

Open Source CFD

Tutorial #2: OpenFOAM with FreeCAD

Incompressible Steady Flow: pitzDaily with FreeCAD - CfdOf

Robert Castilla

Department of Fluid Mechanics

2025-26

Version 1.0

Learning Objectives:

- Create 3D geometry with FreeCAD
- Setup CFD simulation in FreeCAD with CfdOf
- Generate volume mesh
- Setup Boundary Conditions
- Run Solver ion FreeCAD
- PostProcessing with paraview

This tutorial introduces the fundamental workflow of Computational Fluid Dynamics (CFD) using OpenFOAM in a FreeCAD with the CfdOf workbench. Through the classic [pitzDaily](#) case, students will learn to set up, run, and analyze an incompressible steady-state flow simulation. The postprocessing, like in Tutorial 1, will be done with paraview.

Contents

1. Introduction and Theoretical Background	3
1.1. What is pitzDaily?	3
1.2. Governing Equations	3
1.3. Case Overview	4

2. FreeCAD Environment Setup	4
2.1. What is FreeCAD and CfdOf	4
2.2. Basic setup	5
3. 3D Geometry generation	5
4. CFD simulation with CfdOF	7
4.1. First steps	7
4.2. Setup of boundary conditions and mesh	8
4.3. Running the Simulation	11
5. Post-processing	12
5.1. Velocity profile on vertical line	13
5.2. Pressure average in inlet	15
6. Common Issues and Troubleshooting	15
6.1. Streamlines stop suddenly or contours have holes	15
6.2. Error: "cannot find blockMeshDict"	16
6.3. Simulation Diverges (Residuals Grow)	16
7. Student Exercises and Extensions	16
7.1. Basic Exercises (Required)	16
7.2. Advanced Exercises (Optional)	16
8. Deliverables and Assessment	16
8.1. Required Files	17
8.2. Report Questions	17
8.3. Submission Format	17
A. Appendix: Useful Terminal Commands Reference	18
A.1. OpenFOAM-Specific	18
A.2. General Linux Commands	18
B. Quick Reference: File Locations in pitzDaily	18

1. Introduction and Theoretical Background

1.1. What is pitzDaily?

The `pitzDaily` case simulates 2D steady, turbulent flow in a backward-facing step configuration. It's included in the OpenFOAM tutorials and serves as an excellent starting point due to its:

- Simple geometry but interesting physics (recirculation zone)
- Complete setup with all necessary files
- Quick runtime (minutes on most computers)

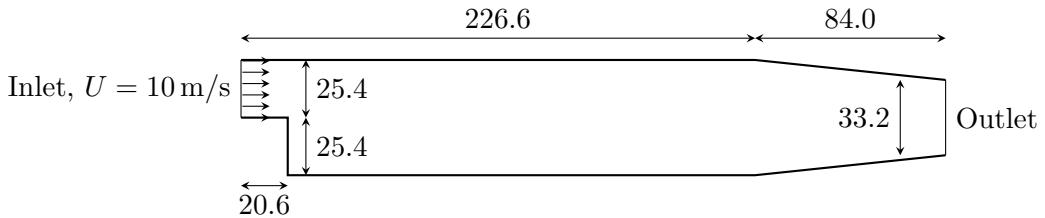


Figure 1: pitzDaily geometry. Lengths are in millimeter.

1.2. Governing Equations

The `simpleFoam` solver implements the incompressible Reynolds-Averaged Navier-Stokes (RANS) equations:

$$\nabla \cdot \mathbf{U} = 0 \quad (1)$$

$$\nabla \cdot (\mathbf{U}\mathbf{U}) = -\nabla p + \nabla \cdot (\nu_{\text{eff}} \nabla \mathbf{U}) \quad (2)$$

where $\nu_{\text{eff}} = \nu + \nu_t$ is the effective viscosity (molecular + turbulence viscosity).

1.3. Case Overview

Parameter	Value
Solver	<code>simpleFoam</code> (steady, incompressible)
Turbulence model	$k-\epsilon$ (standard)
Flow regime	Turbulent
Geometry	2D backward-facing step
Reynolds number	$\sim 2 \times 10^4$ (based on step height)
Expected runtime	5-10 seconds

Table 1: pitzDaily case specifications

2. FreeCAD Environment Setup

2.1. What is FreeCAD and CfdOf

[FreeCAD](#) is a free, open-source parametric 3D CAD modeler with a modular architecture. It is extensible through workbenches.

[CfdOf](#) is an workbench for FreeCAD which provides a basic CFD workflow.

CfdOf is not installed in FreeCAD by default. Instructions for the installation can be found in its home page.

Figure 2 show the startup window of FreeCAD.

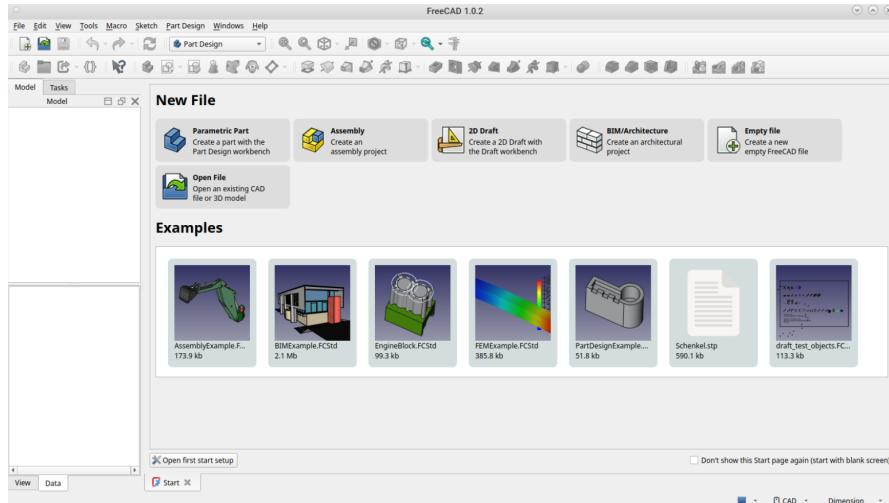


Figure 2: Startup screen of FreeCAD

2.2. Basic setup

By default, FreeCAD is configured to work with millimeters. Since natural OpenFOAM system of units is the International one, the first task is not change the default unit system. To do that, navigate to **Edit → Preferences → General** and change the default unit system to meters.

3. 3D Geometry generation

Clik on **New** to create a new document. By default, the **Part Design** workbench is activated.

Click on **sketch** button (see Figure 3) to create a new sketch

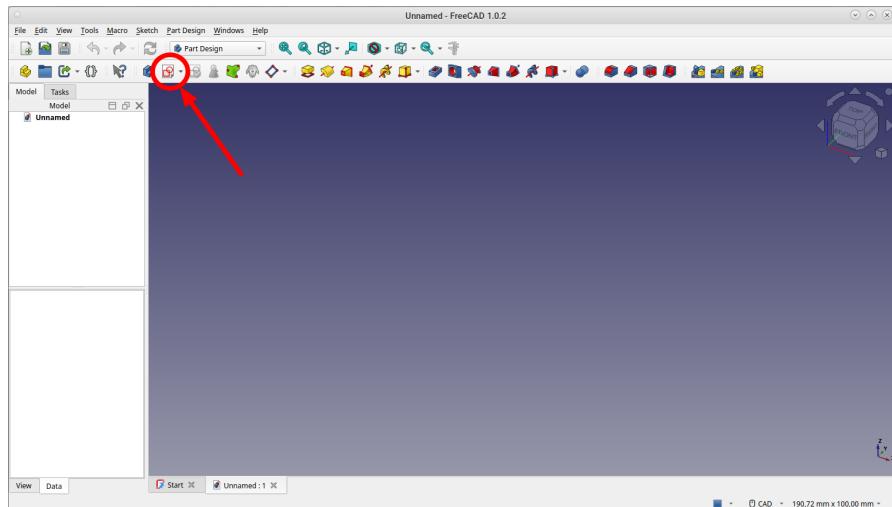


Figure 3: To make the initial sketch

Select plane XY and click **OK**.

Loosely draw a polyline following the shape of the pitzDaily geometry (Figure 1). Don't worry about constraints and dimension; it will be defined later (Figure 4).

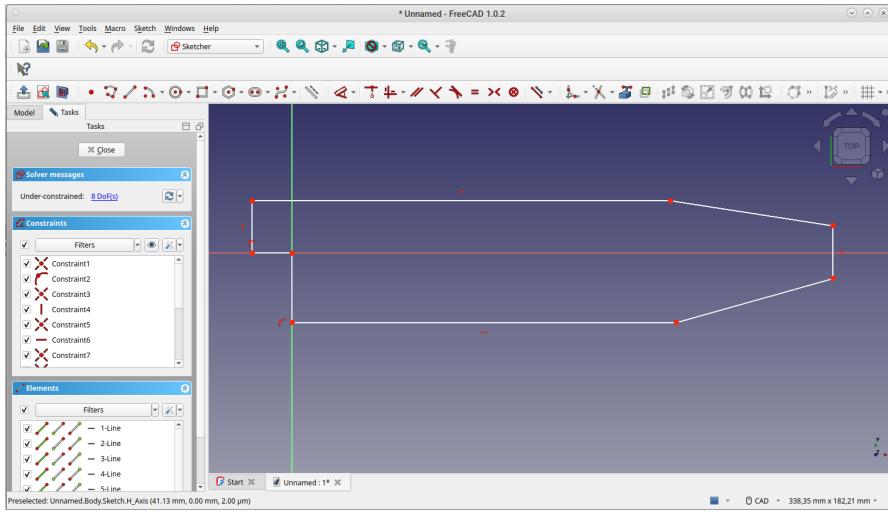


Figure 4: The sketch, without correct dimensions

Now the correct dimensions can be defined with the constraints tools. When the constraints are right, the sketch becomes green (Figure 5).

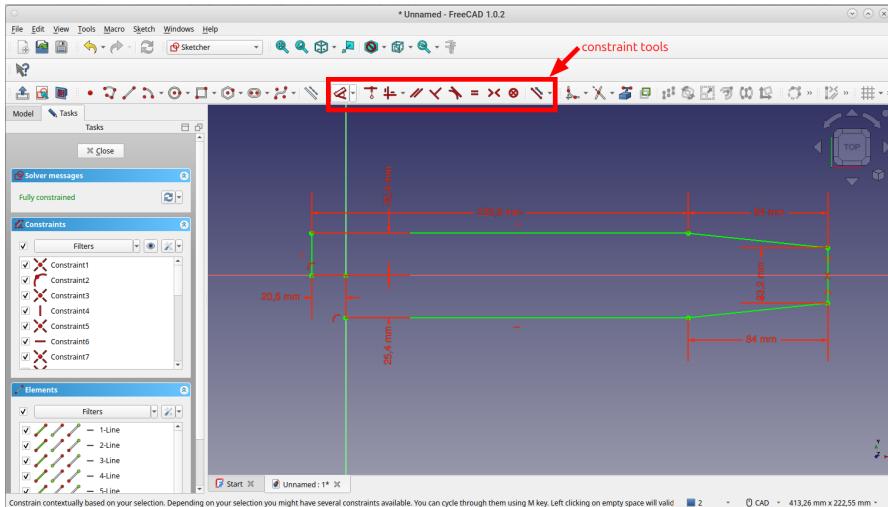


Figure 5: Sketch with dimensions and constraints.

The sketch can be closed by clicking **Close**. The last step is to extrude this sketch with the **Pad** tool. Choose a length of 4 mm (Figure 6)

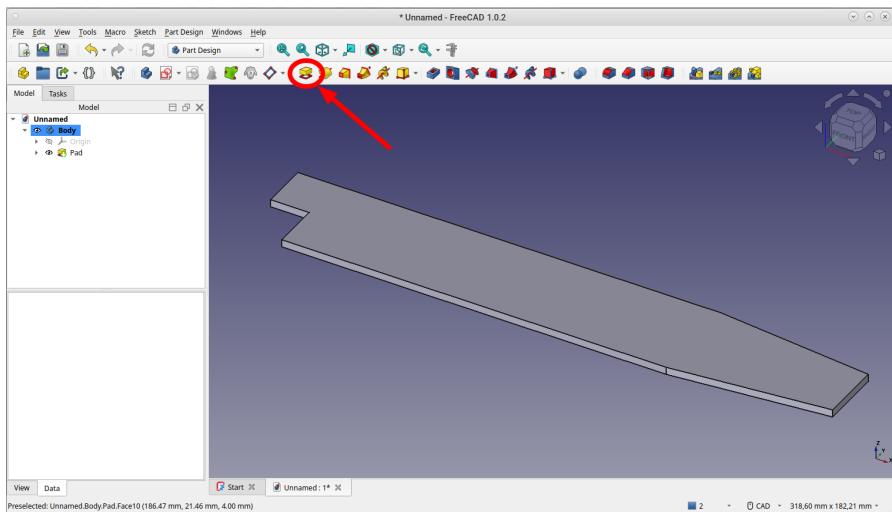


Figure 6: 3D part for the pitzDaily geometry.

4. CFD simulation with CfdOF

4.1. First steps

Select the **CfdOF** workbench and, with the **Pad** selected, click on the **Analyses Container** icon.

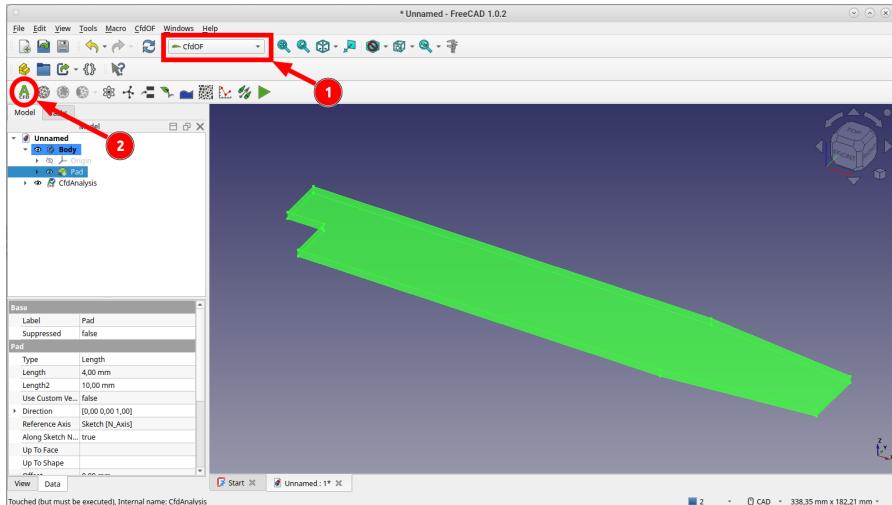


Figure 7: CfdOF workbench and analysis container

CfdOF will create the OpenFOAM case on disk in order to make the simulation. By default, the cases are stored in the **/tmp** folder. Change this default folder to

`~/Tutorials/Tutorial_2` in **CfdOF** → **Open preferences** → **Default output directory**. Make the folder, if necessary. Click **OK** to save the options and close the preferences window.

Expand the **CfdAnalysis**, and double-click on **Physics model**. Select:

- Steady
- Single phase
- Isothermal
- Viscous
- RANS → kOmegaSST

The window can be closed with **OK**.

Double-click on **Fluid Properties** and click on **OK** to select **Air**, the default option.

4.2. Setup of boundary conditions and mesh

Select the face for the inlet and click on the icon for **Fluid boundary**.

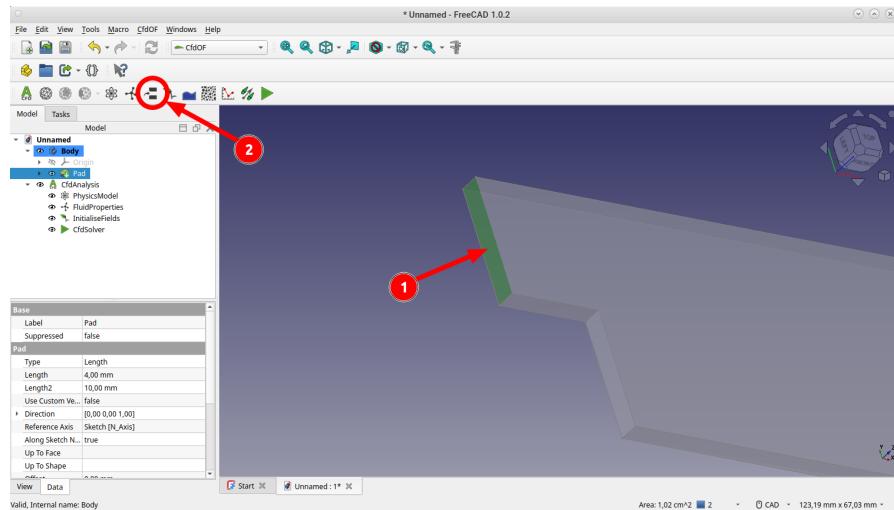


Figure 8: Selection of the inlet boundary condition.

Define the boundary with:

- Boundary: **Inlet**
- Sub-type: **Uniform velocity**
- Velocity: 10 m/s in the X direction

- Pressure: 100 kPa
- Turbulence specifications:
 - TKE (k) : $0.375 \text{ m}^2/\text{s}^2$
 - SDR (ω) : 440.15 Hz

Click **OK** to define the inlet boundary condition.

Repeat the process for

outlet: 1 face. Boundary **Outlet**, Sub-type **Static Pressure**, with pressure 0 kPa (relative pressure).

wall: 6 faces. Several faces can be selected with the **Ctrl** key. Boundary **Wall**, Sub-type **No-slip**.

constraint: 2 faces, normal to Z direction. Boundary **Constraint**, Sub-type **Symmetry**

Make the mesh by selecting the **Pad** and clicking on the **CFD mesh** icon

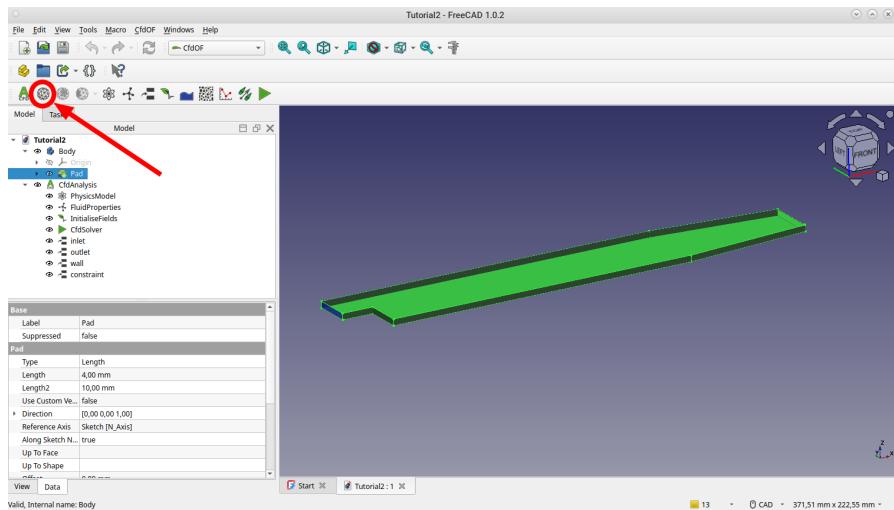


Figure 9: Making the mesh

Keep **cfMesh** as mesh utility, and write **1 mm** as **Base element size**. Write the mesh case on disk. If desired, the mesh case can be explored and edited. Generate the mesh with **Run mesher**. It will take just some seconds. The mesh can be visualized directly in FreeCAD with **Load surface mesh** (Figure 10) or with **ParaView** (Figure 11).

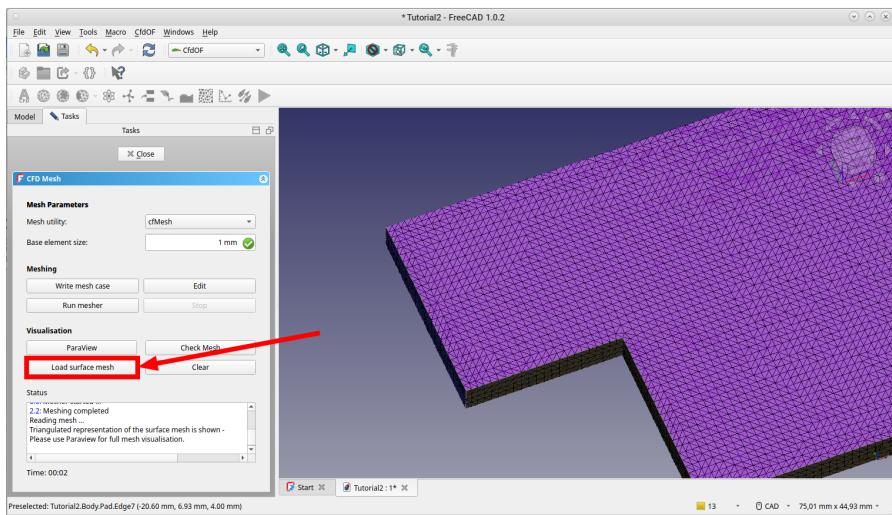


Figure 10: Mesh visualized in FreeCAD

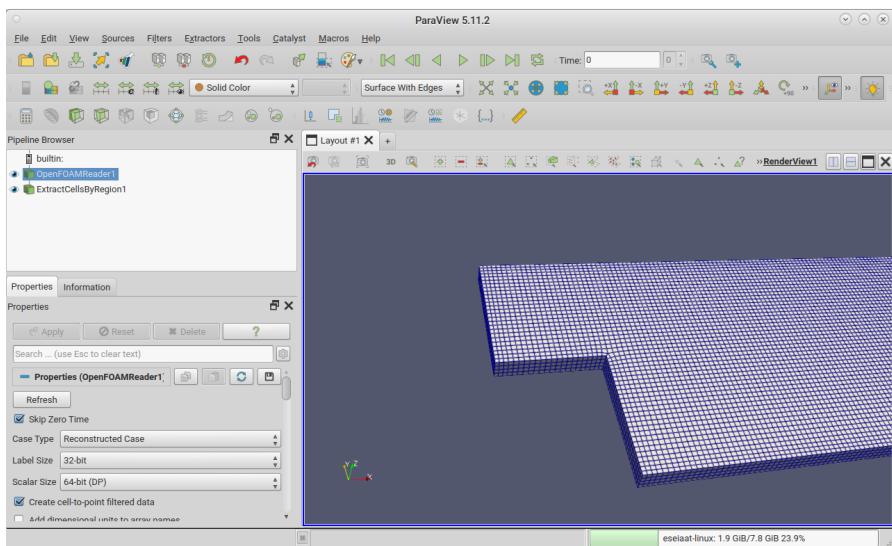


Figure 11: Mesh visualized with ParaView.

Finally, since the solver is an iterative process, an initial state of the problem has to be provided. It is done with the **InitialiseFields** icon (Figure 12).

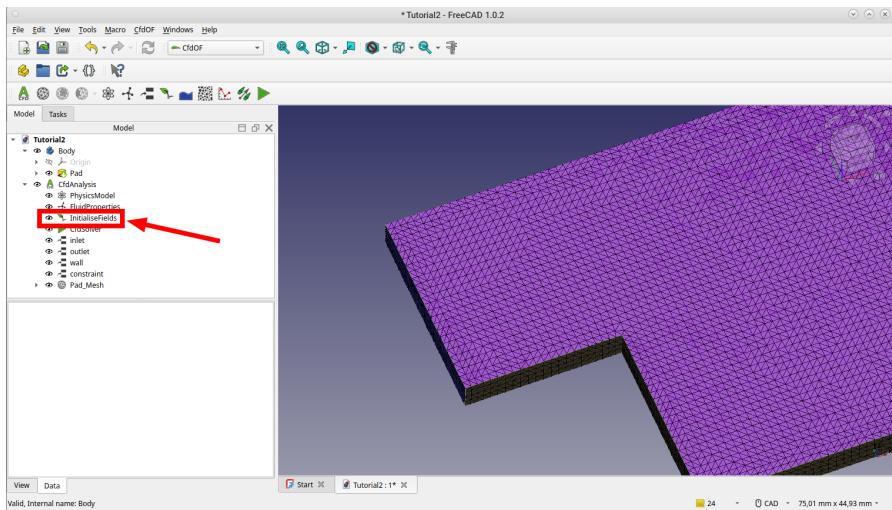


Figure 12: Icon to initialise the fields for the iterative computation.

For **velocity**, keep the potential flow. For **pressure** use the value from the **outlet** boundary, and for **turbulence**, from the **inlet** one. Click **OK**.

4.3. Running the Simulation

Before running the solver, the program has to be setup to use only one processor.

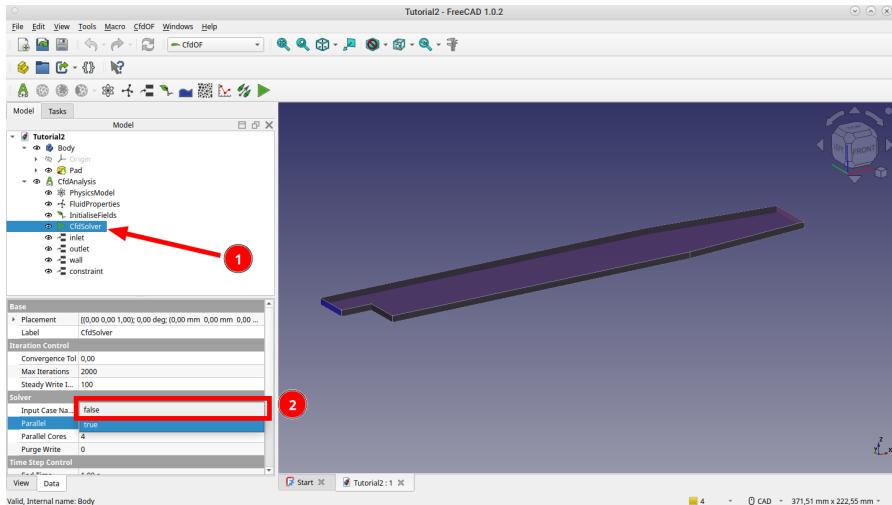


Figure 13: Selection of one core for serial simulation.

To run the solver, double-click on the icon **Cfdsolver**, and write the case to the disk.

Click on the **Run** button and the **Residuals** of the equations will be visualised on the screen.

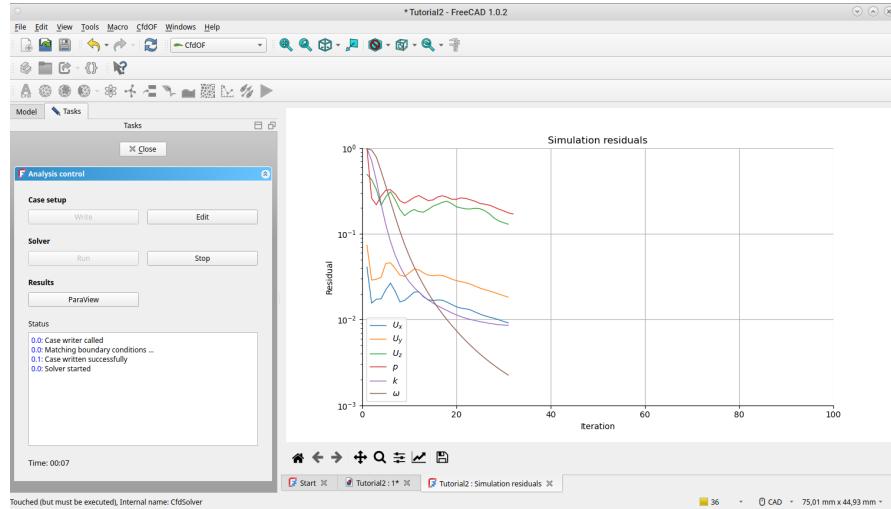


Figure 14: Residuals of the equations in the iterative process.

The simulation will stop when all the residuals reach a value below 10^{-3}

5. Post-processing

The results of the simulation can be processed with **ParaView**, in the same way as in Tutorial 1.

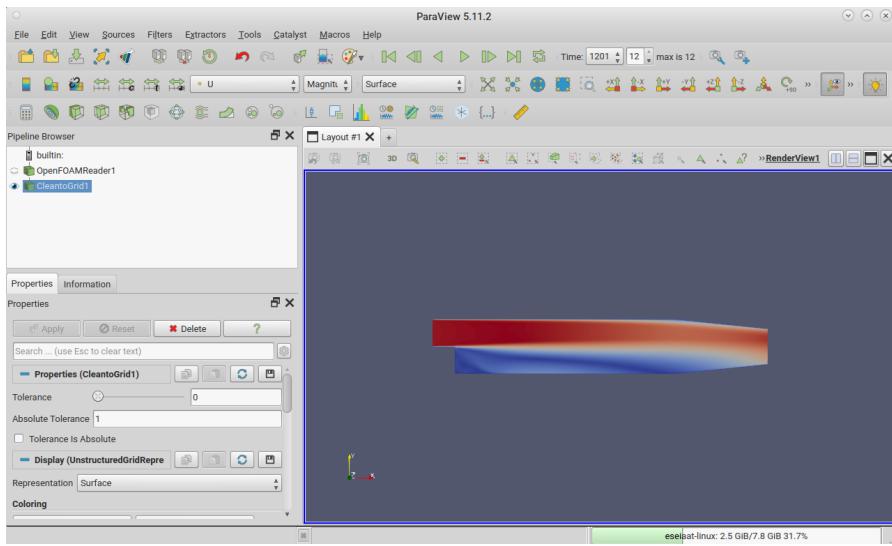


Figure 15: Simulation results in **ParaView**.

When **CfdOf** launches **ParaView**, it automatically applies a "**Clean to Grid**" filter. This filter:

1. **Removes duplicate vertices:** Merges points that occupy the same spatial location (tolerance: 10^{-6} m)
2. **Converts polyhedral cells:** OpenFOAM's arbitrary polyhedra → ParaView's standard element types
3. **Fixes surface normals:** Ensures all face normals point outward from the domain
4. **Enables advanced filters:** Many ParaView operations require regular grid structure

IMPORTANT: For most visualization tasks, keep this filter active. It ensures reliable rendering of pressure contours, velocity vectors, and streamline plots.

5.1. Velocity profile on vertical line

Let's compute the profile of the x component of the velocity along a vertical line. Click on the **Plot Over Line** icon (Figure 16). Alternatively, this filter can be activated with **Filter → Data Analysis → Plot Over Line**

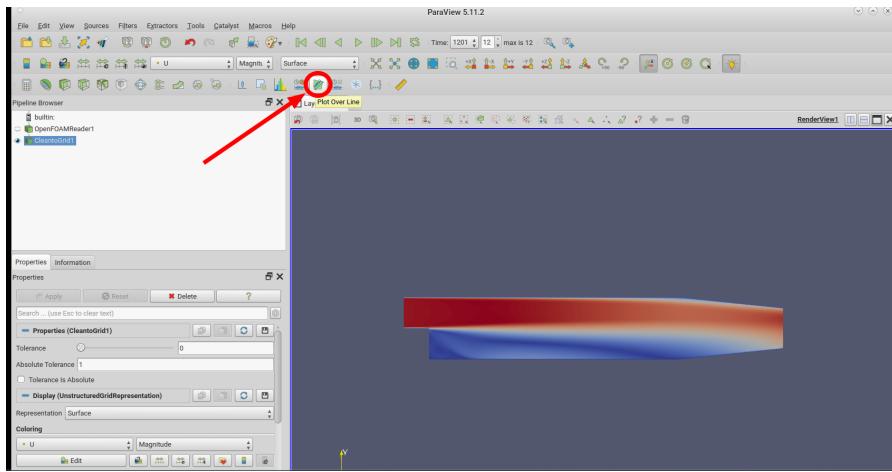


Figure 16: To plot variables over a line

Define the line from the point $(0.0254, -0.0254, 0.002)$ to the point $(0.0254, 0.0254, 0.002)$, with a resolution of 200 points, and click on **Apply**. In a window to the right the plot of several variables will be shown. By scrolling down the list of variables to be plotted can be modified. Deselect all but **U_X**, and put **Points_Y** as **X Array Name** (Figure 17).

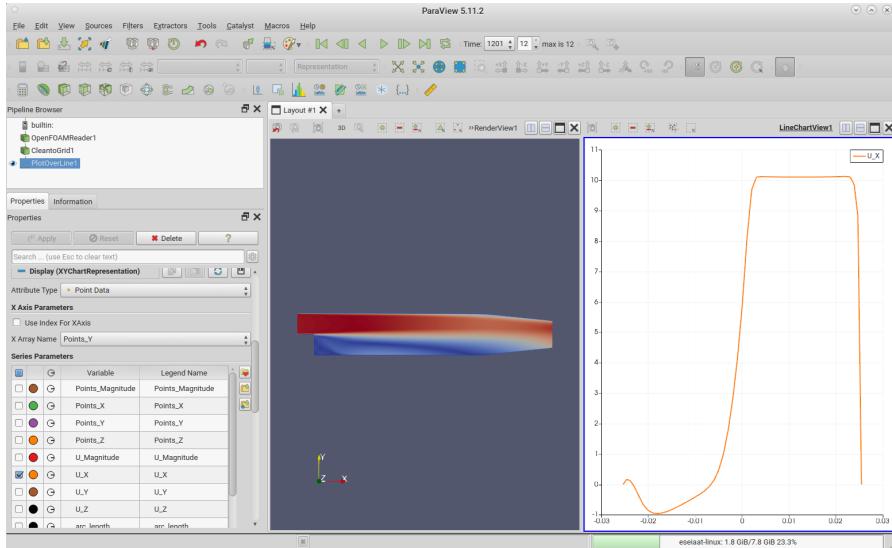


Figure 17: Plot of x component of velocity over a vertical line

The data of this plot can be saved as a CSV file with **File → Save data**. Select **csv** as file type, write a name like **U.csv** or similar and click on **OK**. Click on **Choose arrays**

to write and select only **U**. Click on **Use Scientific Notation** and select. Click on **OK**. The generated file looks like that:

```
"U:0","U:1","U:2","Points:0","Points:1","Points:2"
5.08012e-16,-3.83383e-18,-2.26019e-21,2.54000e-02,-2.54000e-02,2.00000e-03
5.70839e-02,-4.30797e-04,-2.53946e-07,2.54000e-02,-2.51460e-02,2.00000e-03
1.14168e-01,-8.61594e-04,-5.07845e-07,2.54000e-02,-2.48920e-02,2.00000e-03
1.68975e-01,-1.13264e-03,-7.54639e-07,2.54000e-02,-2.46380e-02,2.00000e-03
1.53013e-01,3.56232e-03,-7.81987e-07,2.54000e-02,-2.43840e-02,2.00000e-03
1.37050e-01,8.25729e-03,-8.09317e-07,2.54000e-02,-2.41300e-02,2.00000e-03
1.21088e-01,1.29523e-02,-8.36630e-07,2.54000e-02,-2.38760e-02,2.00000e-03
...
...
```

It can be used to make a plot with any plotting software, as [matplotlib](#) for python, [gnuplot](#) or [Veusz](#).

5.2. Pressure average in inlet

Maintaining this flow requires a difference of energy (total pressure) between inlet and outlet. To estimate this difference, first the total pressure of the flow has to be computed. This can be done with the filter [Calculator](#) in ParaView. Note that in terms of kinematic pressure, total pressure is defined as

$$p_0 = p + \frac{1}{2}v^2 \quad (3)$$

Acknowledgments: The author acknowledges the assistance of [DeepSeek](#), an AI assistant created by DeepSeek Company, for providing valuable support in structuring this document, formatting LaTeX code, and developing the tutorial content.

Disclaimer: All content has been reviewed, modified, and validated by the author. The author takes full responsibility for the accuracy, completeness, and educational quality of this material.