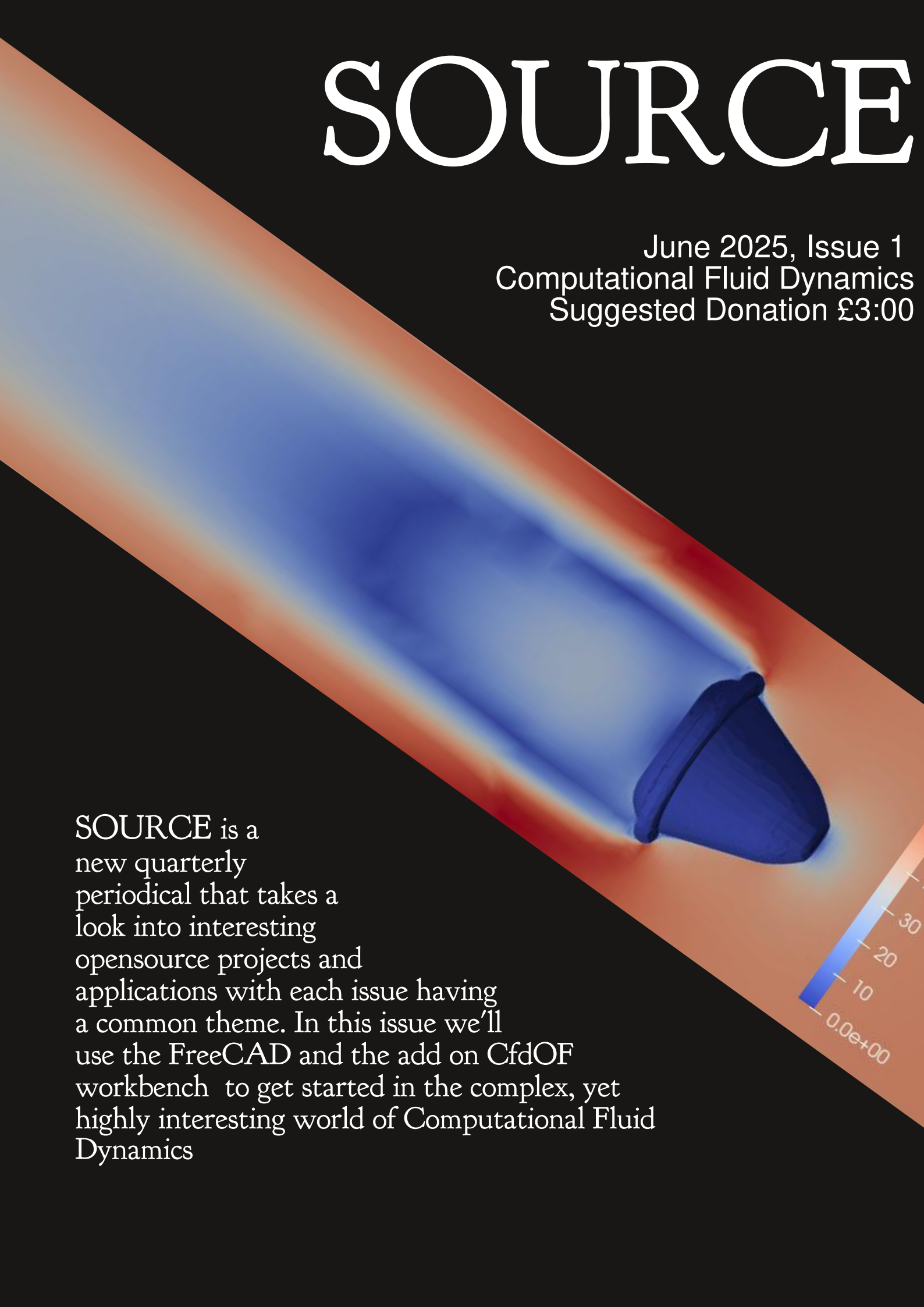


SOURCE

June 2025, Issue 1
Computational Fluid Dynamics
Suggested Donation £3:00

SOURCE is a new quarterly periodical that takes a look into interesting opensource projects and applications with each issue having a common theme. In this issue we'll use the FreeCAD and the add on CfdOF workbench to get started in the complex, yet highly interesting world of Computational Fluid Dynamics





Welcome to SOURCE



Hello, I'm Jo, AKA Concretedog, and first of all, thank you for taking interest in SOURCE.

I've been writing about technology, maker/hardware hacker/tinkerer culture for a while now but I often have stories, ideas or longer form tutorials that don't fit easily in any of my clients platforms.

So SOURCE is an attempt to fill this gap. The idea so far is for quarterly issues on a suggested donation or "pay what you feel" model. No subscriptions. It's going to focus on Opensource projects and technologies, be it hardware, software or possibly other areas of open culture, open governance and more. We'll see!

In this first issue we'll take a medium dive into the complex world of Computational Fluid Dynamics (CFD), learning just about enough to be dangerous. This issue is an experiment and my CFD knowledge is self taught in the last few months. It's also my first time attempting page layout, so things will get better as I learn Scribus. I'd love feedback and thoughts. Come and find me on the fediverse, I am @concretedog on mastodon.

[Donate via Paypal here](#)

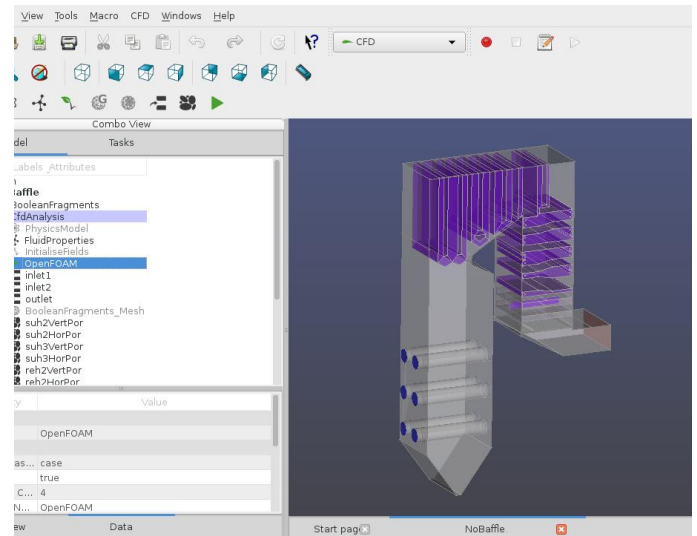
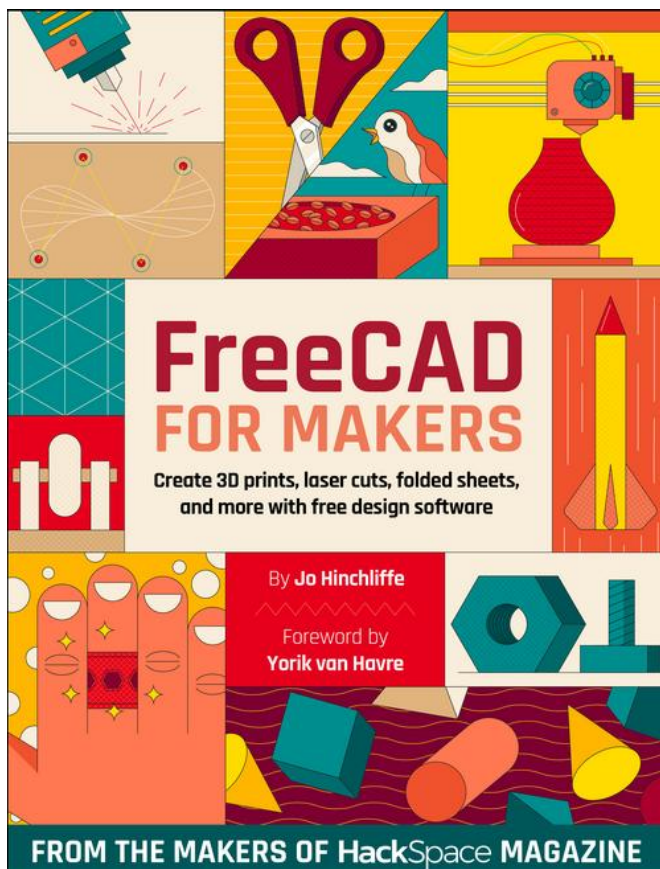
[Or Via KoFi here](#)

The Project files for this issue, and the source files for this issue are available

Getting Started, Installing the CfdOF Workbench in FreeCAD

Computational Fluid Dynamics (CFD) is complex! You should view this tutorial as a great first few steps on a very long and complex path, but do be assured, it's a fabulous and interesting journey.

CFD, whether you are using opensource or closed source proprietary systems, is usually performed with a collection of tools and techniques rather than just one piece of software. Wonderfully, the free and opensource FreeCAD has an add-on called the CfdOF workbench. Once installed and configured this has all these tools and creates a streamlined GUI based approach to CFD. Rather than re-invent the wheel there has recently been a [post on the FreeCAD News blog](#) detailing the installation of the CfdOF workbench, so your first port of call is to go and read that and get your CFD environment set up.

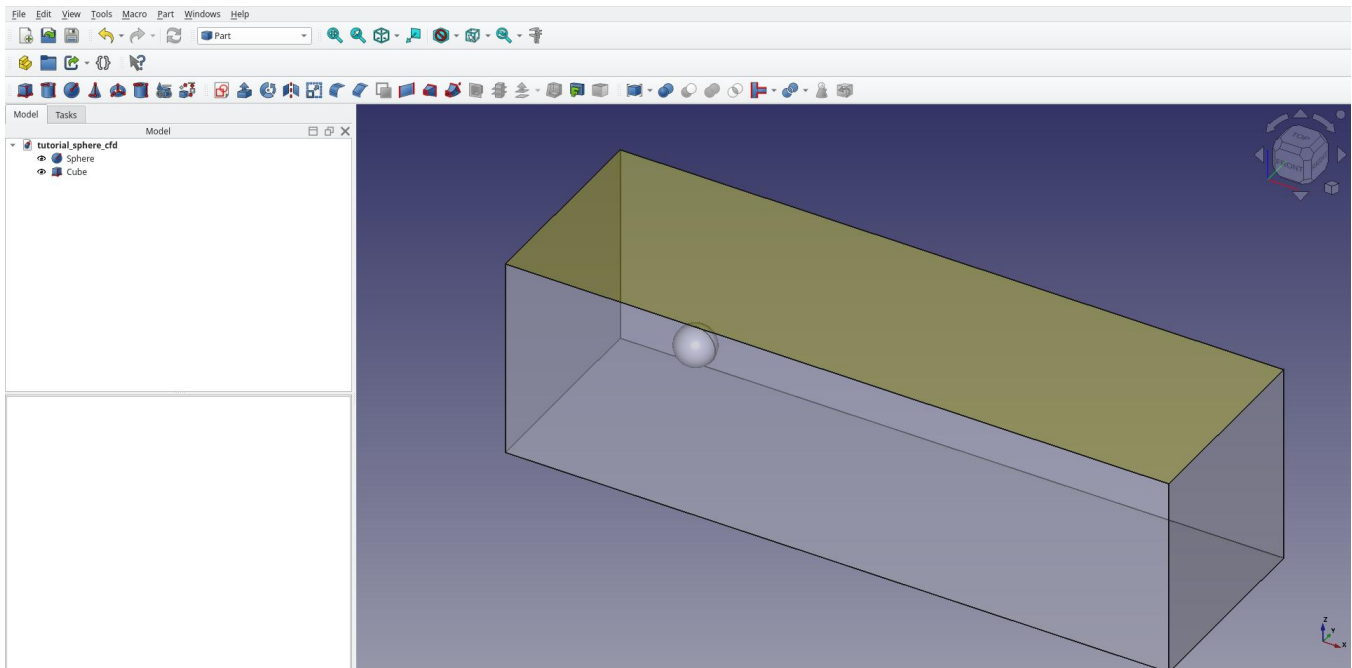


If you havent explored FreeCAD then obviously you'll need to install that, but you probably don't need to be as accomplished a CAD engineer as you might imagine to work through the content in this issue. We'll spend very little time making some very simple CAD models (literally a sphere and some cuboids) so you really don't need to know very much.

If you are in the absolute beginner position with FreeCAD then there is a free to download PDF book by the author available over on the [Raspberry Pi Press](#) but if you prefer to work from video tutorials then there are plenty of getting started videos out there in the community. Often MangoJelly and JokoEngineering are given as good examples of Youtube accounts with excellent FreeCAD content. That said, all you need to know for the CFD content in this issue is how to create, size and move items on the Part workbench and how to add external workbenches from the Addon manager.

So, when you are ready, FreeCAD is installed and the CfdOF workbench is installed with its pre-requisites, turn the page and let's get started with CFD!

Setting Up the Base Object and The Computational Domain



Starting in a new project we'll begin by modelling the target object for our computational fluid dynamic analysis. This of course could be anything, but let's start simple! We'll create a sphere and we'll insert it into a fluid domain, which is a fancy way of saying we'll put it inside a defined area filled with some kind of fluid. In our case a large volume we can think of as a virtual wind tunnel.

In this tutorial we will name tool icon's using the Tool Tip names that appear when you hover the mouse pointer over them. Which is a great way to explore FreeCAD.

In the Part workbench click the Sphere tool icon and create a sphere that is 60mm diameter. Next let's create a large cube. Click the cube tool icon and then in the cube dialogue change the length, width, depth dimensions to 1000mm, 300mm, 300mm. Right click on the Cube item in the file tree and then use "toggle transparency" to set the cube to transparent.

Next lets either use the "transform" function or the position input boxes in the cube dialogue to move the cube relative to the sphere. Let's move the cube 200mm in the X direction. Then move it 150mm in the Y direction and 150 in the Z direction. This means our sphere is central in the YZ axis relative to the cube with most of the cube on the positive X side of the sphere. This should look like the above image..

With our 2 objects created we are now going to create a "cut" item of the two objects removing the sphere from the cube. Click the cube item first in the tree view, then hold the control key and click to select the sphere. Then click the "Cut" tool. This will create a Cut item in the file tree.

In the coming steps this Cut object will be our main computational domain. We will create a mesh version of this geometry and it will be on points in this mesh that the analysis occurs. Obviously the finer the mesh then the better the analysis, but also the finer the mesh, the more points and more data there are. CFD analysis is often a compromise balanced between the granularity of data needed, and the available computational power and time. A common approach that helps to make this balance a little more beneficial is adding mesh refinements.

For now let's create a cube for our mesh refinement. Click the cube icon and in the cube dialogue create a cuboid that is 250mm on each axis and move this cube to be centred around the zero point around our sphere.

Finally, before we move onto our next stages it can make it a little easier if we rename the items in our file tree. Let's rename the Cut item to "sphere_in_main_cube" and rename our smaller Cube001 as "mesh_refinement_cube".

Let's get Meshy!

Moving to the CfdOF workbench we now need to set up our CFD analysis container. As an aside if you are familiar with the Path workbench this is similar to the idea of setting up a Path Job where all the CAM information is placed. To set up a CFD analysis container simply click the "Analysis container" tool icon.

You should now see a "CfdAnalysis" item appear in the file tree with a collection of items nested under it. To begin setting up the mesh of our fluid domain select the "sphere_in_main_cube" item and then click the "CFD mesh" icon. In the resulting dialogue ensure that the mesh utility is set to "cfmesh" and then set the base element size. This is the coarsest mesh size definition and our refinements will be smaller fractions of it. We set this to 10mm. Click "Close". You should now see a mesh item has appeared inside our Cfd analysis container named "Cute_Mesh". Highlight this object and then click the "Mesh refinement" icon. In the dialogue set the type of refinement as "volume refinement" and set the relative element size. We went with 0.5, so this means that the mesh elements within this area will be half the size of the main element, we set the main element earlier to 10mm so these will be 5mm. Note that we aren't going for a particularly hi resolution analysis, we are just going for something that will process fairly quickly. Then in the reference section of the dialogue use the select object dropdown menu to select the "mesh_refinement_cube" object we created, this will add an item "solid1" in the selection window with a tick box, tick this tickbox. Then click OK.

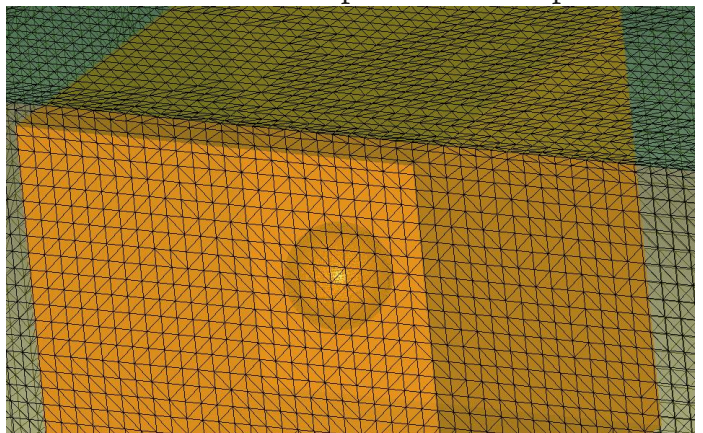
For our final mesh refinement, which will be

around the surface of our sphere, again highlight the "Cut_Mesh" mesh item placed within the Cfd analysis container. Then once again click the "mesh refinement" tool icon. In the dialogue set the type of refinement this time to "surface" and a relative element size to 0.2 again meaning that with a base element size value of 10mm the element sizes for this surface mesh will be 2mm. Again, all these values are just for this test case, in a real Cfd analysis you may need to be far more rigorous.

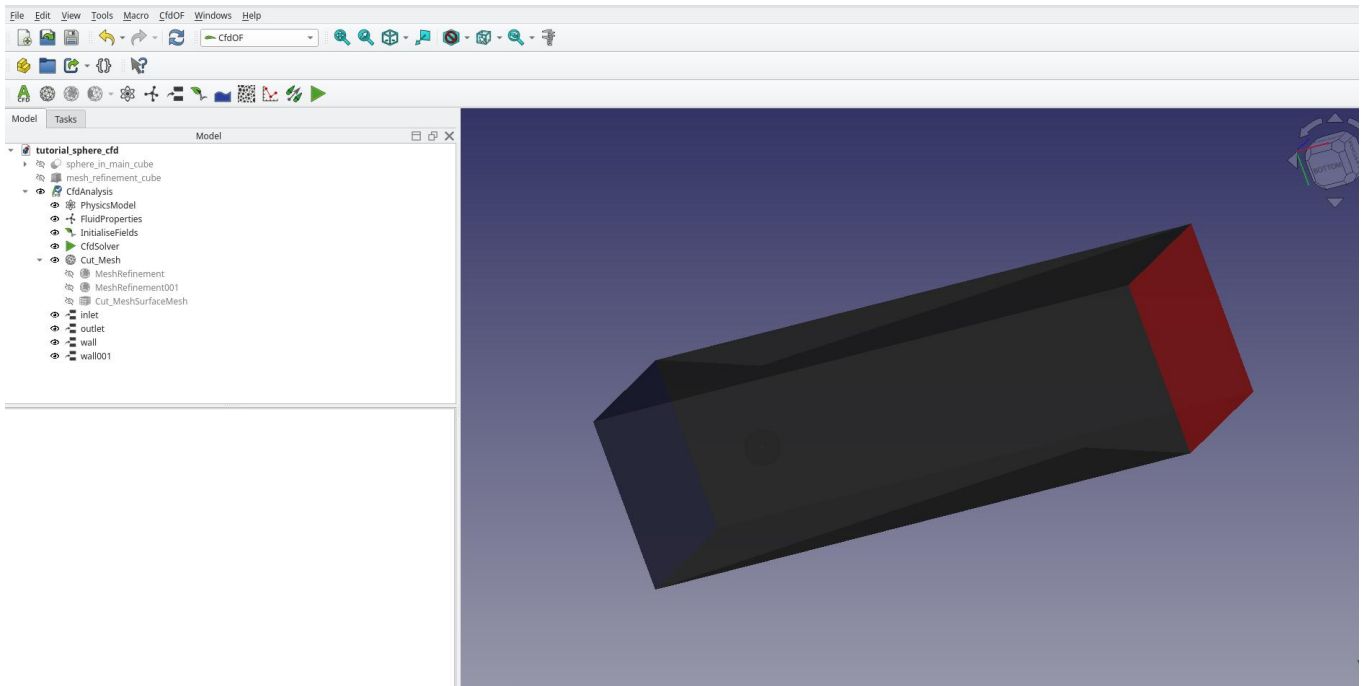
Next click the "boundary layers" item and set the number of layers to "5" and leave all other settings as their default values. We finally need to specify the sphere surface. Scrolling down the dialogue you can use the "select from list" option by first selecting the "sphere_in_main_cube" item and in the "Select components" section you can see all the faces, the first 6 faces listed will be the faces of the cuboid so we can deselect those and leave only the surface of the sphere (face 7) selected. We can click the OK button to then close the mesh refinement dialogue.

Take a moment to save your project!

The mesh is now well defined and, although we'll need to repeat this later in the process, we can test the mesh is working. Highlight the "Cut_Mesh" object inside the Cfd analysis container and in the dialogue first click "write mesh case" and when that process has completed you can click the "run mesher" button. The mesh creation may take a little time. Once completed you can click the "load surface mesh" button to load the mesh item in the preview window. With some exploration you should be able to see the different mesh element sizes in different sections of the mesh, not you can right click and set the surface mesh to transparent which helps.



Setting up the Boundary Conditions



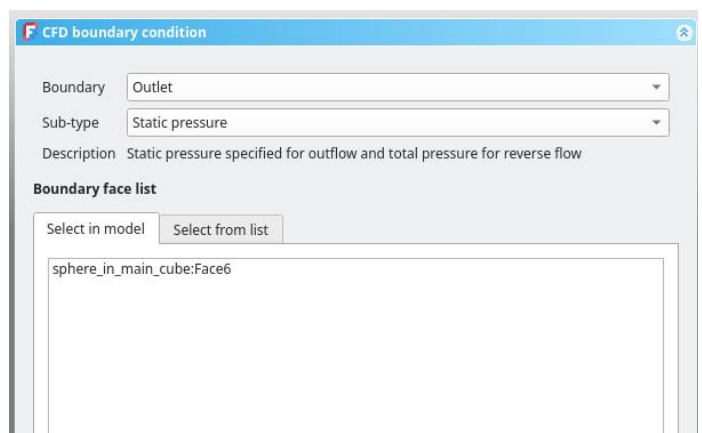
The next phase of this Cfd tutorial is to define the boundary conditions. In this particular case it is essentially defining which parts of the geometry form the walls, the inlets and outlets for our virtual wind tunnel. Additionally we'll also need to define the type of boundary our test sphere will have.

In this section we'll work primarily using the original geometry items to select the reference surfaces/boundaries, not the mesh items we created in the previous section. To make this easier it's a good idea to highlight the mesh items inside the Cfd analysis container and then use the space bar to toggle them to invisible.

To begin, in the preview window let's click to select the left hand end of the large cube that contains our sphere. This is the end that the cube is nearest to. We'll define this boundary as an inlet. Do this by clicking the "Fluid boundary." icon. In the dialogue select "Inlet" with the sub selections "Uniform Velocity" with velocity set as "Cartesian Co-ordinates" (is this ?components?). Before closing this dialogue we can set the velocity of the fluid flow, for this example we will set our flow in the Ux box and

give it a positive value as the fluid will move in the positive direction along the X axis. We will later define the fluid as air, but for now we can say, for whatever fluid is travelling through this inlet, we'll set the velocity at 33m/s. Once defined click OK to finish defining the inlet boundary.

Next let's set a second boundary condition as an outlet. For our wind tunnel you can probably guess that this will be the opposite end of the cuboid from our Inlet. Select that face of the cube and again click the "Fluid boundary" icon. In the dialogue define the boundary as an "outlet" the sub type as "static pressure" Then click OK.

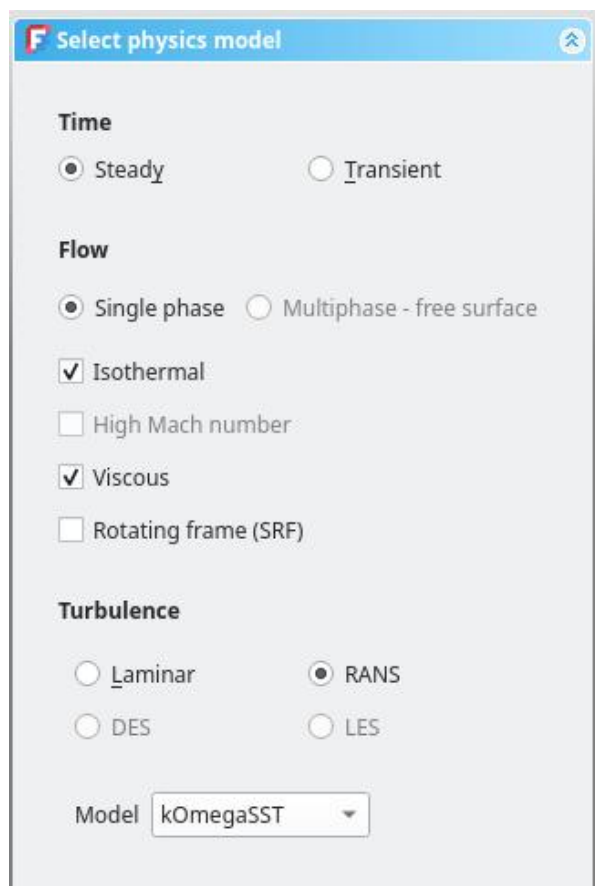


Next let's set the boundary conditions for the remaining 4 surface of our cube. Hold the control key whilst selecting the four faces of the cube in the preview window and then once again click the "Fluid boundary" tool icon. We can then simply set the boundary as "wall" and set the sub-type to "slip". Click OK to close the dialogue.

Remember, we are only scratching the surface in this tutorial. Almost every term we use can lead to hours of research and learning in a CFD context. Notably different boundary conditions, walls that are "slip" or "no-slip" for example are a large subject worthy of further reading

Our final boundary conditions that we need to set up are for the test object itself, our sphere. To do this it is potentially easier to select "Face 7" of the sphere within our "sphere_in_main_cube" cut object using the selection menu inside the "Fluid boundary dialogue". To do this, click the "Fluid boundary" tool icon with no geometry selected. In the resulting dialogue set the boundary to "wall" with the sub-type set as "No-slip (viscous)". Use the "Select from list" and then the "Select Components" panels to select face 7 of the sphere_in_main_cube object.

Setting up the Physics Model



To set up the Physics Model double click on the Physics model item in the Cfd Analysis container. In the "Select physics model" dialogue that opens create the following settings. Set the "Time" to "Steady" and the "Flow" to "Single Phase" and check the "Isothermal" and the "Viscous" tickboxes.

In the "Turbulence" sections select "RANS" with the "Model" dropdown menu set to "kOmegaSST". Everything below the "Turbulence" section can be left at the default settings. Click OK.

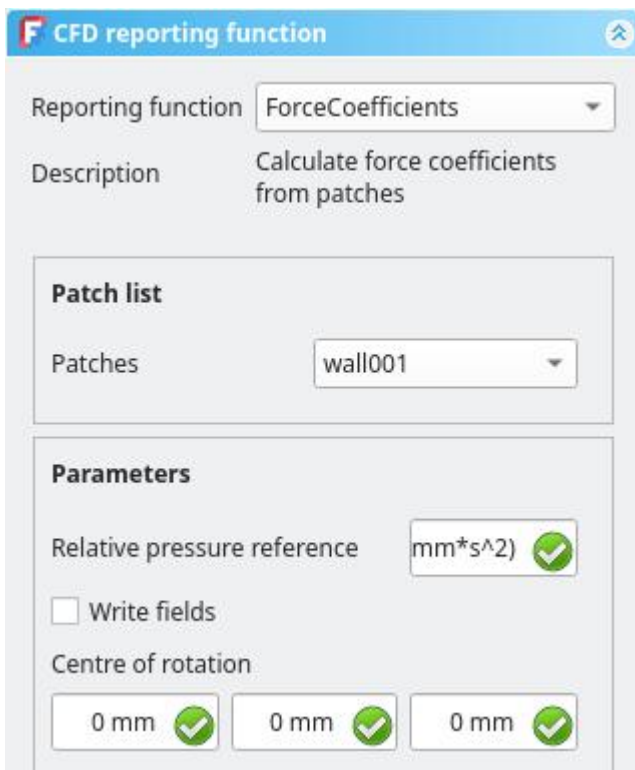
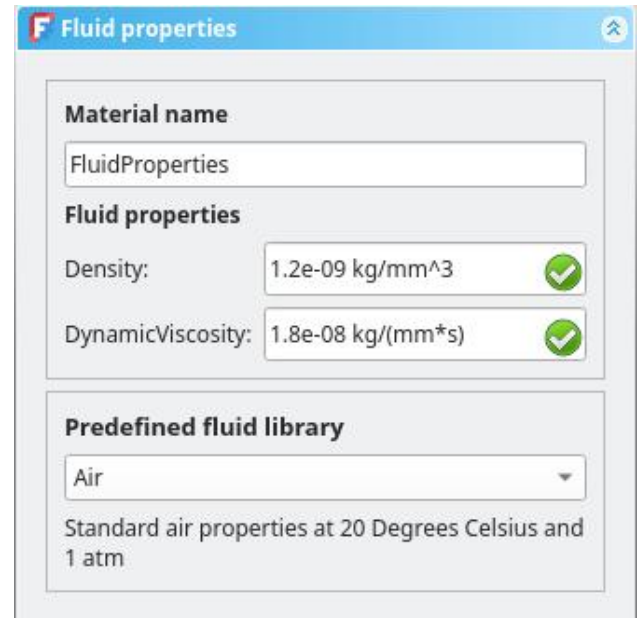
RANS stands for Reynolds Averaged Navier Stokes equations and is just one type of turbulence model used in Cfd. As turbulence occurs in most situations where we might perform Cfd analysis it's a fascinating area to know a little more about. A useful primer for RANS and other turbulence models can be found at.

<https://www.idealsimulations.com/resources/turbulence-models-in-cfd/>

Set Fluid Properties and Initialise the Fields, optionally adding a Reporting Function

Back in the Cfd analysis container the item below the physics models is "FluidProperties". This is relatively intuitive, double click on the "FluidProperties" item and in the Fluid Properties dialogue select "Air" from the "predefined fluids library". Notice that, if you know the Density and Dynamic Viscosity values, you can define any fluid you like in this dialogue and name it. To use this function select the "Custom" option from the "predefined fluid library" dropdown menu. With "Air" selected click OK to close the dialogue.

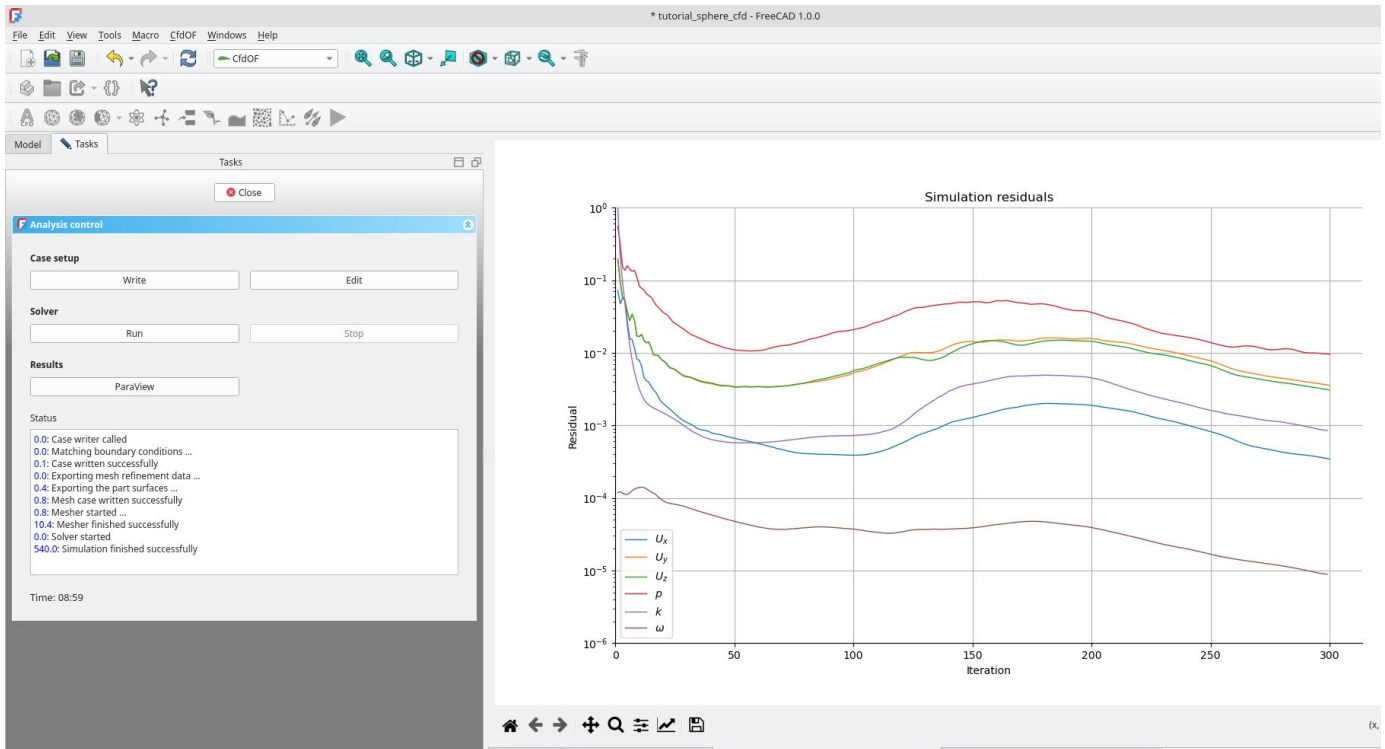
Below the fluid properties item in the tree view you'll find "InitialiseFields". Double click to launch the "Initialise flow fields" dialogue. For our example you probably won't need to change anything but double check that the "Velocity" and "Pressure" options are both set to "Potential Flow". Click OK.



We are now at a point where we could run the solver to perform the analysis should we choose too, however the FreeCAD CdFOF workbench has additional tools to allow us to receive more data from our simulation. We can use the "Reporting function" tool icon to add an additional report to the analysis. As our analysis is emulating a wind tunnel we can add a reporting function to show the drag and lift forces for example.

To do this click the "Reporting function" tool item and you will see a "CFD Reporting Function" dialogue. Towards the top of the dialogue you'll see an area called "Patch List" with a dropdown menu. The dropdown menu has a list of all the boundaries we defined in the project earlier. Select the boundary item that is the surface of our sphere "wall001". At the very top of the dialogue you are asked to choose which function you would like to report on. Let's choose "Force Co-efficients". We can, for this analysis leave all other settings in the dialogue as their defaults.

Running the CFD Analysis Solver



Excitingly we are now at the point where we run the CFD solver! First let's double click on the main mesh item in the analysis container, re write the mesh case and re-run the mesh case to make everything up to date. If you omit this step when you start the actual solver it will probably prompt you to re-mesh at this point.

Having remeshed next let's single click to highlight the "CfdSolver" object in the file tree. Below the file tree view you will see the parameters pertaining to the solver. Under the sub heading "Iteration control" you'll see that you can set a numerical value for the "Max Iterations". Let's set this value to 300. Another approach to this can be to allow the analysis to run until the results "converge" this is again a large subject within CFD and so for now let's ignore it but a suggested area for further research is to look at the terms "convergence" and "residuals" in CFD.

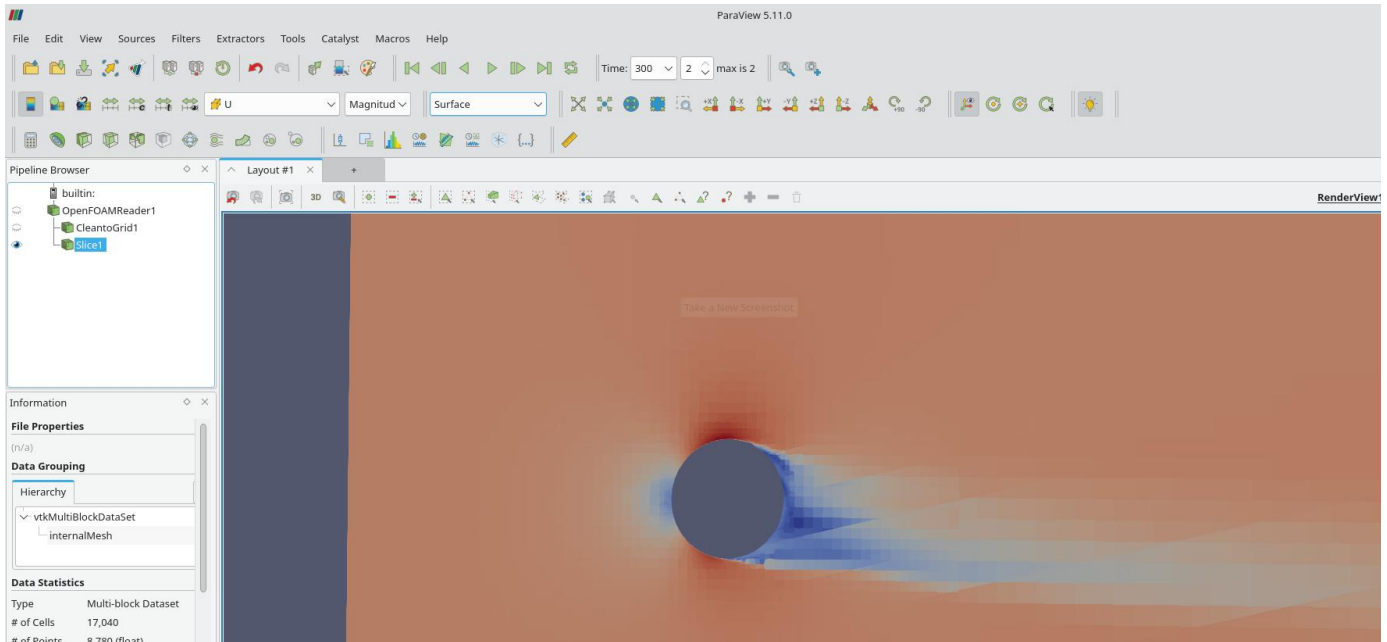
Another item you can set is under the sub heading "Solver" and is "Parallel Cores". This sets the amount of cores within your computer the solver will request to use to perform the analysis. If you

have this set too high then you may well see an error message that begins "There are not enough slots available in the system to satisfy the 4 slots that were requested by the application:" if you do, reduce this value. This example will run happily in a few minutes with only 4 cores being requested on a pretty old i7 laptop. If reduced to 2 cores it takes around 11 minutes. If you have a nice 16 core machine you are golden!

With those parameters adjusted we can now double click on the "CfdSolver" item in the file tree and in the resulting "Analysis control" dialogue we can first click the "write" button under the "Case setup" heading and then we can click the "run" button under the "Solver" sub heading.

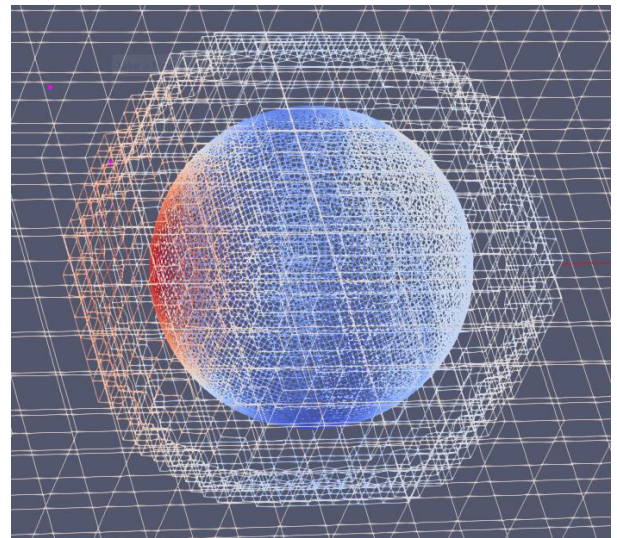
Hopefully you will now see a message that the solve has started and after a few seconds two new preview windows will open, one showing the progress chart of simulation residuals and the other showing the report function we requested so will have coefficients for drag and lift at each iteration point.

Post Processing in Paraview



When you set up the CfdOF workbench as well as the various meshing systems and the OpenFOAM application you also installed ParaView. ParaView is a data visualisation application that also can help you to analyse the data you have created in your CFD simulations. Again, it's fair to say that ParaView is complex and a complete set of instructions would be book like in length. As a very mere sample of what we can do though once our simulation is complete click the "ParaView" button in the "Results" section of the "Analysis control" dialogue we just used to run the solver.

You should see ParaView launch as a separate application with our simulation case data loaded in. Use representation dropdown to click things like "wireframe" to see through the mesh surface. Another useful tool to explore is the "slice" tool which allows you to slice a flat plane through the computational domain to get cross section views of your test object. Using the XXXX dropdown menu it's trivial to switch between "U" and "p" with "U" being the velocity of air, and "p" being pressure shown as a magnitude gradient within the computed domain.



As you can probably tell, similar to this whole process, Paraview and post processing of CFD data is worthy of hundreds of pages of tutorials in its own right. There are heaps of resources out there online but a decent set of Paraview tutorials can be found on the "Premier Aerodynamics" youtube channel linked below.

<https://www.youtube.com/watch?v=IM6oOK4QeR4>

Thanks for Reading!

Thanks so much for reading if you made it this far. I hope you've found some use or inspiration from the content. I have ideas for the next couple of issues and I'm imagining they will be slightly larger and may include some other content as well as a single focus tutorial similar to this one. One idea is instead of ever having adverts, we might have "subverts" the subvert being I'll pick a cool opensource project and make a small advert for it, however the project won't pay for the "subvert" in fact I'll make a small donation to the opensource project taken from donations to SOURCE.

Just one of many ideas!

Jo AKA Concretedog.

[Donate via Paypal here](#)

[Or Via KoFi here](#)