STAR-CCM+ INTRO

by Saaim :D

Introduction

Welcome! This document was made for the UoL AeroLabs team, especially those that are in the CFD team alongside me! This is a guide and summary of everything I covered in the workshop on the 29th.

So, you can just use this if you missed it! A couple of things I wanted to say before you start:

1. STAR-CCM+ is an industrial grade CFD software, used by huge companies like Boeing, JLR, Siemens, Ford and a lot of F1 teams!
2. Since its industrial grade, it means that there's a learning curve to using the software, so don't be upset if something doesn't work, it is probably because of your mesh (we will go into this later).
3. I still learn things every day when it comes to CFD, so I will be exposed to new knowledge just like you all! The main aim is to get as many quality runs as we can to see what works, and what doesn't, so we can do our best to win the competition.
4. Anyone that has no idea what CFD is or what it does, have a look at this website here: https://airshaper.com/#sample-simulations
    Scroll down and you will see a few sample simulations that they have.
    Play around and see how it works, change the position of airflow and you can begin to understand how air moves at certain parts of an object.

I hope this doc helps even if it's a little bit.

- Saaim (CFD Team Lead)

Downloading

As of writing this, STAR-CCM+ should already be installed on every computer at uni, you need a POD license key to run it, which should already be working fine for the computers at the university.

To use the software at home, you would need to ask a professor for it, I linked his email below.

This key is updated every May, so just be wary of that. Also, don't share the license outside of the uni.

If you are a nerd (that's a joke) and want to download it at home, you can find it here: https://myfiles.le.ac.uk/myfiles

The instructions are all there, but you do need some storage available for it.

If you have any issues, you can email Dr James Jewkes -> jwj5@leicester.ac.uk
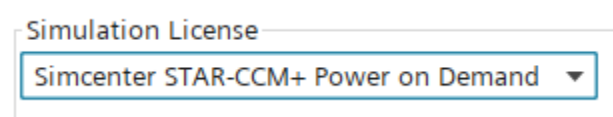
Geometry

Before you even start a run in STAR-CCM+, you would need some sort of CAD model, whether that is in SolidWorks, ANSYS, Siemens NX or even Blender.

Before you export your model, please make sure that it is a watertight solid, basically meaning that it shouldn't have any tiny gaps, fillets, or edges, miniscule details that won't really affect aerodynamics, but it would make the software have a seizure when trying to process it.

When exporting, try and make it a Parasolid or STEP file, but STL works fine too. Sometimes the software won't give you an option to export it as these files, which is okay, that's mostly because it's a student version so not everything is accessible.

Starting off

Alright, you loaded up STAR-CCM, great, now the first thing to do is check the top left corner, press file -> new. Then make sure the simulation license is set to Power on Demand if it isn't already.

Simulation License

Simcenter STAR-CCM+ Power on Demand ▼

Look on the left side, press geometry -> 3D-CAD Models, and then at the top left, press

file -> import -> import surface mesh.

Then you can select your units if you want, ignore everything else.

Press "OK" when you are done!

Fluid Domains

We need to make the domain where our simulation takes place, basically a box to trap our airflow in!

Under geometry, right click on "parts" -> new shape parts -> block

Then increase the size of the block to whatever you desire! Preferably, it should be 5-10 times the length/width/height of your model by ALL sides. It ensures that the boundary conditions don't interfere with the flow! In English, it means that the walls won't stop the air from giving valid results.

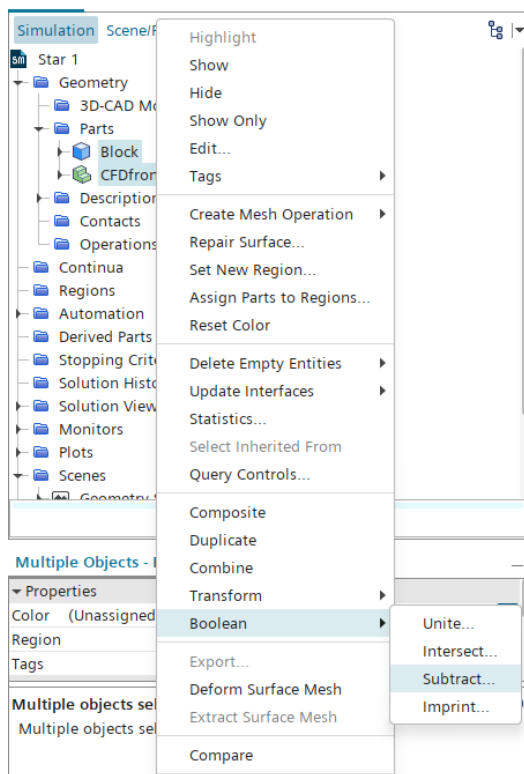It should look something like this!

If you have a shape that is symmetrical, then you could just make the box half of it, since the results will be the same for both
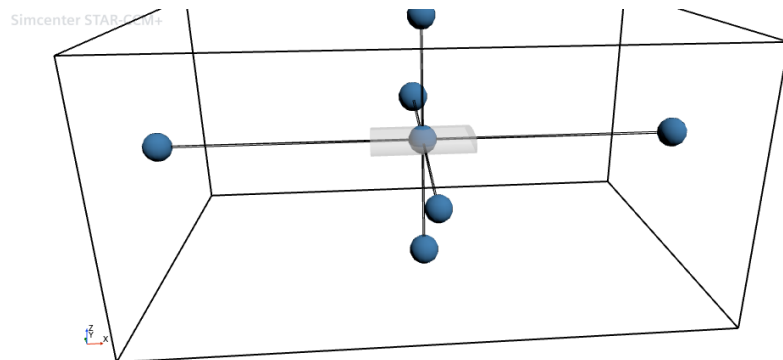
sides!
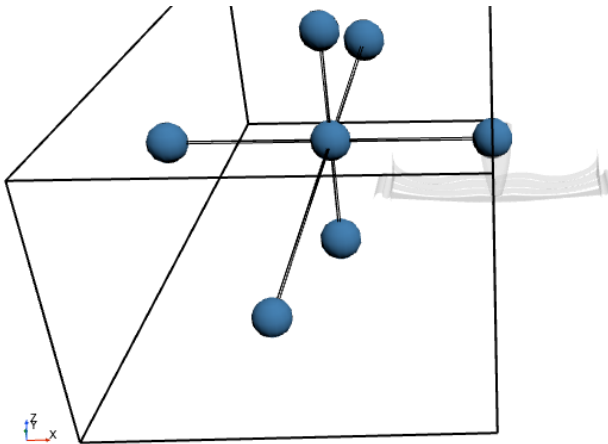
Now after you are done with making the block, click "create" then "close"

The next step is to subtract the solid from the domain. This is done to simplify the simulation, and it creates a watertight domain that is fully closed.

Click the block, hold CTRL and click your model, then right click -> Boolean -> Subtract.

We do this so that the software knows where the air can flow and where it can't!!

At this point, you may get a message saying that your model is "not closed" or "manifold". This means that your model is either not watertight, aka it has tiny gaps, or has overlapping faces. So, try and fix those back in SolidWorks.

Go to the subtract you just made -> surfaces -> right click block surface and choose to split by patch.

Click on the wall, which is in front of your model, the one where you want the air to flow into, call this the "inlet" and click create. Same with the one behind your model, name this "outlet", do the same with the rest of the walls, naming them like side wall 1, bottom, roof, etc. For the last wall, just click close and rename it manually as well as the object inside (aerofoil).
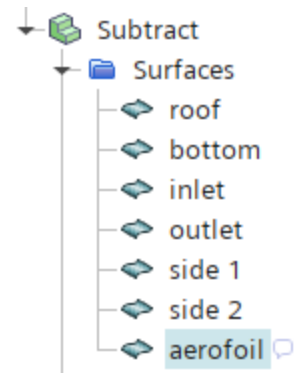
It will look like this ------>

Then, right-click on subtract on the left, then "assign parts to regions" and change the bottom left option to "create boundary for each part surface" and click apply.
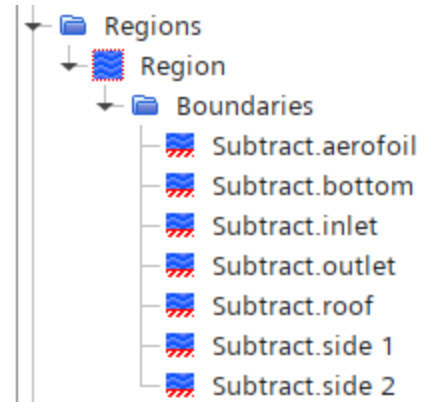
It looks like this!
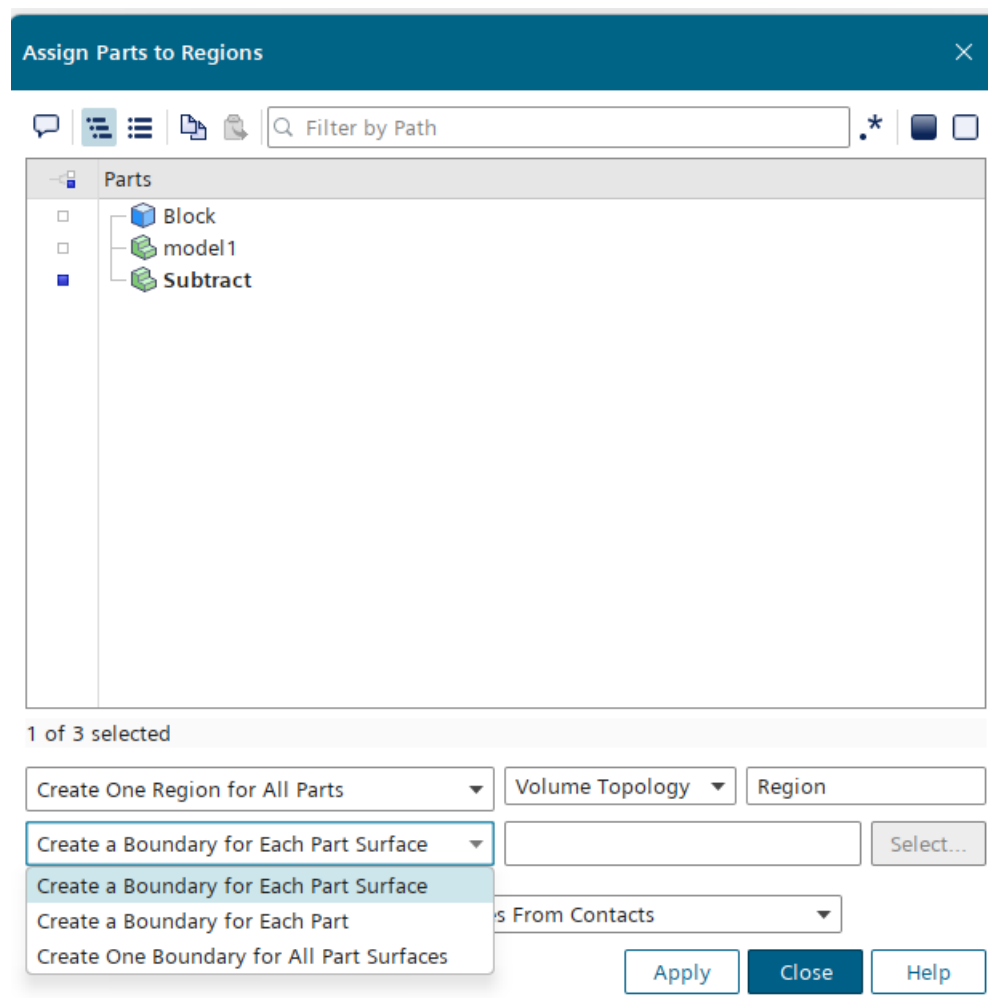
To know that you did this step right, you would be able to see a region that was created.

It should look like this -------->

btw, well done for getting up to this point, the number of tears I had to get here was too much :D

Adding physics properties!

On the left side, right click on "continua" -> new -> physics continuum (whatever that means)
Click physics -> models

From here, these are the only options you need to click:
Gas, Steady, segregated flow, 3D, turbulence, k-Omega, constant density

There may be more options that we will explore in the future, but for now, this is it! Don't go changing the value of gravity or something...

Adding initial conditions

Under physics 1, go to initial conditions and change these:
Velocity - 20m/s (check the axis for the correct direction)
Pressure - 101325 (air pressure at sea level)

Now back to the left side, go to regions -> region -> boundaries -> subtract-inlet
at the bottom (in properties) scroll to find "type" and change it to velocity inlet. Do the same with the outlet and change the type to pressure outlet.
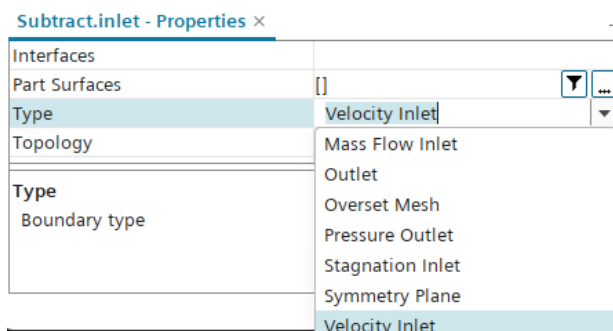
Then you need to put in the values for velocity and pressure by going into "physics values" under both subtract-inlet and subtract-outlet respectively.

Now we must make our walls have no friction, because when the UAV is flying in the air, there are no walls, I think..

Go to the each of the walls in region -> physics conditions -> shear stress specification -> slip
Repeat for the other 3 walls!
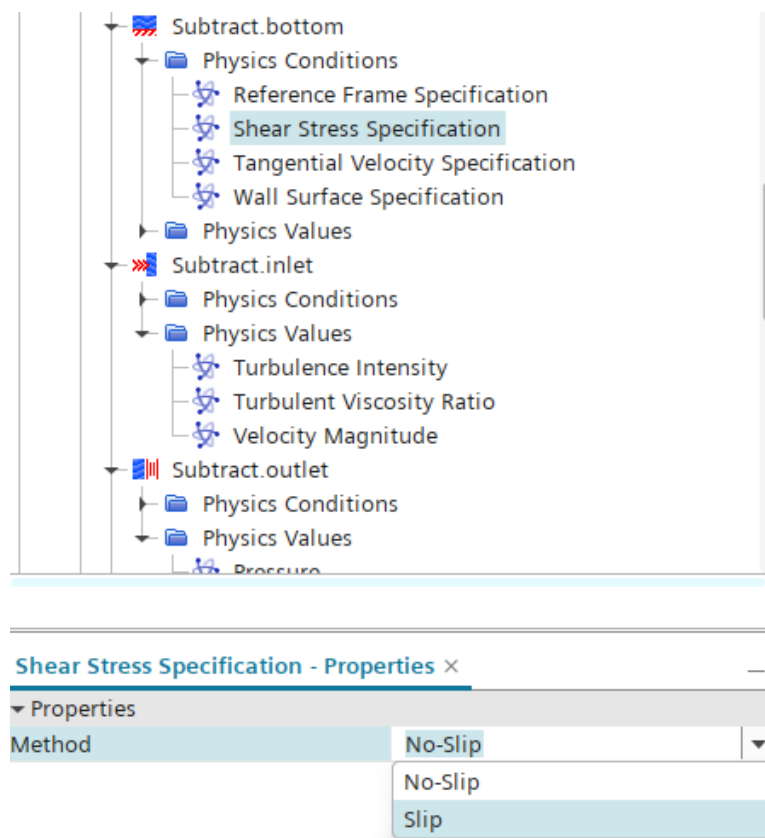
Meshing

Wow we are finally here, okay this part is a little bit long so sit tight.

On the left-hand side, right click on operations -> new -> mesh -> automated mesh
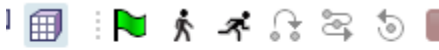
select these options:

Enabled Meshers
- ☑ Surface Remesher
- ☑ Trimmed Cell Mesher
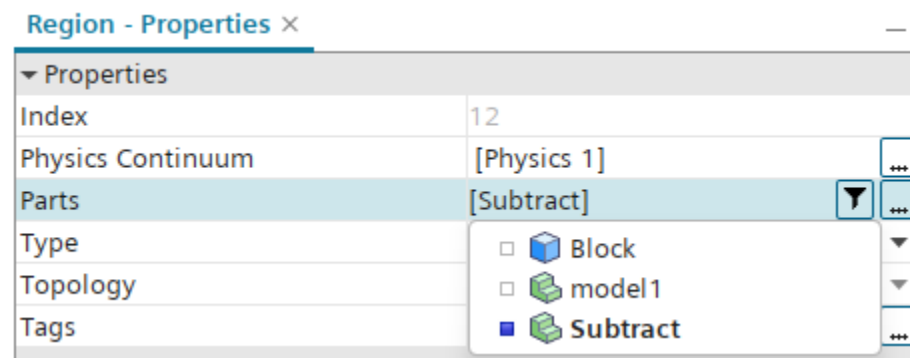- ☑ Prism Layer Mesher

Using polyhedral layer meshing is slower but more accurate, we can use that for future, more complex iterations.

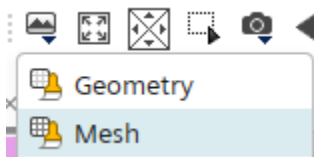Click "OK" and then execute it by pressing this button at the top:

Generate Volume Mesh

If you happen to get an error saying that the part needs to be assigned to the region, then just check that you have added the part in region, under properties:
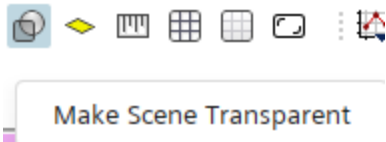
Region - Properties ✕

▼ Properties	
Index	12
Physics Continuum	[Physics 1]
Parts	[Subtract]
☐ 🧊 Block	
☐ 🟢 model1	
■ 🟢 Subtract	
Type	
Topology	
Tags	

Now let's check how good the mesh came out!!

At the top, look for this icon:

- 🖼 Geometry
- 🖼 Mesh

then press this:

Make Scene Transparent

So below we can see that our mesh... isn't looking too good...
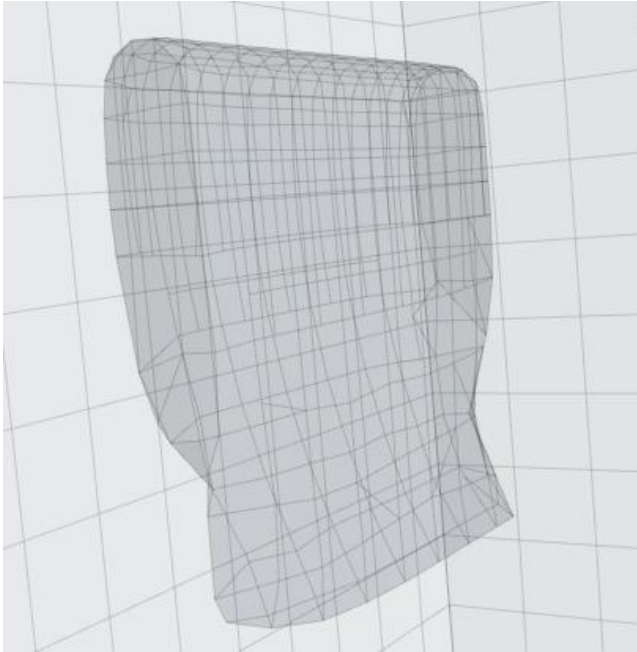
At the trailing edge, the mesh really deforms and has no structure at all.

So, let's fix that.
Go back to operations -> Automated Mesh -> default controls -> base size
Right now it should be on 1m, so change it to 0.1 or 0.2 instead. The base size is basically the size of one square in the mesh.
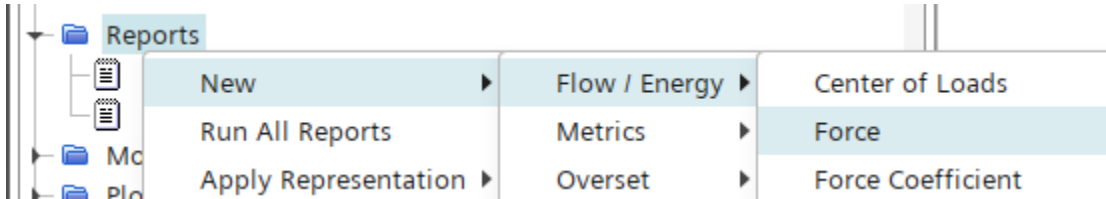
Just like that, our mesh is much more polished! This will take longer to get results BUT it's worth it!
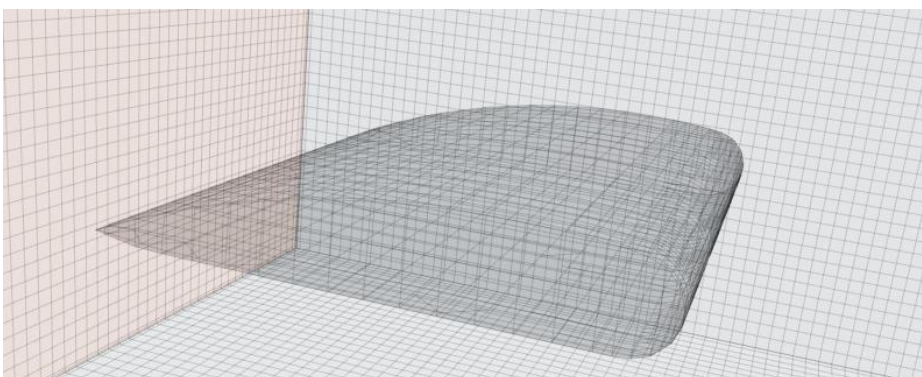

Post processing

Congratulations on finishing pre-processing! That was pretty long, now we can run the simulation and actually get good results!! This part is easier, so let's get into it!

First we need to make a report, so we can get values for lift, drag, etc.

Rename the force as "lift" and repeat it again, naming the new one as "drag".

Next, we need to state which direction these forces will be acting. So based on the axis on the bottom left, input 1 or -1 into the correct direction. Here's my example below.
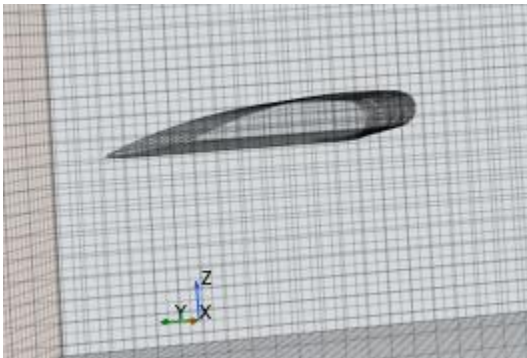

Air will flow to the y-axis, so the direction of drag will be [0.0, 1.0, 0.0], and the direction of lift will be [0.0, 0.0, 1.0] since air will go in the direction of the z-axis. If it goes in the opposite way, then use -1. Under "reference pressure" in the same area, put 101325Pa for both lift and drag.

Now go into "parts" under the same menu, select your aerofoil in both the parts and region section like below. Do this for both lift and drag.
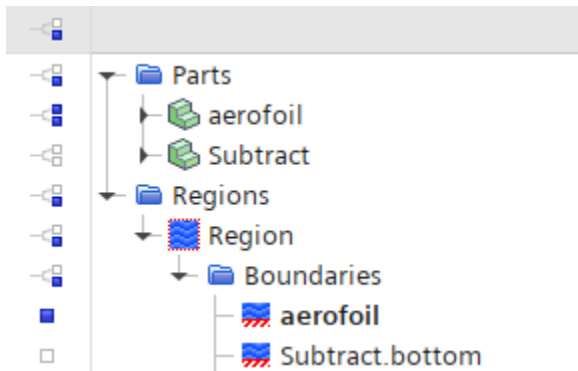
Select both the drag and lift reports, right click -> create monitor and plot from report -> single plot

Now click "run", or press CTRL+R.
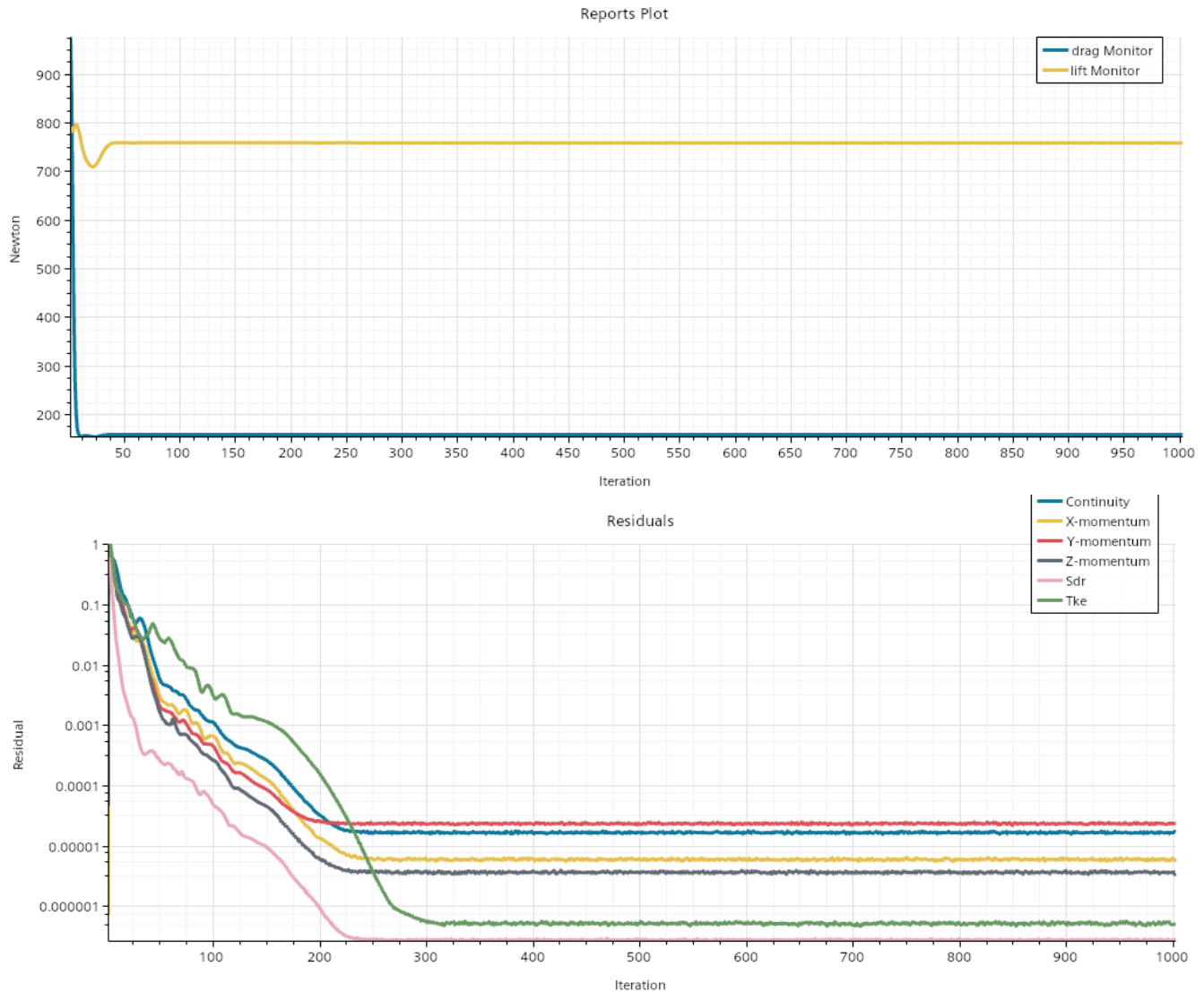
Just like that, we have some results, these flatten out and converge after hundreds of iterations, it shows that our results are reliable. Our lift is 760N and the drag is around 150N, which is unrealistic for us because the aerofoil was completely random haha. But this is the correct method!!
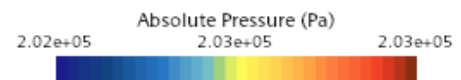
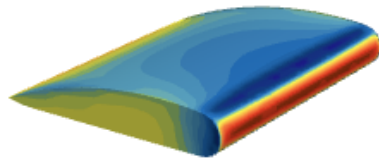Let's get some visual results now!!

Visual results!
On the left-hand side, find scenes, right click -> new -> scalar
Inside of scalar scene 1 -> scalar -> parts

Under parts in properties, select the aerofoil in the parts folder and the regions folder.

On the bottom right is says "select function", this is where we can choose what we want to see!

This is one for absolute pressure, you can also change the colours in the properties area.

But, what is CFD without some streamlines? Let's get those too!
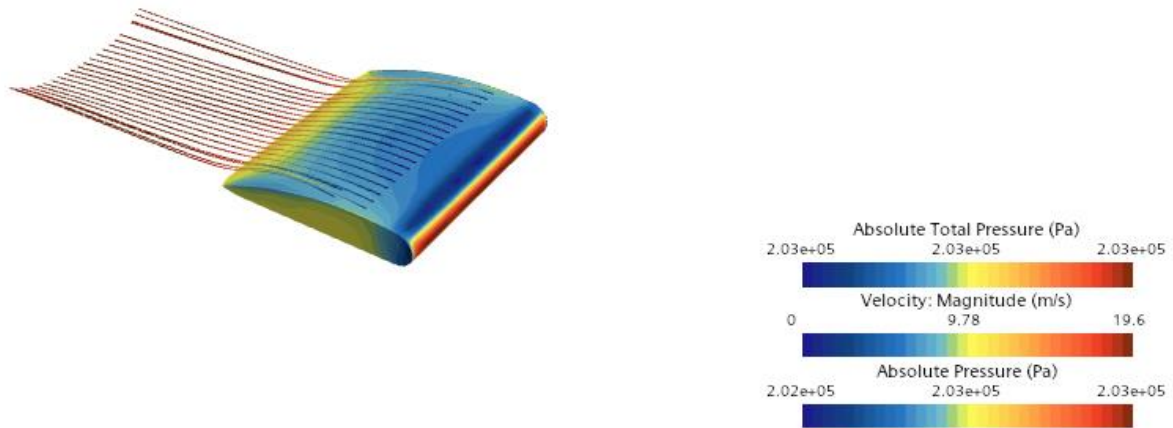
Streamlines

Go to "derived parts" on the left-hand side -> new -> streamline

Select the aerofoil in "input parts" and "seed parts" like we have done before, both in parts and the region.

You can change the resolution to get more streamlines!

Close that menu and then pick "velocity magnitude" in the "select function" box on the bottom right.

There we go! Streamlines can be seen too!

In our future simulations, we will be changing the AoA a lot (Angle of Attack), to do this, go to geometry -> parts -> right click aerofoil -> transform -> rotate -> choose desired angle.

Then remesh and run the simulation again!! My simulation was done at an angle of 0.

Final words

Congratulations for getting all the way to the end of this little guide, I really hope it made CFD a little bit easier to replicate and understand.

This isn't meant to be perfect, it's meant to get us started. Every simulation you run adds a small piece to our understanding of what's happening on the wings we design.

I'm still learning this too, and that's the point, we'll figure it out together, share what works, and build something solid as a team. Don't be afraid to experiment, break things, and ask questions. That's exactly how we improve.

Thank you for reading!

- Saaim