

# **Volute Tutorial**

## **Rapid Adoption Kit (RAK)**

Product Version: Fidelity 2022.1  
June 2022

---

## Copyright Statement

© 2022 Cadence Design Systems, Inc. All rights reserved worldwide. Cadence and the Cadence logo are registered trademarks of Cadence Design Systems, Inc. All others are the property of their respective holders.

---

# Contents

---

Purpose.....	4
Audience .....	4
Terms.....	4
Overview .....	4
Geometry .....	4
Starting Fidelity.....	5
Geometry Context .....	5
Domain Context.....	9
Mesh Context .....	10
Simulation Context .....	15
Monitor the Convergence .....	16
Result Analysis Context.....	17
Support .....	21
Feedback .....	21

### Purpose

The purpose of this RAK is to guide you through the basic process of importing a geometry, creating a volume mesh, solving the flow field in the domain, and post-processing the CFD results. This RAK is solely intended to show the basic usage of Fidelity and is not designed to provide a thorough description of all the available options and parameters.

### Audience

This document is intended for Fidelity users who wish to be guided through the end-to-end CFD workflow.

### Terms

CFD            Computational Fluid Dynamics

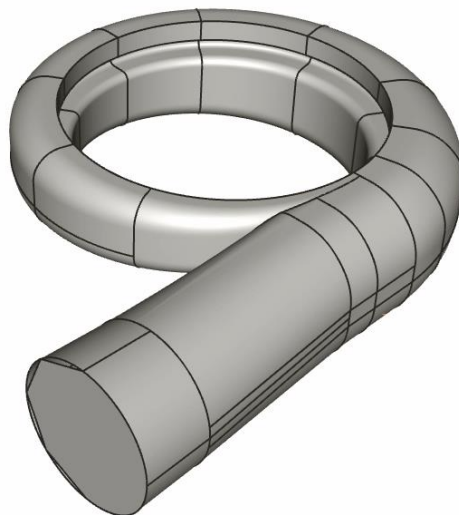
CAD            Computer Aided Design

### Overview


This tutorial contains a brief overview of the operations necessary to create an unstructured volume mesh, solve the flow field with a density-based CFD solver, and post-process the results. This exercise will help you to become familiar with the different contexts in Fidelity, from Geometry to Result Analysis.

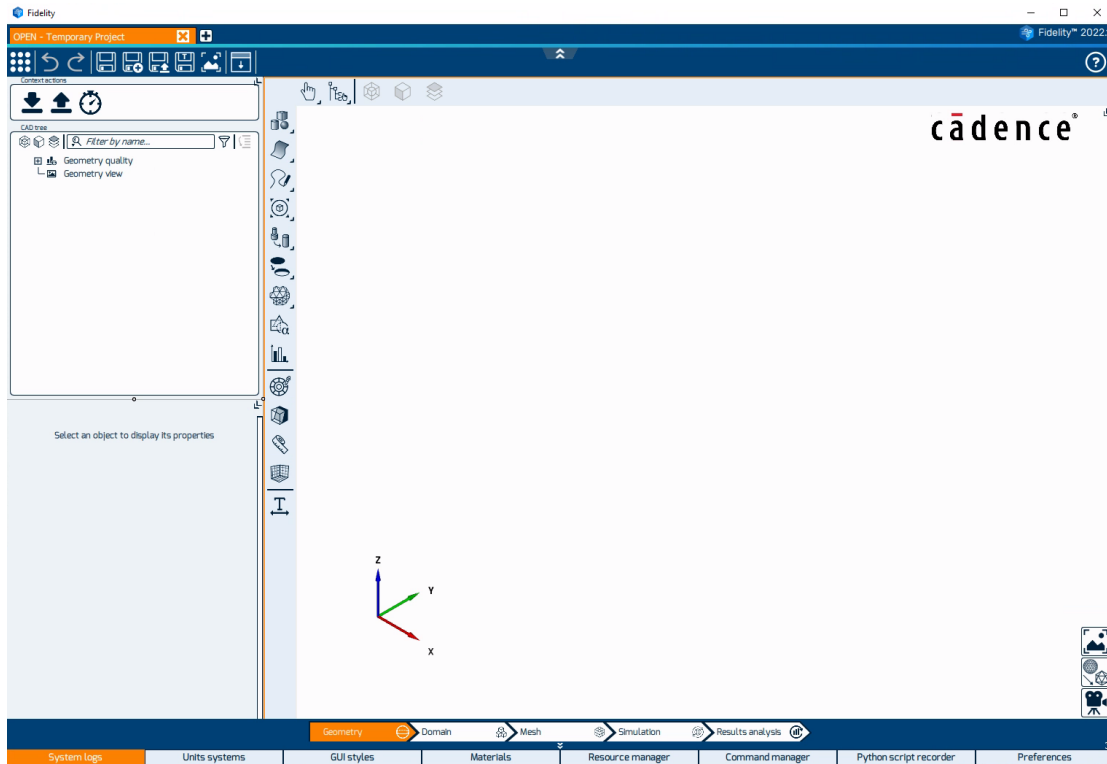
### Geometry

The geometry under consideration is a turbocompressor volute.



## Starting Fidelity


1. Launch OMNISTM.
2. Select the **OPEN** template to create a full CFD project from geometry definition to post-processing.
3. Save the project .

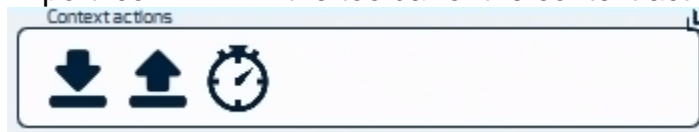


## Geometry Context



The geometry STEP file used in this tutorial can be downloaded from [this page](#). Extract and save this file to a local directory before using it.

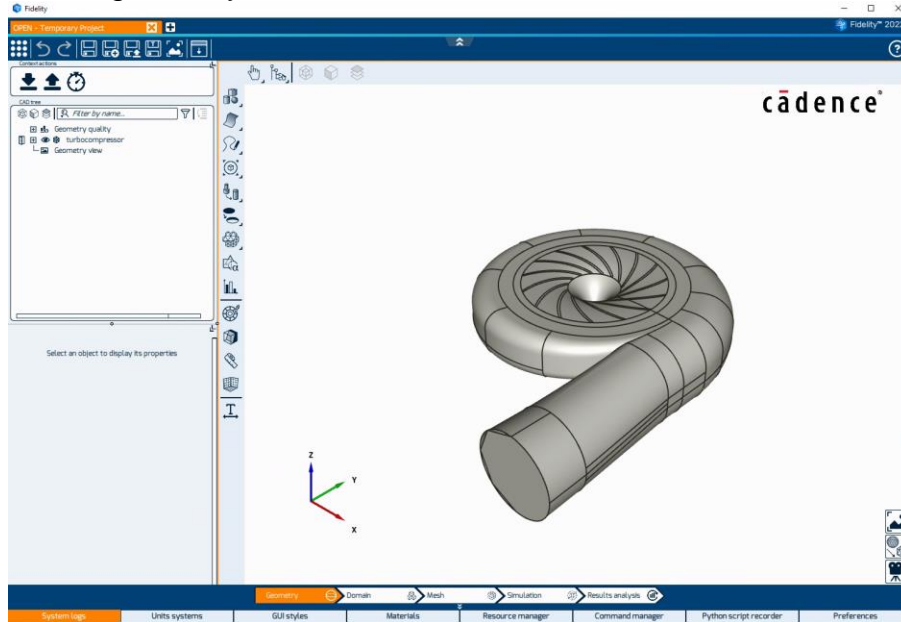
1. Click on the import icon  in the toolbar of the context actions toolbar.



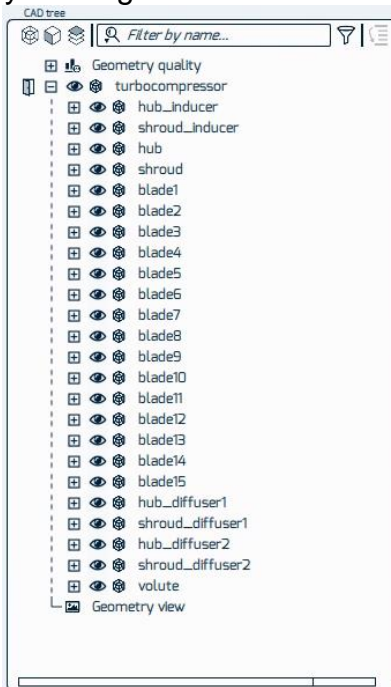
2. Select the CAD file *turbocompressor.step* in the file chooser.
3. Keep the tessellation quality to **Fine**.
4. Click on **Import** file.

## Volute Tutorial for the On-Cloud Platform

The imported CAD geometry is loaded and shown in the 3D view.



5. Expand the CAD tree by clicking on the  icon.



6. Right-click on **volute** and select **Isolate** to show the volute only in the 3D view.

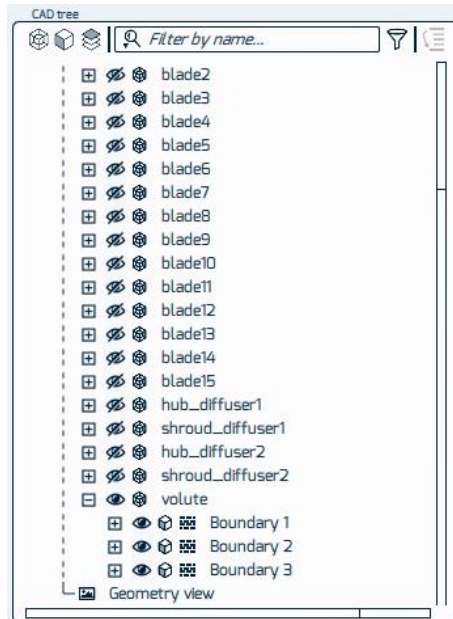
7. With the **volute** assembly selected in the tree, select the icon **Regroup Surfaces**



in the toolbar. Keep the **Feature Angle** equal to 40 degrees and click **Apply**.






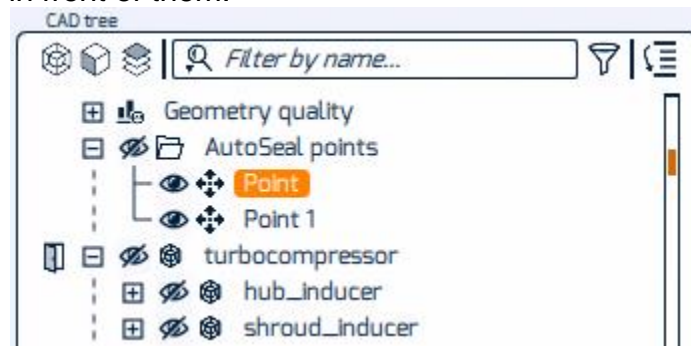
The volute assembly should now contain 3 boundaries.



8. Since the outlet patch does not properly close the volute, it is better to delete it. Select the volute outlet in the tree (**Boundary 2**), right-click on it and delete it.

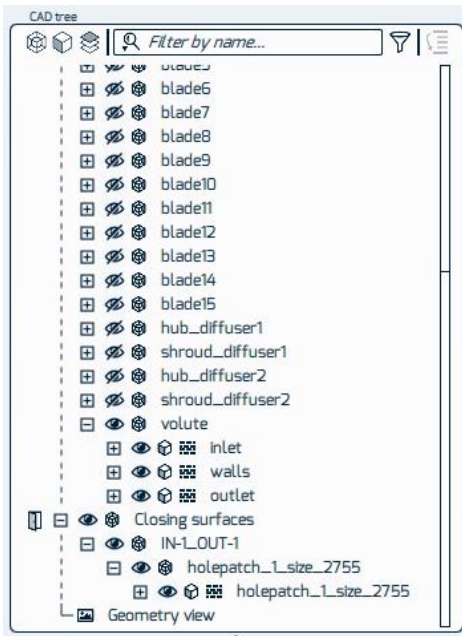
We will now close the volute model with a new outlet boundary.

9. In the toolbar, select the icon **Close holes**  and then select **Autoseal** .
10. Increase the **Minimum hole size** to 0.01.
11. Click on **Add Inside point** and **Add Outside point** so that two new points are added to the CAD tree. You can show or hide the points by toggling on or off the  symbol in front of them.



12. Click and drag the yellow dots in the 3D view to move the **inside** points outside of the volute geometry and the **outside** point within the volute geometry.
13. Select both points and the volute assembly in the tree using the Ctrl key and click **Apply** to launch the Autoseal process. When the process is completed, an additional closing surface is generated and added to the tree.
14. Locate the newly created boundary in the tree by expanding the Closing Surface assembly, right-click on the boundary, and **Copy**. Then, select the **volute** assembly, right-click and **Paste** the outlet boundary within the volute assembly.
15. Rename the inlet and outlet boundaries for clarity by right-clicking on the boundary names and selecting **Rename**.






## Domain Context



1. Switch to the domain context by clicking in the workflow. A domain is a region where the physical quantities (pressure, temperature, velocity, ...) will be solved.

2. In the context actions toolbar, click on **Create/Edit Domain** .

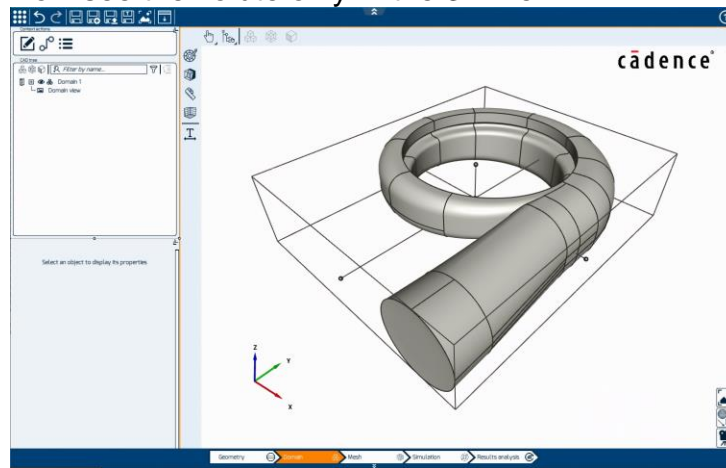


3. Click on the + symbol to create a new domain.



4. Select the **volute** assembly only in the tree by expanding the turbocompressor assembly to the right level.

5. Click on **Add assemblies**.
6. Click on **Close**.
7. You should now see the volute only in the 3D view.



8. Move the seed point by dragging the black dots at the end of the 3 intersecting black lines so that the seed point is located within the volute.

The specified seed point should not be too close to the geometry boundary. The distance between the seed point and the geometry boundary should be at least two times the local size of the cell containing the seed point. If this condition is not fulfilled, the mesh generation may fail, or the generated mesh may be invalid.

9. Change the boundary condition types of the inlet and outlet boundaries by clicking on them in the tree and changing the **BC Type** in the bottom left panel.

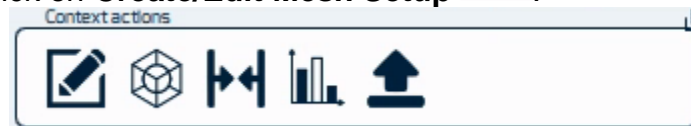
## Mesh Context



In this context, we will set up and generate an unstructured mesh for the volute using Fidelity /Hexpress.

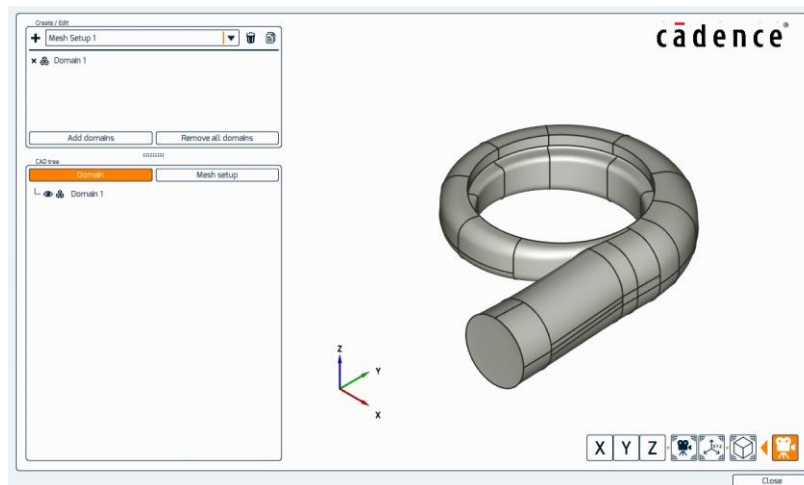
1. Switch to the mesh context by clicking in the workflow.

2. In the pie, click on **Create/Edit Mesh Setup** .



3. Click on the **+** symbol and select **Hexpress** to create a new mesh setup.
4. Select **Domain 1** in the tree and **Add domains** it to the mesh setup.

5. **Close** the widget.

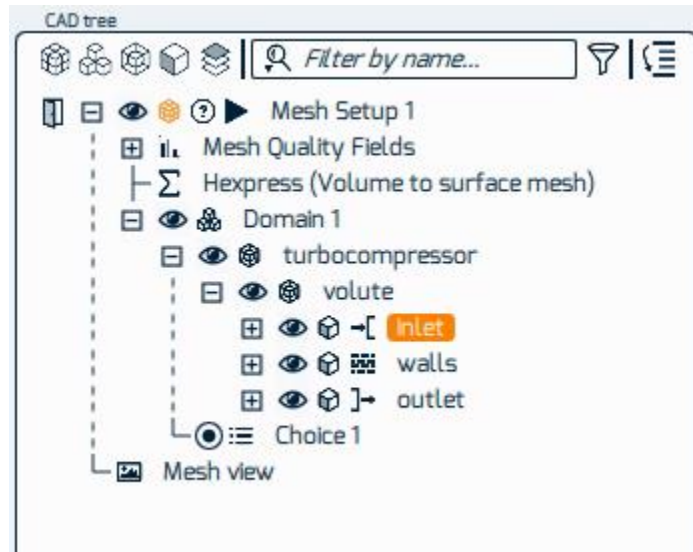





6. Select **Mesh Setup 1** in the tree and go down to the **Global** tab. In the **Global** tab, apply the following settings:
- Change the cells type to **Full Hexa**,
  - Keep the initial mesh type to **Cartesian**,
  - Set the initial cell size to 0.01 [m].

In the 3D view, the initial cell size is represented for the volute domain.




7. In the tree, expand the domain by clicking on the **+** symbols until the boundaries are listed.




9. Select the inlet boundary. In the local tab, activate the **Uniform** checkbox and set the refinement level to 1. The cell size on the surface is modified automatically.
10. Select the boundary walls.
  - a. In the Local tab, activate the **Auto edge proximity** checkbox. Set the **Min cell size** to 0.001 [m].
  - b. In the local tab, activate the **Curvature** checkbox. Keep the **Divisions for curvature** equal to 8 and set the **Min cell size** to 0.001 [m].
  - c. Activate the **Viscous Layers** checkbox and set the following settings:
    - i. Keep the **Insertion method** to **Inflation**,
    - ii. Increase the **First layer thickness** to 0.0001 [m],
    - iii. Keep the **Stretching ratio** equals to 1.2,
    - iv. Activate the **Floating** number of layers,
    - v. Keep the **Target, Minimum** and **Maximum** number of layers to 15, 5, and 20, respectively.
11. Save the project .
12. Click on the **Mesh Setup 1** in the tree, go to the **Control** tab, and click on the  symbol to open the resource control.
13. The parallel option is activated by default. Keep the default settings.
14. Click **Start** or on the  icon in front of the **Mesh Setup 1** in the tree to launch the mesh generation.

The progress of the mesh generation can be monitored at the mesh setup level in the tree. At the end of the mesh generation, a pop-up appears to inform that the mesh has been generated.


15. Close the info window.

In front of the Mesh Setup for Volute in the family tree, a  icon is appearing to inform that a successful 3D mesh is available.

16. Save the project .

18. In the tree, right-click **Mesh Setup 1** and select **toggle mesh/geometry** to visualize the generated mesh. In the 3D view, the volute mesh is displayed on the solid walls, inlet and outlet.




19. Check the mesh quality statistics by selecting the **Show mesh quality**  icon in the context actions toolbar.

20. Open the **Hexpress** tab to check the number of cells and mesh quality criteria.

21. Close the mesh quality statistics window.

22. Load the mesh for post-processing:

a. Select **Mesh Setup 1** in the tree.

b. In the **Control** tab, select the resources control  button next to the **load results** button. The parallel option is not activated by default.

c. Keep the default settings and click on **Load Results**.

It is mandatory to load the mesh before performing any post-processing step. The spinning circle in front of the mesh setup in the tree indicates that the mesh is loaded and available for analysis.

23. In the toolbar, click on the **create new cutting plane** icon .

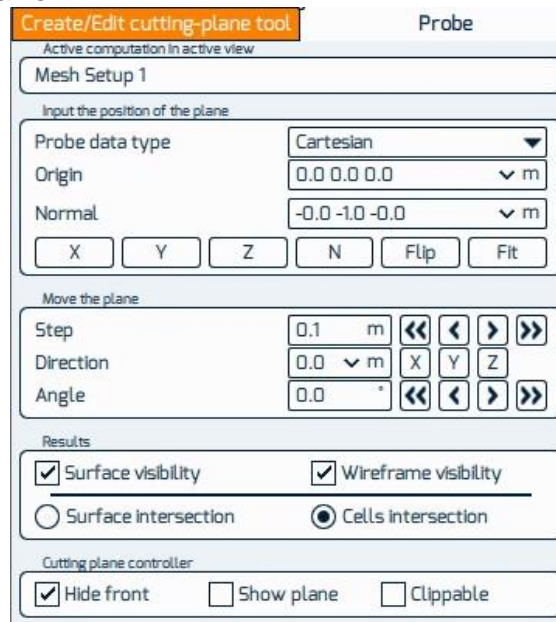
24. Change the parameters of the cutting plane in the property panel under the tab **create/edit cutting-plane tool** as follows:

25. Change the **Origin** to 0 0 0

26. Change the position of the plane to **Y** in order to see the middle cut section of the volute.

27. **Flip** the cutting plane if necessary.

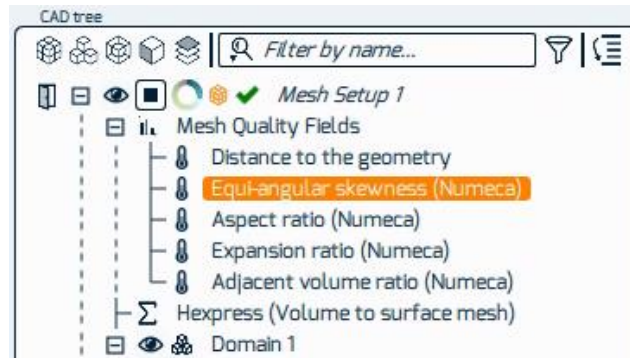
28. Activate both **surface visibility** and **wireframe visibility**.
29. Toggle **cells intersection**.
30. Deactivate **show plane**.



The surface mesh of the volute on the cutting plane is now shown.



31. Deactivate the option **Hide front** to see the whole volute again.
32. Make the volute transparent by right-clicking on the **volute** assembly, selecting **Rendering Options** and activating the **Transparency** option and deactivating the **Wireframe** option.
33. On top of the tree, expand the **Mesh Quality Fields** menu and select the **Equi-angular skewness**.

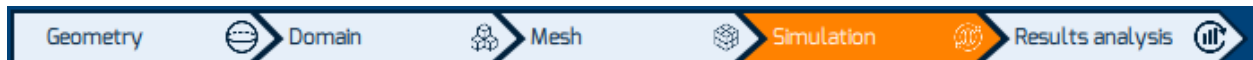


34. Click on **Create cell range**.
35. Change the **minimum** to 0.8 and keep the **maximum** to 1.
36. Click on **Compute**, **Show cells**, **Show cell locations** to see the location of the cells with the highest skewness.






37. Click on the **stop the task** icon in front of the **Mesh Setup 1** in the tree to stop the mesh analysis.

## Simulation Context



In this context, we will set up a flow simulation in Fidelity /Open.

1. Switch to the simulation context by clicking in the workflow.
2. In the context actions toolbar, click on **Create/Edit** .
3. Click on the **+** symbol and select **Open solver** to create a new simulation.
4. Select **Domain 1** in the tree and **Add domains to add** it to the simulation.

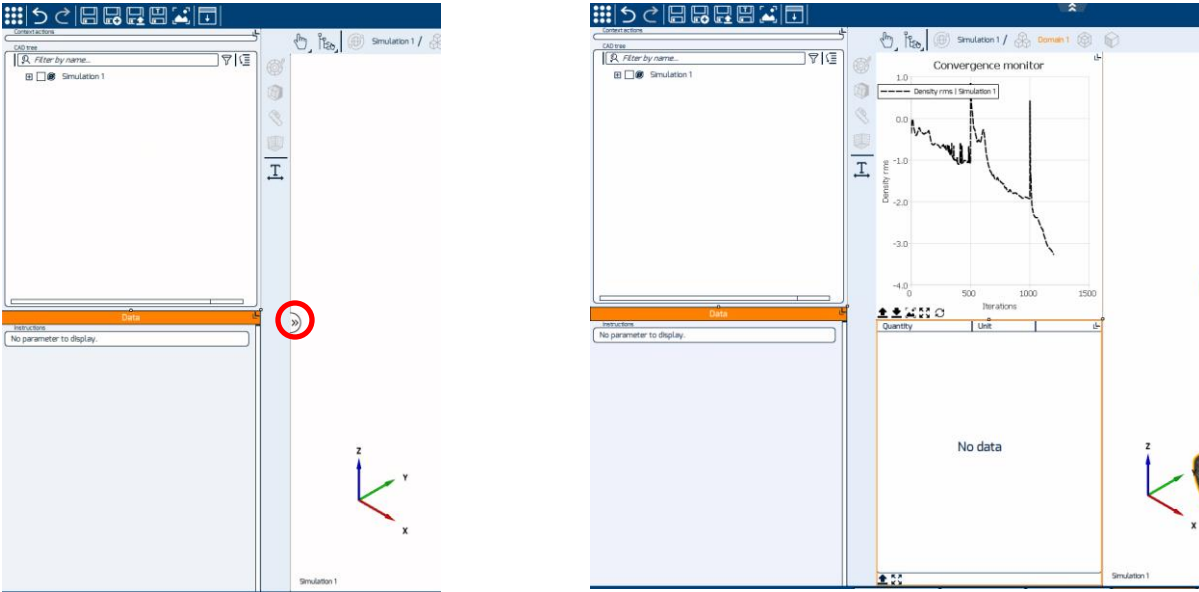
5. **Close.**
6. Select the **Simulation 1** in the tree and switch to the **Numerics** tab.
7. Activate the [CPU booster](#).
8. In the Output tab, select the variables **Mach Number**, **Total Pressure**, and **Y+**.
9. Expand the tree to see individual boundaries.
10. Select the **inlet** boundary and apply the following boundary conditions:
  - a. Subsonic – Cylindrical – Velocity direction ( $V$  extrapolated)
  - b.  $V_r/V = 0.4$
  - c.  $V_t/V = -0.9$
  - d.  $V_z/V = 0$
  - e. Absolute total pressure = 120,000 [Pa]
  - f. Keep the other values as default.
11. Select the **outlet** boundary and apply the following boundary conditions:
  - a. Subsonic – Cartesian – Static pressure imposed
  - b. Static pressure = 118,000 [Pa]
12. Select the **Domain 1** in the tree and go down to the **Materials** tab. Keep the default fluid, **Air (Perfect Gas)**.
13. In the **Physics** tab, choose the **k-omega SST** turbulence model and deactivate the option **With extended wall functions**.
14. In the **Initial Solution** tab, keep the default values.
15. Select the **Simulation 1** in the tree.
16. In the **Control** tab, select the resources control  button and activate the **Parallel** option.
17. Activate the option **Additional analysis every 10 [s]** to turn on the co-processing while the simulation is running.
18. Save the project .
19. Click on **Start** to launch the simulation.

## Monitor the Convergence

While a simulation is running, the convergence can be monitored directly in Fidelity.

1. Click on the double arrow that separates the CAD tree from the 3D view to show the convergence monitor.





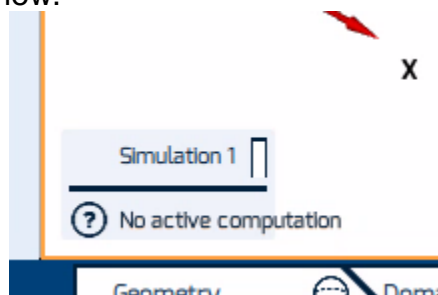
2. Left-click anywhere in the plot section of the monitor to activate different quantities and display their convergence histories.

The progress of the simulation can be observed at the simulation level in the tree. At the end of the simulation, acknowledge the warning that the convergence criteria could not be reached but all the global quantities as presented below have reached their convergence.

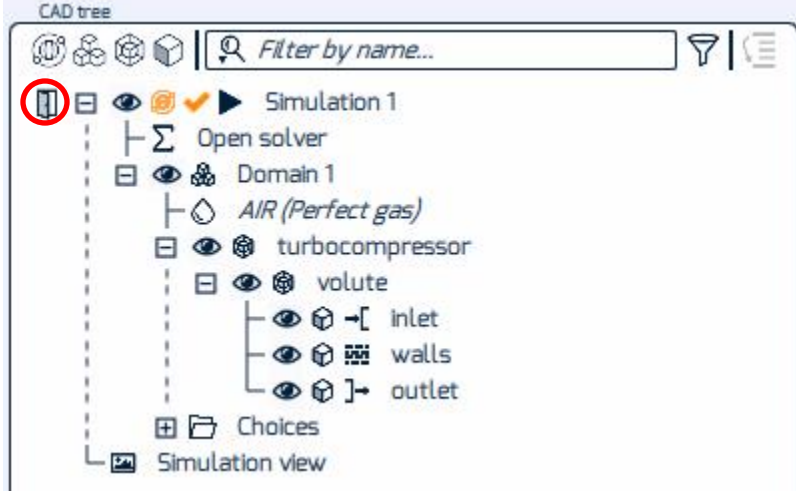
## Result Analysis Context




1. Switch to the **Results analysis** context by clicking in the workflow.
2. Ensure **Simulation 1** is available for post-processing by selecting it in the bottom left corner of the 3D view.





3. Ensure **Simulation 1** is loaded in the CAD tree by setting its state to loaded



4. Select the **Simulation 1** in the CAD tree, right-click on it and select **Rendering options**.
5. Activate then deactivate the **Wireframe** visibility check box and activate the **Transparency**.
6. Close the rendering options dialog window.
7. With the **Simulation 1** selected in the tree, click on the **Control** tab.
8. Click on the **resources** control button .
9. Unselect the **parallel** option. In this case, the results analysis will be run in serial mode.
10. Click on **Load results** to load the solution, unless the simulation is still running. If the simulation is still running, the co-processing will allow you to visualize the flow field without loading the results.

It is mandatory to load the solution in order to perform the results analysis of a simulation after the simulation has completed running. The spinning circle in front of the simulation (e.g. **Simulation 1**) in the tree indicates that the results are loaded and the solution is ready to be post-processed.

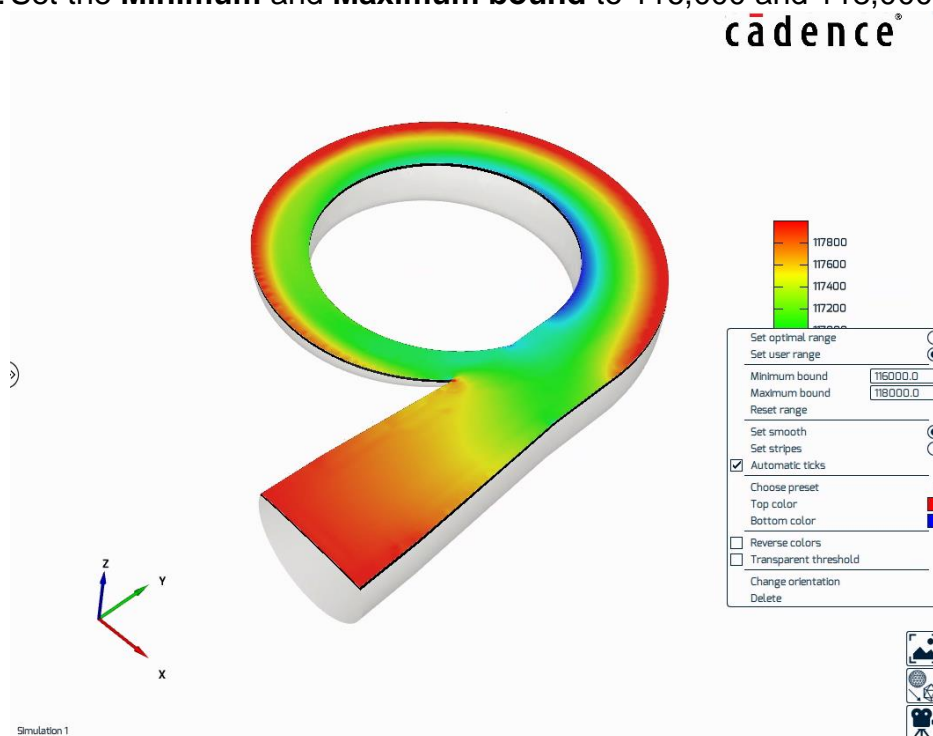
11. Create a new cutting plane by clicking on the **create new cutting plane**  symbol at the bottom right of the 3D view.
12. Change the parameters of the cutting plane in the property panel under the tab **create/edit cutting-plane tool** as follows:
13. Change the **Origin** to 0 0 0.1
14. Change the position of the plane to **Z** in order to see a transversal section of the volute.
15. Activate the **surface visibility**.
16. Deactivate **show plane**.


17. Click on **Create color contour**  in the toolbar.
18. Select **Static Pressure** in the field drop-down menu.
19. Click **Apply**.


Color contours, local values, isolines and other post-processing representations are named probes in OMNIS™. The entities at which these probes are applied to are referred as supports.

A support with loading post-processing state is represented with yellow and gray stripes in the view. The probes will then appear as soon as results are available.

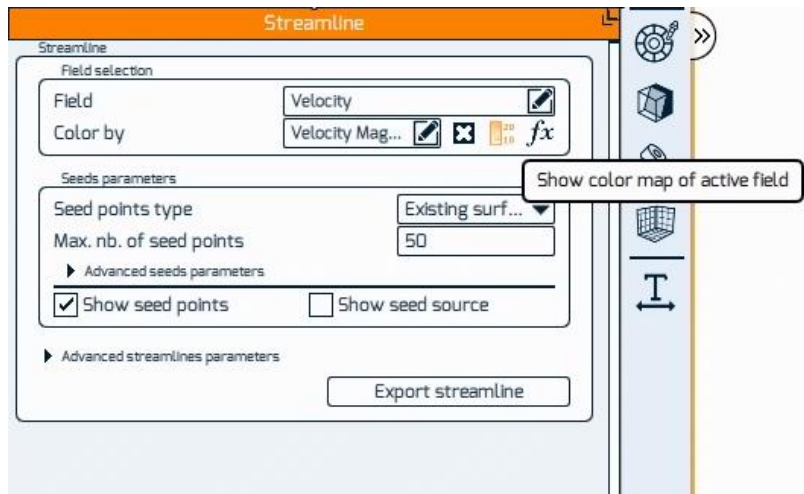
20. Right-click on the color bar in the 3D view and activate the option **Set user range**.
21. Set the **Minimum** and **Maximum bound** to 116,000 and 118,000.



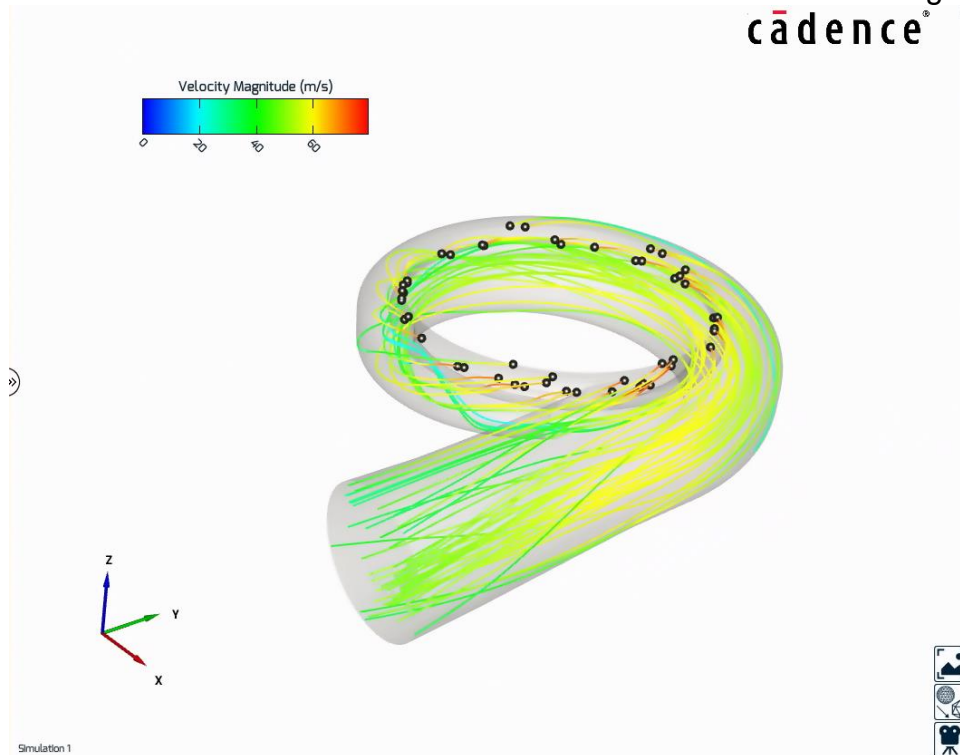
22. Deactivate the **Hide front** option in the cutting plane properties and hide the cutting plane by clicking on the  symbol in front of it in the tree.



23. From the toolbar, click on the option **Create Streamlines** .
24. In the **Field** drop-down menu, select **Velocity**.
25. Activate the **Color by** option and select **Velocity Magnitude**.
26. For the **Seed point type**, select **Existing surface**.

27. Increase the **Max. nb. of seed points** to 50.
28. Extend the tree to access the boundaries and select the **inlet** boundary.
29. Click **Apply**.
30. Select the **Streamline 1** object in the tree and click on the **Show color map of active field** to show the color bar in the 3D view.



The 3D view with active streamlines should now look like the image below:



31. Save a picture of the current view by clicking on the **Save image**  icon on the bottom right of the 3D window.
32. In the **Save image** options, deactivate the **Snapshot mode** and elevate the quality to **High**.
33. Click **OK**.
34. Click on the **Stop the task** icon in front of **Simulation 1** in the tree to stop the results analysis.
35. Save the project  .

## Support

Cadence Learning and Support Portal provides access to support resources, including an extensive knowledge base, access to software updates for Cadence products, and the ability to interact with Cadence Customer Support. Visit <https://support.cadence.com>.

## Feedback

Email comments, questions, and suggestions to [content\\_feedback@cadence.com](mailto:content_feedback@cadence.com).