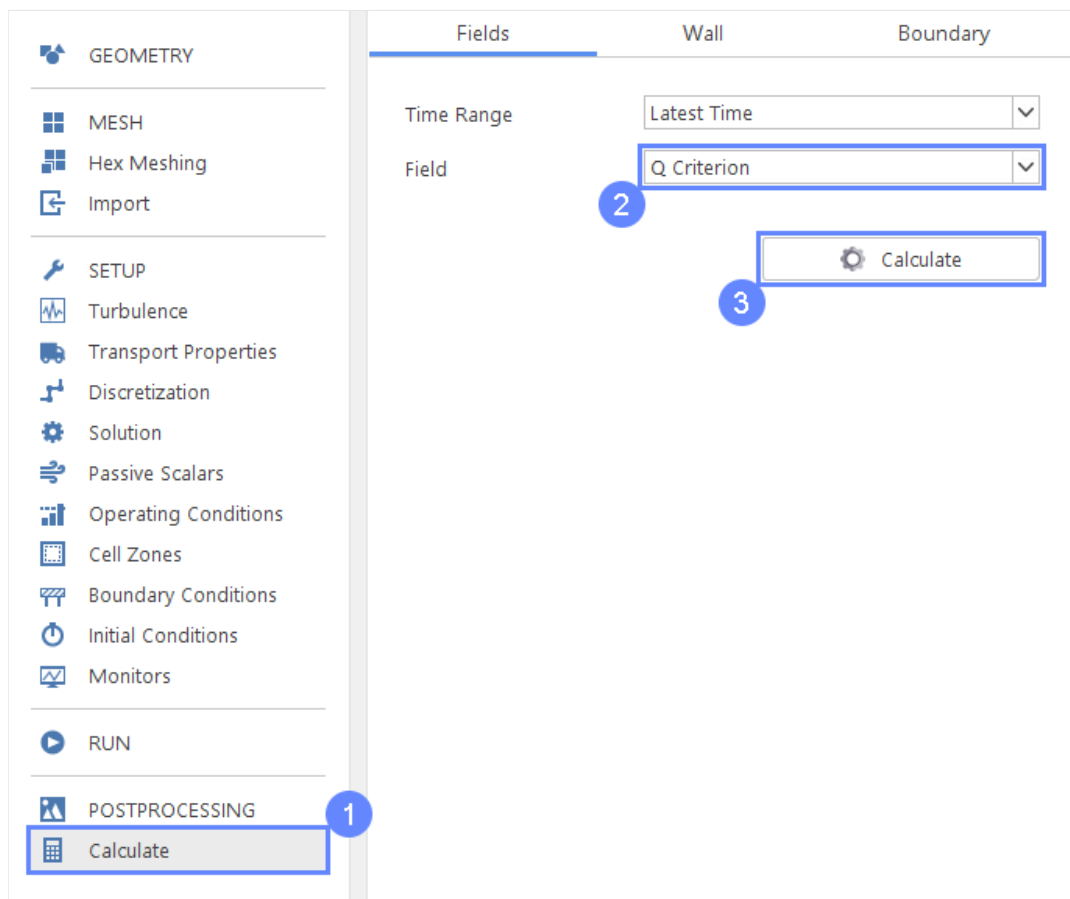
1   Click on `Forces` tab to display Force Monitor

Model	Residuals	Force	Moment

1

35. Calculate Additional Fields

When a simulation is finished, we want to calculate additional solution field to be used in postprocessing.

1   Go to `Calculate` panel

2   Select `Q Criterion` field

3   `Calculate` the field

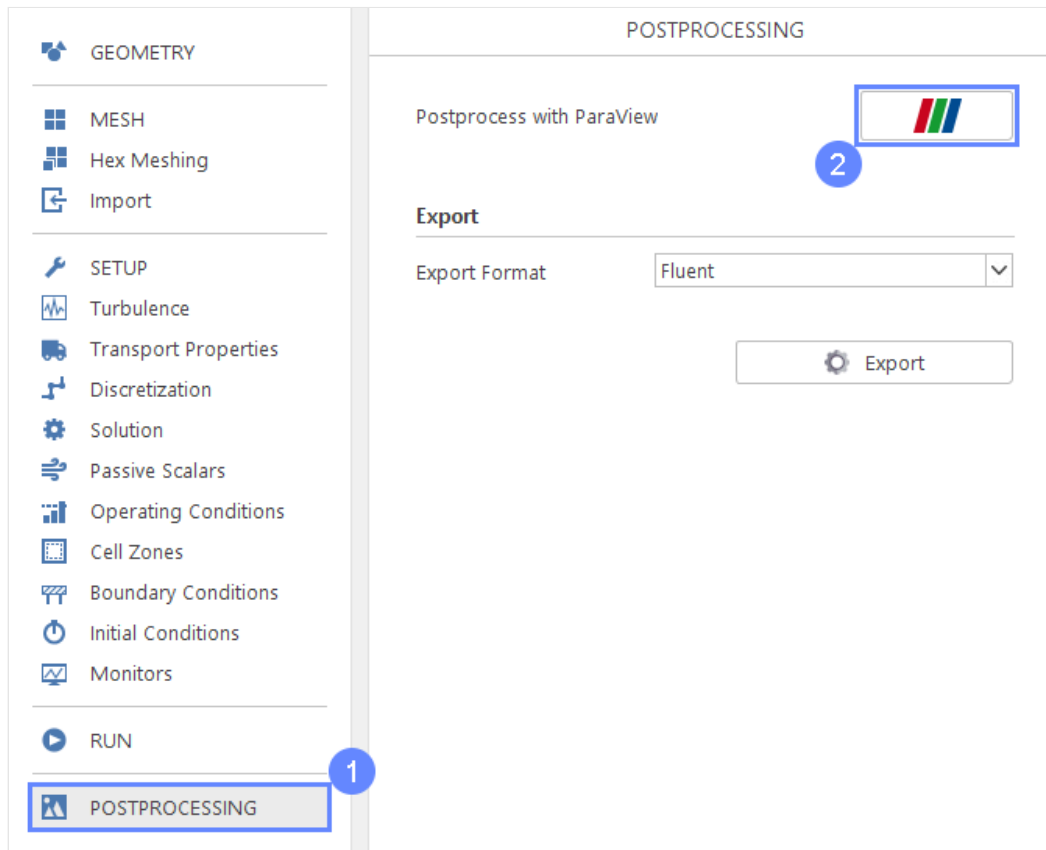Please Note: After clicking the Calculate button, SimFlow will calculate the new flow variable, which will be stored in your project folder and will be accessible in the ParaView.

Fields	Wall	Boundary

Time Range   Latest Time ▾

Field   Q Criterion ▾

(2)

⚙ Calculate

(3)

GEOMETRY

MESH
Hex Meshing
Import

SETUP
Turbulence
Transport Properties
Discretization
Solution
Passive Scalars
Operating Conditions
Cell Zones
Boundary Conditions
Initial Conditions
Monitors

RUN

POSTPROCESSING   **(1)**
Calculate

---

36. Start Postprocessing with ParaView

1   Go to `Postprocessing` panel

2   Run `ParaView`

GEOMETRY

MESH
Hex Meshing
Import

SETUP
Turbulence
Transport Properties
Discretization
Solution
Passive Scalars
Operating Conditions
Cell Zones
Boundary Conditions
Initial Conditions
Monitors

RUN

POSTPROCESSING

Postprocess with ParaView

Export

Export Format   Fluent

Export

---

37. ParaView - Load Results

Now we are loading results into the ParaView

1   Click `Last Frame` to select the last result set

2   Click `Apply` to load results into ParaView

3   After loading results they will be shown in the 3D graphic window
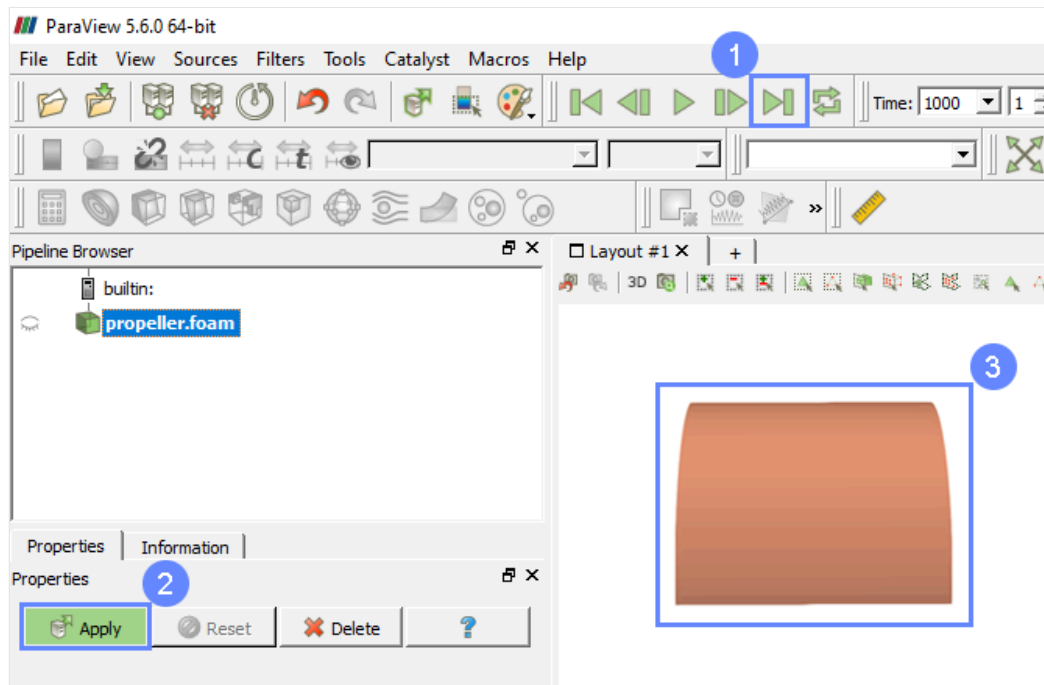
38. ParaView - Display Q Contour (I)

To show the propeller we will create now a contour.

1. Make sure you have your case selected

2. Click `Contour`
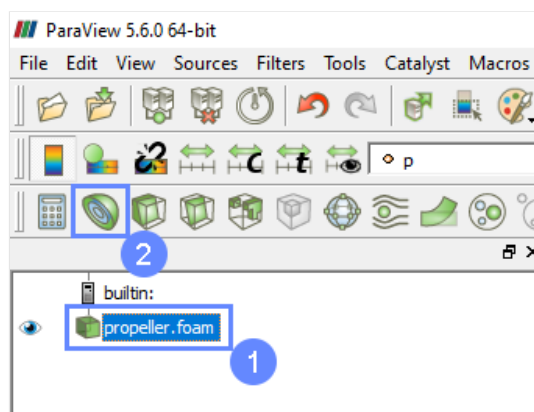
39. ParaView - Display Q Contour (II)

1. Set `Contour by` to `Q`

2. Set contour range to `500` and click enter
   (double click to edit visible value)

3. Click `Apply` to create new contour

4. After applying changes the contour will be shown in the 3D window
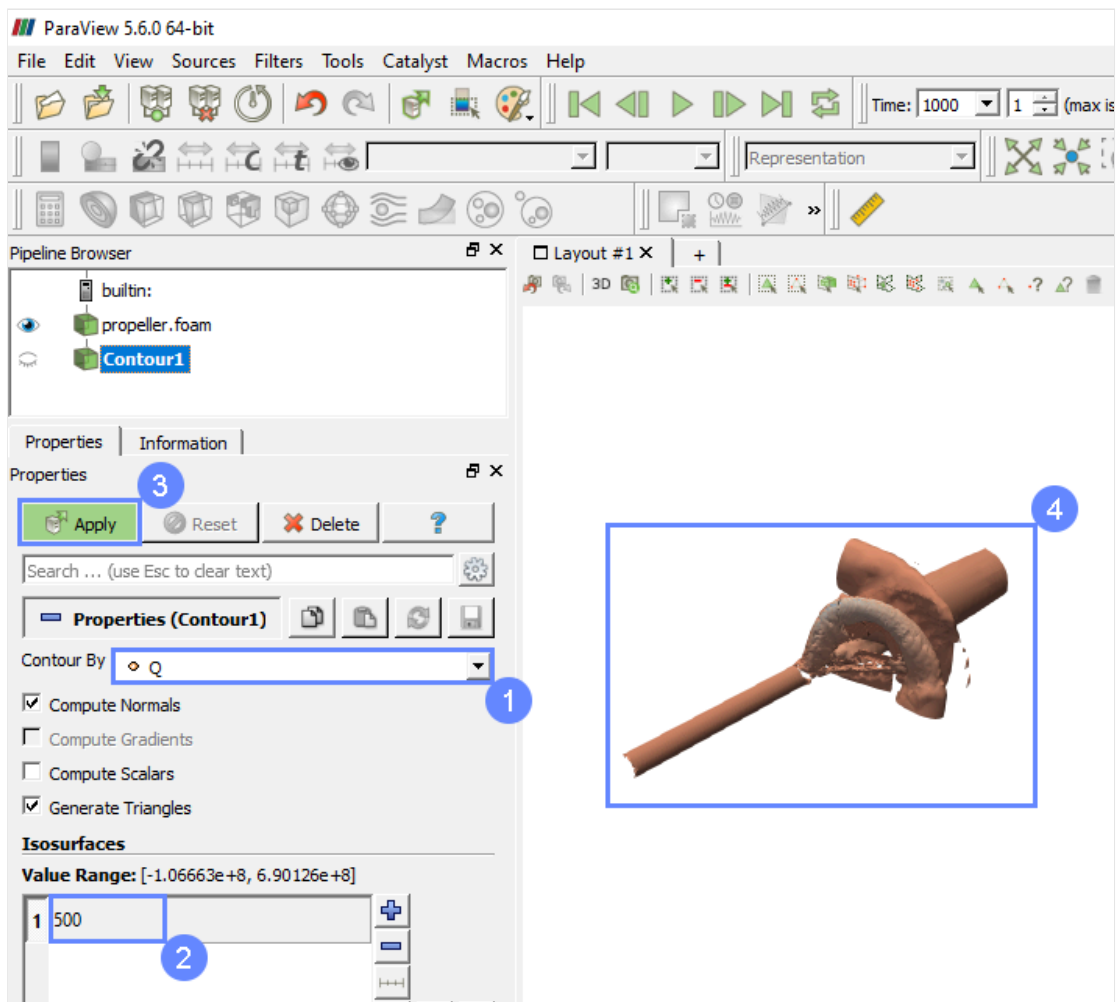
40. ParaView - Show Full Propeller (I)

As you can see there is only a quarter of the propeller visible and we would like to show the full geometry. In order to do that we have to duplicate and transform a partial propeller.

1.   Make sure you have `Contour1` selected

2.   From the `Sources` Menu select `Search` option to open a search window

3.   Start typing the word `transform`

4.   Select `Transform` option

5.   Make sure you have `Transform1` selected

6.   Define `90` degree rotation about the X axis and click enter

7.   Click `Apply` 2 times to create transformation

We have to repeat the previous step to make another transformation

- Transorm1 → Transform2

1. Make sure you have `Transform1` selected
2. From the `Sources` Menu select `Search` option to open a search window
3. Start typing the word `transform`
4. Select `Transform` option
5. Make sure you have `Transform2` selected
6. Define `90` degree rotation about the X axis and click enter
7. Click `Apply` 2 times to create transformation

We have to repeat the previous step to make another transformation
- Transorm2 → Transform3

1   Make sure you have `Transform2` selected

2   From the `Sources` Menu select `Search` option to open a search window

3   Start typing the word `transform`

4   Select `Transform` option

5   Make sure you have `Transform3` selected

6   Define `90` degree rotation about the X axis and click enter

7   Click `Apply` 2 times to create transformation
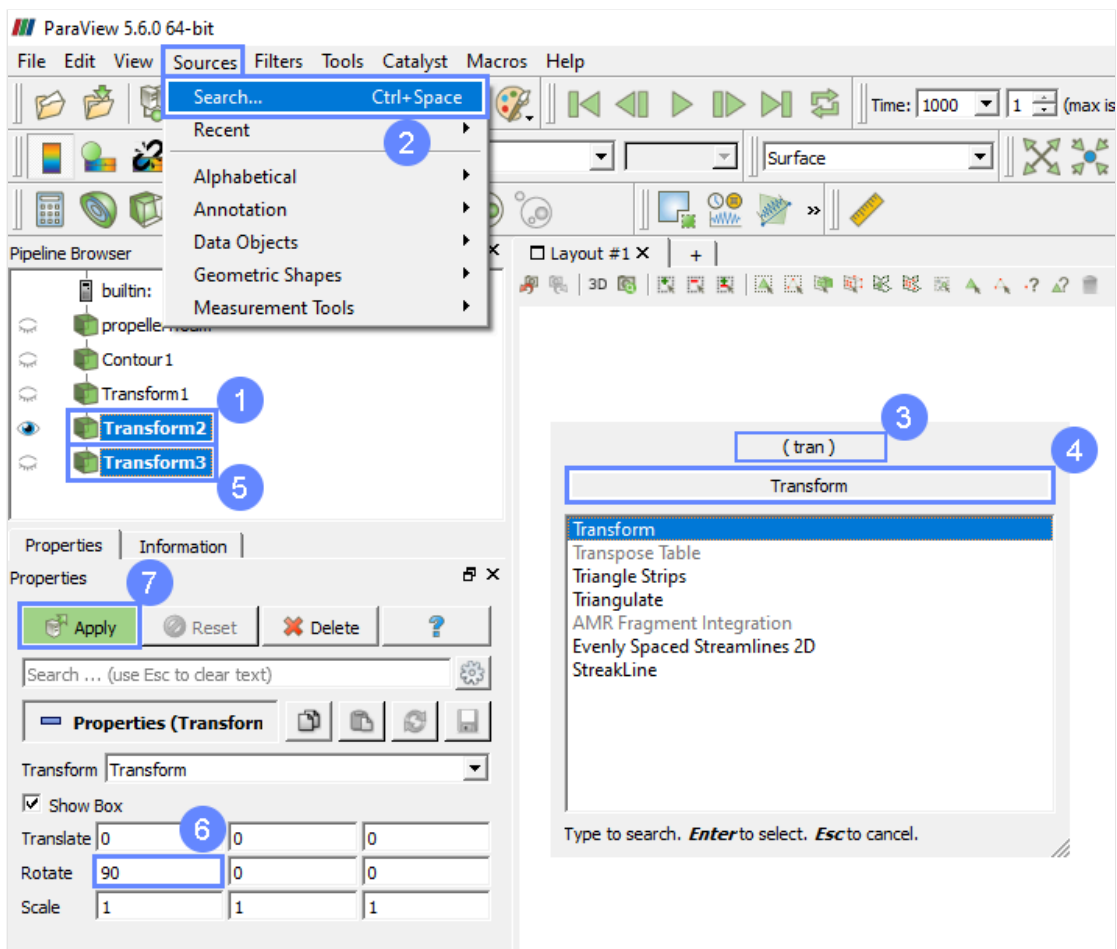
43. Display Full Propeller

To make the full geometry visible, we have to change the visibility of all geometries

1. Make sure the contour and all its transformations are visible

2. Select `Contour1`

3. Choose pressure `p` from the coloring list

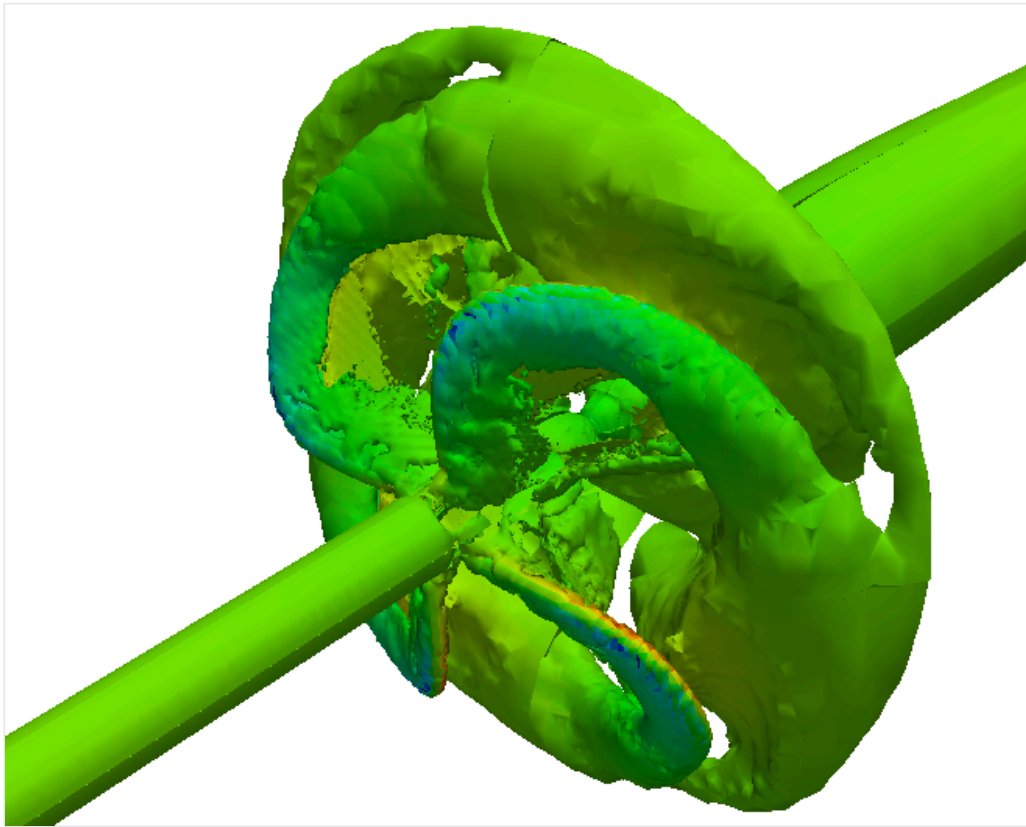4. Click `Toggle Color Legend Visibility` and choose `Blue to Red Rainbow`
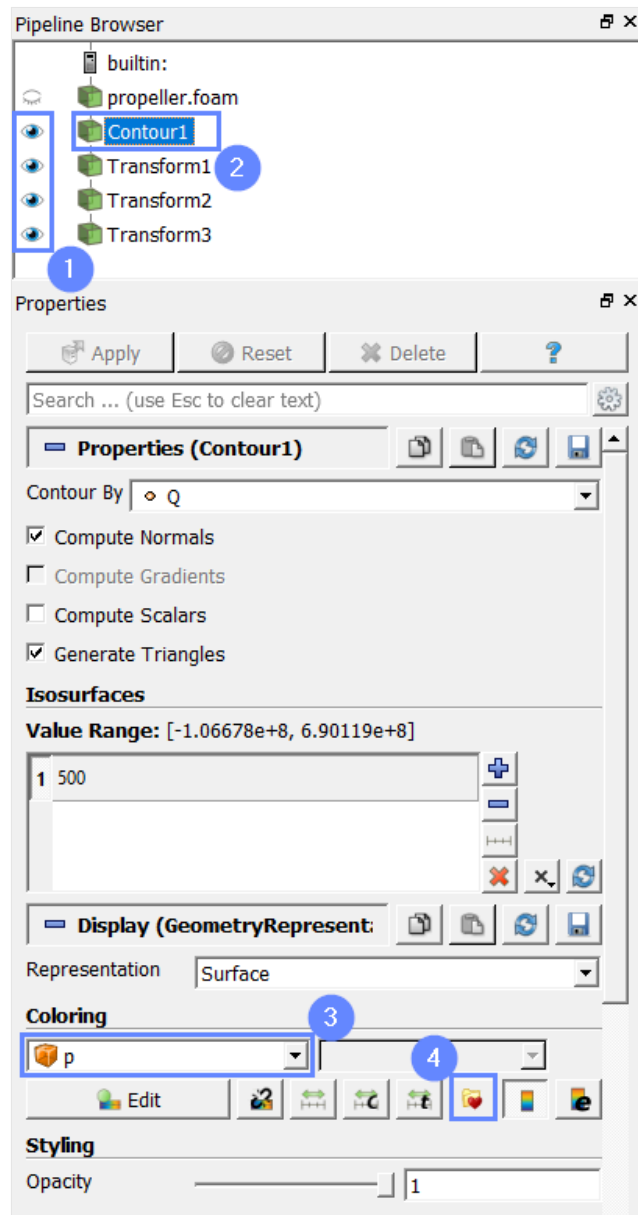
44. Final Results

In the 3D view the contour of full propeller should appear

45. Advanced Postprocessing with ParaView

This concludes the tutorial, covering all the aspects we intended to showcase. For a finely tuned presentation of the results, you may take advantage of the more advanced features.

In ParaView, you can display streamlines, contour plots, vector fields, line or time plots, calculating volume or surface integrals and create animations.
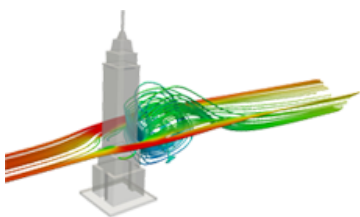
To familiarize yourself with the ParaView capabilities, it's worth checking out our video tutorial, *Paraview CFD Tutorial - Advanced Postprocessing in ParaView*, in which we demonstrate some of the most commonly used post-processing techniques.

[Paraview CFD Tutorial - Advanced Postprocessing in ParaView](#)

SimFlow
CFD Software made EASY

SimFlow CFD

OpenFOAM GUI

CFD Software

Download

Pricing

Resources

Validation

Tutorials

What is CFD

CFD Simulation

CFD Analysis

Faq

Company

License Agreement

Privacy Policy

About

Contact