

An open-source project: OpenFOAM for aerodynamics

Summer 2020



Project/Course aim & layout

This project aims to develop your engineering abilities & knowledge of OpenFOAM for aerodynamics applications. This will not be a strictly taught course. Instead, participants are encouraged to learn via individually completing a project. Participants will be encouraged to develop their problem-solving skills by tackling the challenge of building an aerodynamics CFD simulation in OpenFOAM. Learning material will be provided to direct and guide the participants, but each participant will be required to build their own simulation. Learning how to overcome problems is a core attribute for CFD users and will be evident when working with OpenFOAM. Support and guidance will be provided throughout the project, when participants encounter problems and difficulties that every experienced CFD user inevitably goes through. Through this project, we hope that participants will gain experience that will benefit them in their academic or industry careers, that they can leverage to continue learning and self-developing if they wish to pursue CFD further.

Project content/timeline

Week 1:

- Installing required software
- Running a built in OpenFOAM tutorial

Week 2:

- Meshing with OpenFOAM

Week 3:

- Boundary conditions
- Initial conditions
- Running your simulation

Week 4:

- Mesh independence study

OpenFOAM: Useful links

This is a collection of sources of information on the web, that can be used for extended learning, or to elaborate on topics in more detail if participants are interested.

User guides:

- OpenFOAM user guide (CFD Direct): <https://cfd.direct/openfoam/user-guide/>
- OpenFOAM Wiki: http://openfoamwiki.net/index.php/Main_Page
- OpenFOAM user guide (ESI group): <https://www.openfoam.com/documentation/user-guide/>
 - PDF of ESI group user guide <http://foam.sourceforge.net/docs/Guides-a4/OpenFOAMUserGuide-A4.pdf>

YouTube channels:

- Josef Nagy: <https://www.youtube.com/channel/UCjdgpuxuAxH9BqheyE82Vvw>
- Calum Douglas: <https://www.youtube.com/channel/UCSnjW1aZwOOzpGOyZKhyWEw>
- OpenFOAM tutorials: <https://www.youtube.com/channel/UCHXaKLIZRQxkSG6dD6ZWvSg>
- Comflics: https://www.youtube.com/channel/UC7DgvEhEWhbsMCt_t5MpSQg
- FoamDude: <https://www.youtube.com/channel/UC-rD0nWz4tBKxPr0cOOPa7A>
- Kenneth Hoste: <https://www.youtube.com/channel/UCqCLabkRddpbHj4wYNFYjAA>
- Holtzmann CFD: https://www.youtube.com/channel/UCLMLJc3tV_tNRIWoVFasRGw

OpenFOAM: Useful links

Dedicated OpenFOAM website:

- Holtzmann CFD <https://holzmann-cfd.com/>

Useful blogs on CFD:

- Tomer's Blog: <https://allaboutcfd-tomersblog.com/>

Online courses:

- Eindhoven University, Sports and Building Aerodynamics (& CFD details, very highly recommended): <https://www.coursera.org/learn/sports-building-aerodynamics>
- Boston University, Introduction to CFD (Highly recommended): <https://www.youtube.com/watch?v=35unQgSaT88>

Other useful guides:

- Australian Government: <https://apps.dtic.mil/dtic/tr/fulltext/u2/1002391.pdf>

Week 1

Week 1: Contents

- Software installation
 - Required software to download and installation (assuming Windows operating system).
 - OpenFOAM download and install for Windows
 - Notepad ++
 - ParaView
- OpenFOAM interface
- Motorbike tutorial
 - Run a sample OpenFOAM simulation
 - View results

Week 1
Software Installation

OpenFOAM download link

OpenFOAM is a Linux based code and has only been adapted for use on a Windows operating system in recent years.

The following link is for up-to-date compilations of OpenFOAM on Cygwin, a Linux emulator that runs on Windows. The company CFDsupport provides and maintains this installation. Windows 7, 8 and 10 are supported.

<https://www.cfdsupport.com/openfoam-for-windows.html>

Instructions for download and installation are given in the next slides.

OpenFOAM download link

The link from the previous slide will direct you to the CFDsupport website. Here, you will find the download link.

The logo for CFDsupport, featuring the text 'CFDsupport' in a white, sans-serif font. Above the text are four curved arrows in red, orange, green, and blue, pointing from left to right.

OpenFOAM® for Windows

Free, natively compiled package for running OpenFOAM® in Windows

[Download OpenFOAM® for Windows here](#)

[Workflow Example OpenFOAM® for Windows here](#)

OpenFOAM download link

Prior to receiving the download file, you must first enter your details and subscribe to the website.

- Enter your details and select the tick box.
- The download button will become selectable.
- Click on the “DOWNLOAD” button.

Please subscribe before download and stay informed about news in OpenFOAM® and our company. Your information is safe. We do not like Spam. You can unsubscribe anytime.

*First Name:

*Last Name:

*Email:


*Phone Number:

*Country:

*Organization:

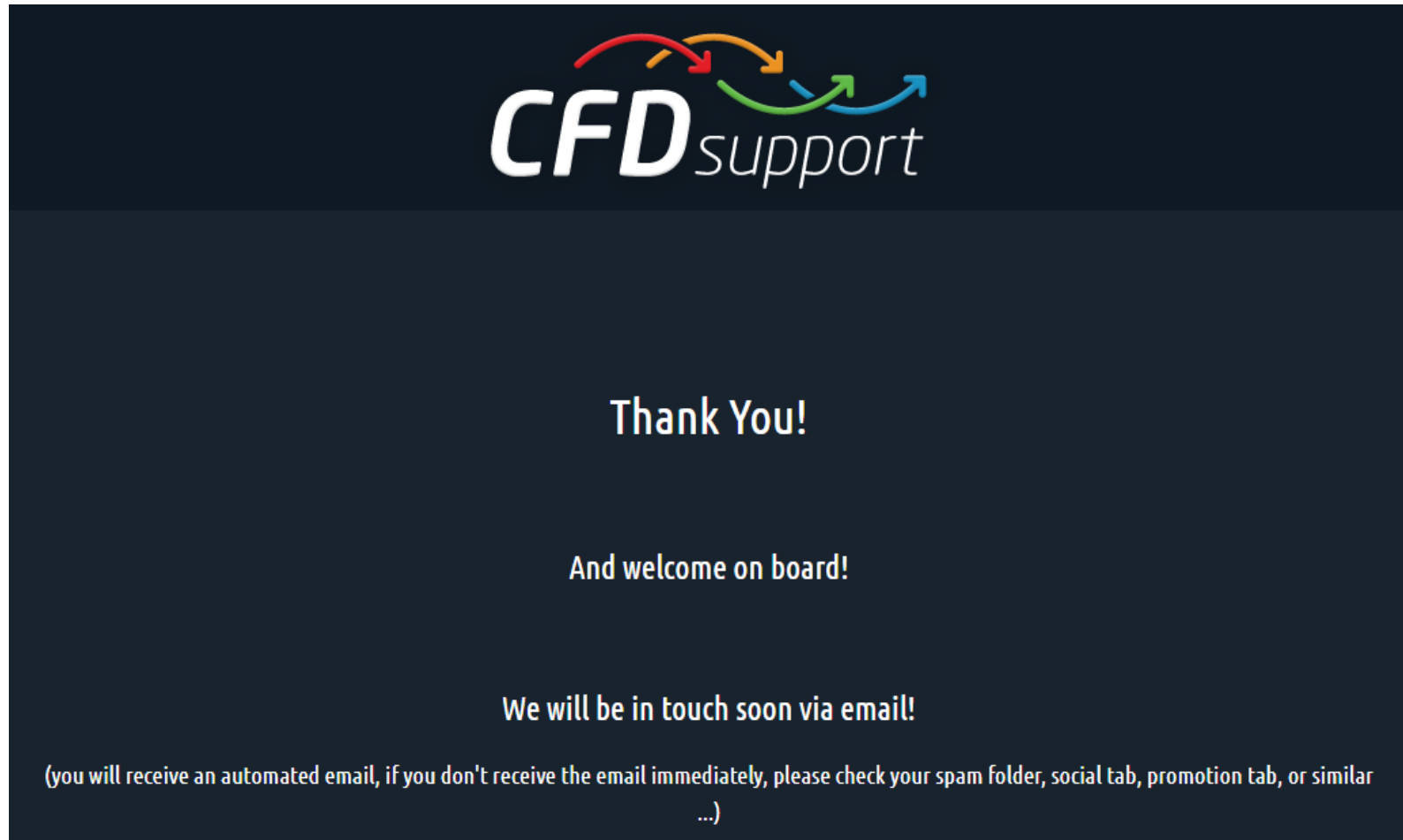
*Interest:

Yes, I have read the [Privacy & Cookie Policy](#) and agree to receive occasional email updates from CFD SUPPORT.



OpenFOAM download link

You will be sent an email with the download link. Select the link and the .exe file will be downloaded.



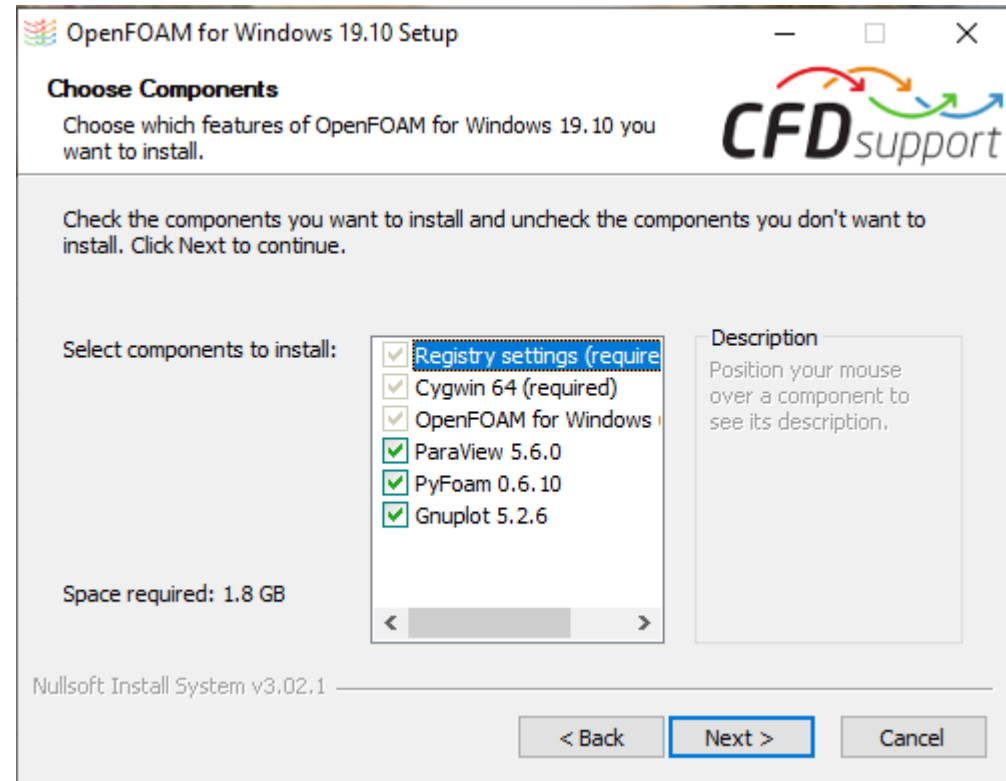
OpenFOAM download link

Double click on the .exe executable to begin the installation process.

Note: your system may require a restart during this process.

When prompted, allow for the installation of additional OpenFOAM extensions (swak4FOAM & others), and also, for additional software such as ParaView (an opensource visualization software) and GnuPlot (an opensource graphing tool).

Follow the installation instructions to complete the installation process.



OpenFOAM download link

Optional

If participants are experienced Linux users, we would encourage you download and compile OpenFOAM from the following link:

<https://openfoam.org/>

This is not a required step for this project. The previous Windows installation from CFDsupport is used for this project to allow for this project to be more accessible to all participants who may not have previous Linux experience.

Notepad++ download link

Notepad++ is an opensource text editor, that is enormously useful for preparing the files required for OpenFOAM.

<https://notepad-plus-plus.org/downloads/>

There are many other free text editors available (and some are possibly better), such as Sublime and others, so if participants are already familiar with another text editor then they are encouraged to stick with that.

ParaView v4.4 download link

ParaView is an opensource post-processing tool, designed for computational simulations. It is the standard tool for post-processing with OpenFOAM.

<https://www.paraview.org/download/> (ensure you select version v4.4)

A newer version of ParaView will be installed with you're OpenFOAM installation. However, older hardware might not be able to run it. Thus, this slightly older version is recommended as it has all the required capabilities that we need but can be run on older computers.

Week 1
OpenFOAM interface

OpenFOAM interface

Perquisites to these slides: OpenFOAM must be installed, as outlined in previous slides.

OpenFOAM does not have a graphical user interface (GUI) as per traditional Windows software. OpenFOAM is designed for Linux operating systems, and thus, it operates from terminal commands. A terminal is where you can type in operations/commands on your computer to run software, or to interact with your files etc.

Program commands are given through the terminal. E.g., to create a mesh, to or to run a solver. The details of the simulation being run are controlled via C++ function files, dictionary files, and library files. Each of these files can be edited using a text editor such as Notepad++. Each simulation requires its own individual function files that are collected together in an organised structure within a simulation directory. This will be discussed further in future slides.

OpenFOAM interface

To use OpenFOAM on Windows, first, you need to find the Run Directory where your simulations will be stored. It will be located in the following directory on your computer within the installed files for OpenFOAM:

```

> This PC > Local Disk (C:) > OpenFOAM > 19.10 > User-dev > run
  
```

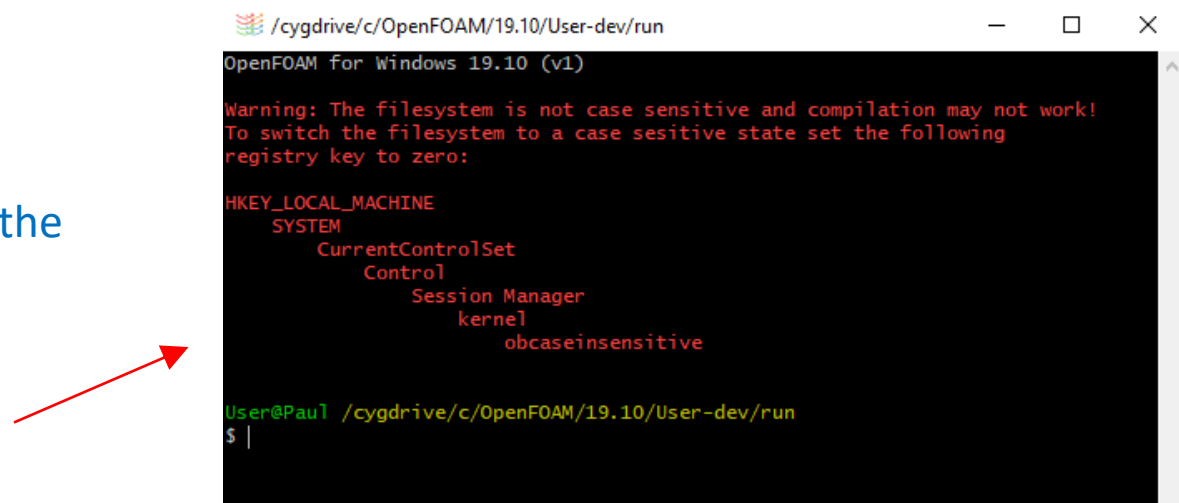
It is recommended that you make a shortcut to this folder on your desktop, so that you can access it quickly and easily.

When you double click on the desktop shortcut for OpenFOAM after it's installation, it will open a Cygwin terminal, which is an emulator for a Linux terminal, and the default directory it will open to will be the above "run" directory



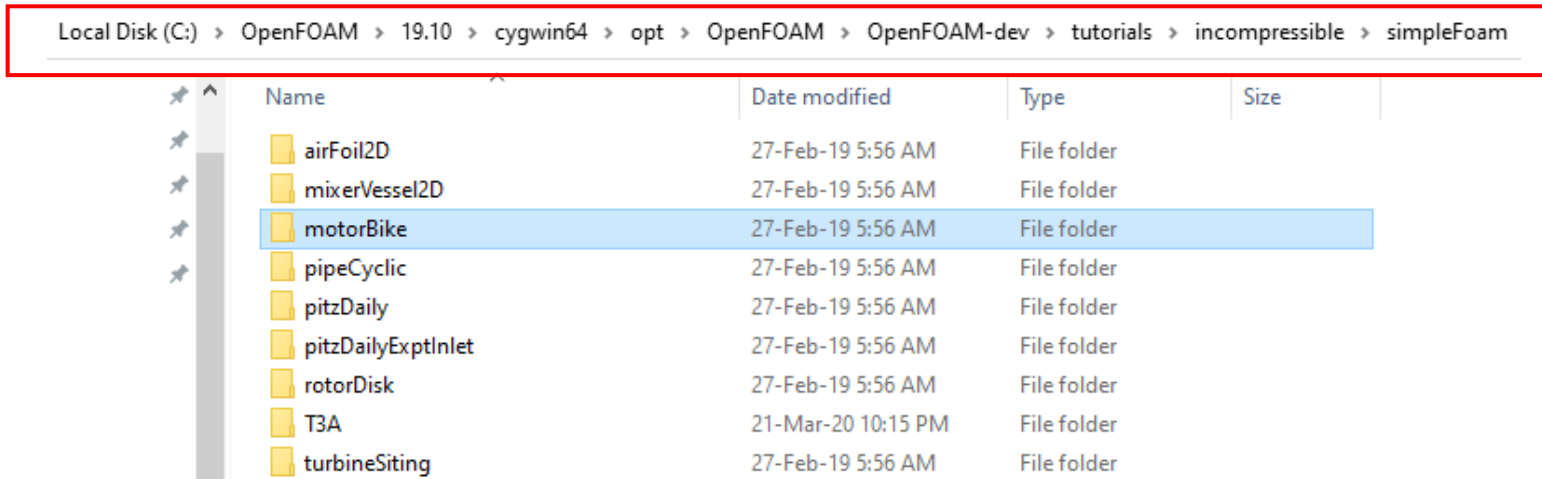
OpenFOAM desktop shortcut, and a shortcut to the "run" directory where the simulations will be located.

Terminal appearance after launching above OpenFOAM desktop shortcut

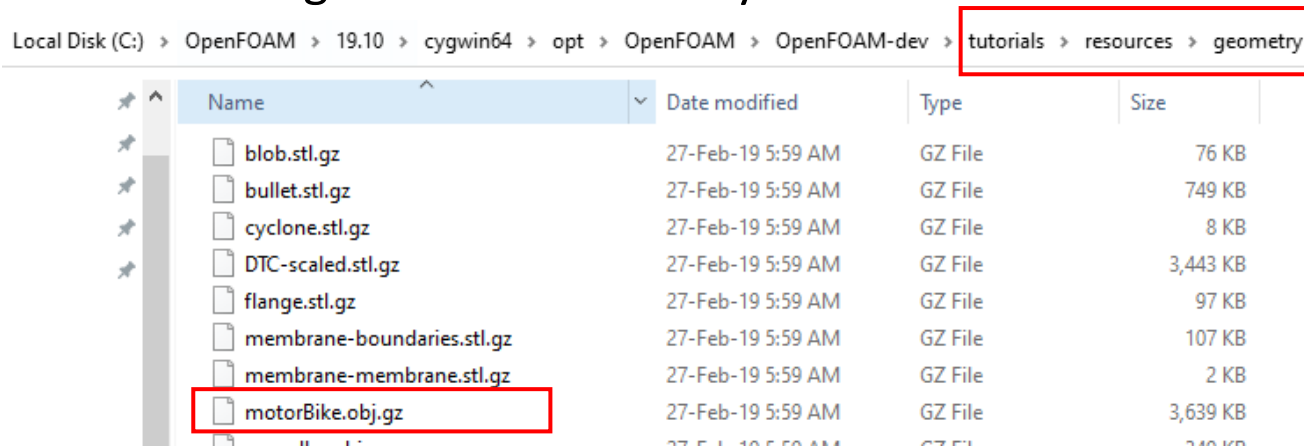


Locating motorbike tutorial

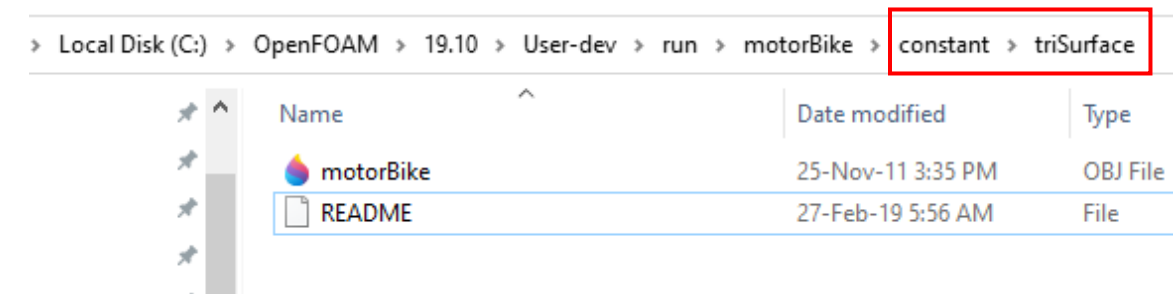
1. The motorbike tutorial within OpenFOAM will be used as the template for this project. The motorbike tutorial can be found in the following directory after OpenFOAM has been installed:



2. The motorbike geometry is located in the following resources directory:

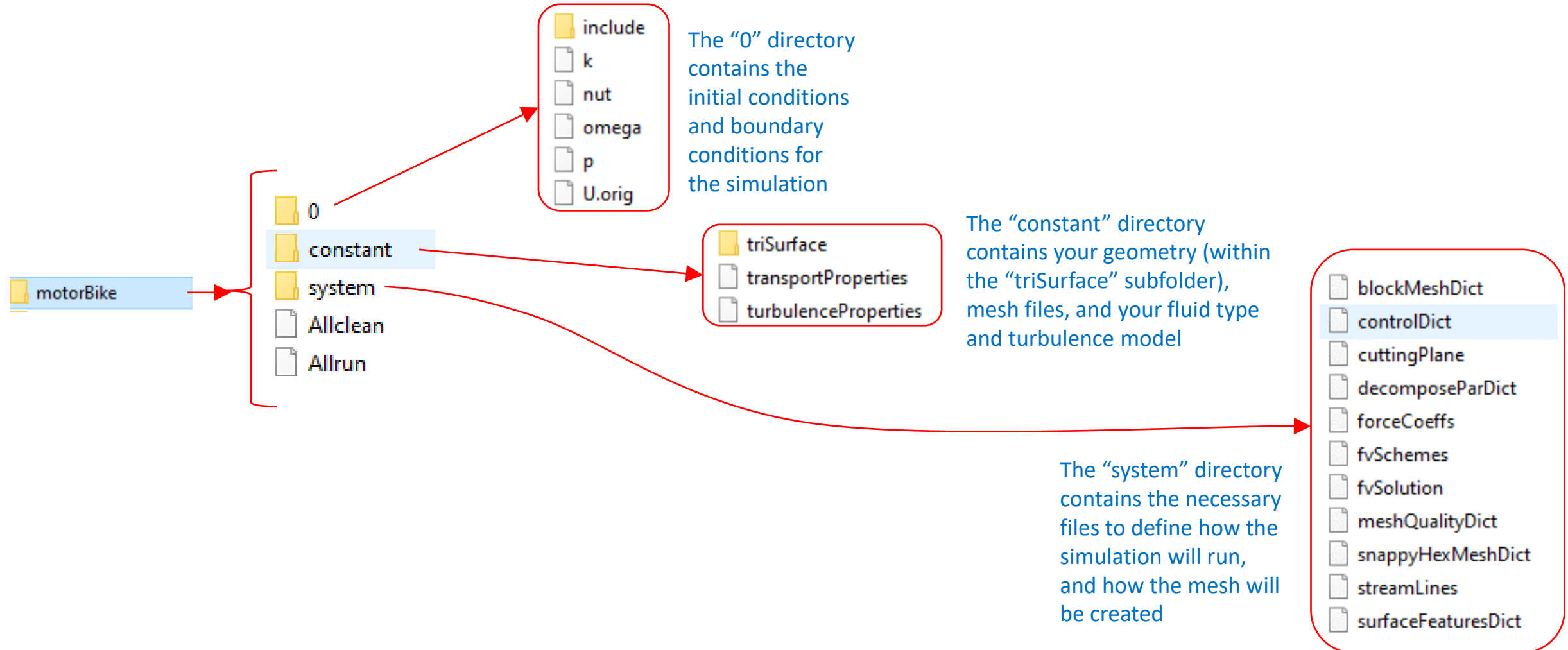


3. Copy the “motorBike” tutorial folder to the “run” directory and copy and then **unzip** the geometry file within the “constant > triSurface” directory.



Locating motorbike tutorial

Content of motorBike simulation:



Basic Linux commands required

The following are basic Linux command that can be used to navigate the terminal to use OpenFOAM

- **cd “directory”**
 - This is the command to open a directory within the terminal. E.g If your folder name is “RearWingSimulation”, then type “**cd RearWingSimulation**” to place the terminal within that directory.
- **cd ..**
 - This is the command to go up one directory, i.e. to exit the directory that you’re currently in.
- **ls**
 - This is the command to list all entities within your current directory. I.e. it will list all files and subdirectories that are in your current directory.

The following link provides an introduction to Linux for those who have no prior experience:

<https://maker.pro/linux/tutorial/basic-linux-commands-for-beginners>

Week 1
Motorbike tutorial

Quick note. How to solve problems?

It is inevitable that participants will run into errors and problems with their project. This is very normal and is expected when building a new OpenFOAM simulation, as it can be a manual process of typing the edits into function files to prepare a new simulation. This slide is to be a guide on how to debug (find errors in the functions) and solving the problems.

1. Double and triple check the slides. The information on each step to be taken is there.
2. Check if the solution has been covered in the troubleshooting document.
3. Search Google, as it is most likely that your question has been answered online previously on one forum or another.
4. If the above fails, then post a question on the dedicated LinkedIn support group. Please ensure that your question provides all the relevant detail. If you need to post a question, please search through the LinkedIn group first to see if somebody else has already asked a similar question, which may have been solved. In addition, if you post a question, please ensure that you also attempt to answer a question from another participant, so that we can learn from each other as a community.
5. If your question does not get answered on the LinkedIn post, email the question to opensourcelearningcf@gmail.com and the course organisers will reply as soon as possible. We ask for some patience with response times, it may take up to a day for you to receive a reply.

OpenFOAM: Running the motorbike tutorial

Open the terminal via the OpenFOAM desktop shortcut.



To run the motorbike tutorial, type the following commands into the terminal one by one, and click enter to run each. These steps will be explained in more detail later in the course.

- The terminal should currently be in the “run” directory after launching OpenFOAM.

```
OpenFOAM for Windows 19.10 (v1)
user@DESKTOP /cygdrive/c/OpenFOAM/19.10/user-dev/run.
$ |
```

- Open the “motorbike” directory by typing the following:

`cd motorbike`

```
user@DESKTOP /cygdrive/c/OpenFOAM/19.10/user-dev/run.
$ cd motorbike

user@DESKTOP /cygdrive/c/OpenFOAM/19.10/user-dev/run/motorbike
$ |
```

- The terminal is now in the “motorbike” directory.

OpenFOAM: Running the motorbike tutorial

- A background mesh needs to be created. Type the following and click enter to run:

blockMesh

```

user@DESKTOP /cygdrive/c/OpenFOAM/19.10/ user-dev/run/motorbike
$ blockMesh
/*-----*/
=====
 \   /  F i e l d      | OpenFOAM: The Open Source CFD Toolbox
  \ /   O peration    | Website: https://openfoam.org
   V    A nd          | Version: dev
  / \   M anipulation |
 /   \
=====
OpenFOAM for Windows 19.04 (v1)
Built by CFD Support, www.cfdsupport.com (based on Symscape).
/*-----*/
Build   : dev-f60147f
Exec    : C:\OpenFOAM\19.10\cygwin64\opt\OpenFOAM\OpenFOAM-dev\platforms\cygwin64\mingw-w64DPInt320pt\bin\blockMesh.exe
Date    : Jul 07 2020
Time    : 19:57:13
Host    : "DESKTOP-HQC2F47"
PID     : 11924
I/O     : uncollated
Case    : C:/OpenFOAM/19.10/maryrose-dev/run/motorbike
nProcs  : 1
fileModificationChecking : Monitoring run-time modified files using timeStampMaster (fileModificationSkew 10)
allowSystemOperations : Allowing user-supplied system call operations
// ***** //
Create time

Deleting polyMesh directory
"C:/OpenFOAM/19.10/maryrose-dev/run/motorbike/constant/polyMesh"
Creating block mesh from
"C:/OpenFOAM/19.10/maryrose-dev/run/motorbike/system/blockMeshDict"
Creating block edges
No non-planar block faces defined
Creating topology blocks
Creating topology patches

Creating block mesh topology

Check topology

Basic statistics
Number of internal faces : 0
Number of boundary faces : 6
Number of defined boundary faces : 6
Number of undefined boundary faces : 0
Checking patch -> block consistency

```

```

Creating block offsets
Creating merge list .

Creating polyMesh from blockMesh
Creating patches
Creating cells
Creating points with scale 1
Block 0 cell size :
i : 1
j : 1
k : 1

There are no merge patch pairs edges

Writing polyMesh
-----
Mesh Information
-----
boundingBox: (-5 -4 0) (15 4 8)
nPoints: 1701
nCells: 1280
nFaces: 4224
nInternalFaces: 3456
-----
Patches
-----
patch 0 (start: 3456 size: 320) name: frontAndBack
patch 1 (start: 3776 size: 64) name: inlet
patch 2 (start: 3840 size: 64) name: outlet
patch 3 (start: 3904 size: 160) name: lowerWall
patch 4 (start: 4064 size: 160) name: upperWall

End

user@DESKTOP /cygdrive/c/OpenFOAM/19.10/ user-dev/run/motorbike
$ |

```

OpenFOAM: Running the motorbike tutorial

- The edges of the geometry must be extracted for mesh refinement purposes. Type the following and run:

`surfaceFeatures`

```
User@Paul /cygdrive/c/OpenFOAM/19.10/User-dev/run/motorBike
$ surfaceFeatures|
```

- The mesh for the motorbike geometry can now be created. Type the following and run:

`snappyHexMesh -overwrite`

```
User@Paul /cygdrive/c/OpenFOAM/19.10/User-dev/run/motorBike
$ snappyHexMesh -overwrite|
```

- The simulation needs to be initialised with a simple solver, before running the RANS solver. RANS = Reynolds-Averaged Navier Stokes, and it calculates an averaged solution. By performing this initialisation, it reduces the risk of the RANS solver diverging early in the full simulation.

`potentialFoam`

```
user@DESKTOP /cygdrive/c/OpenFOAM/19.10/ user-dev/run/motorbike
$ potentialFoam
```

- Launch the RANS simulation solver. Type the following and run:

`simpleFoam`

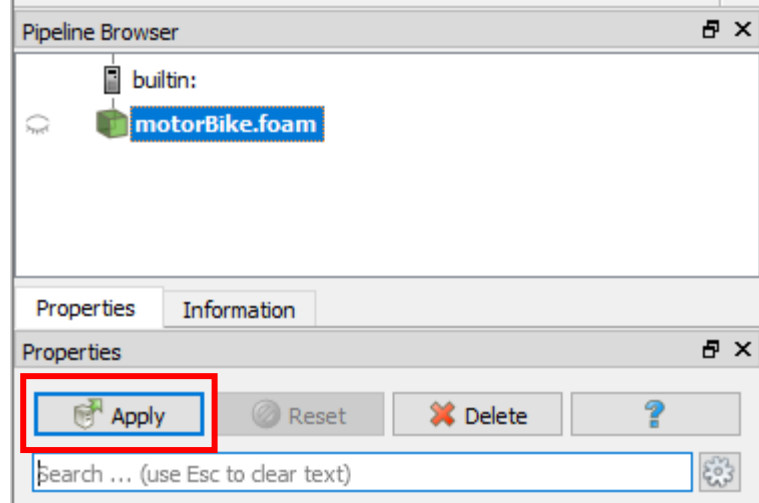
```
user@DESKTOP /cygdrive/c/OpenFOAM/19.10/ user-dev/run/motorbike
$ simpleFoam
```

OpenFOAM: Running the motorbike tutorial

- To view the results, we use a post-processing software. The post-processing software ParaView can be launched by typing the following command in the terminal:

`paraFoam`

- ParaView will automatically read in the results from the directory that you are currently in.
- Select the “Apply” button in the “Properties” tab to view the results.

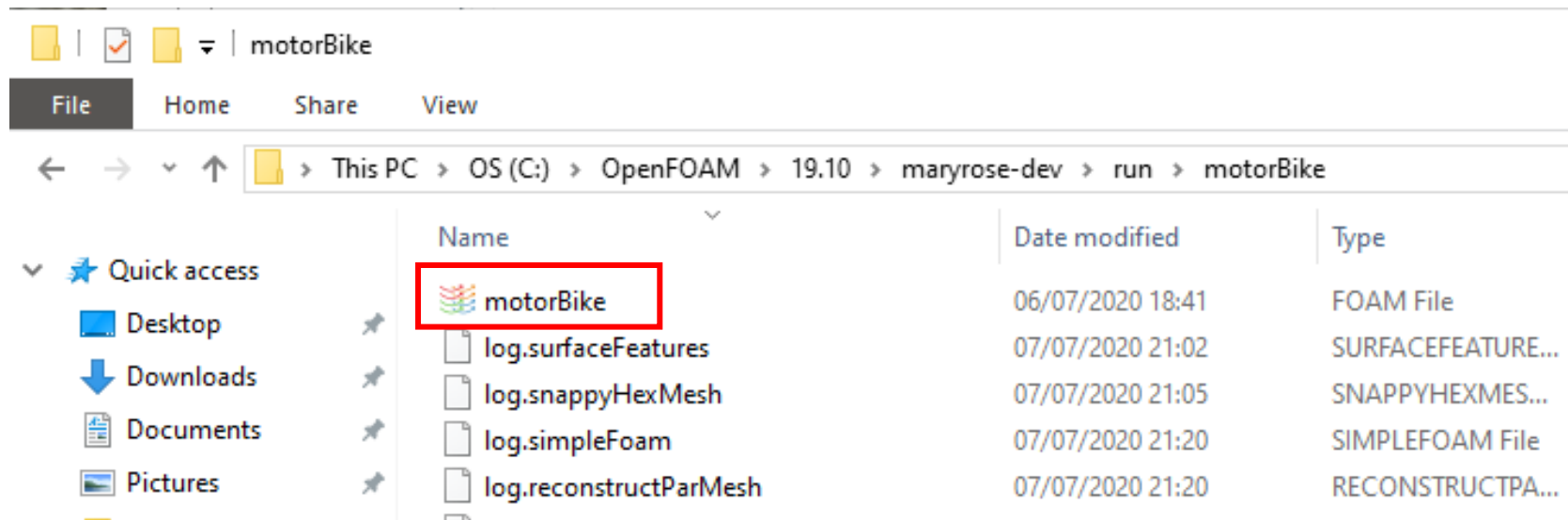


- If you encounter any issues or errors, consult the troubleshooting guide.

ParaView post-processing

Tip

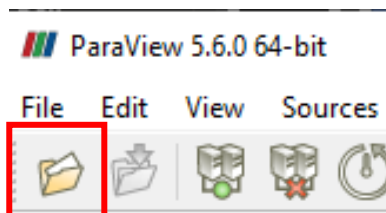
- To open OpenFOAM files from the ParaView program, you need a “.foam” file. When you run “paraFoam” in the terminal, it creates a temporary .foam file within your directory.
- Before exiting out of ParaView, make a copy of this temporary .foam file for viewing the results at a later time.
- Keep this copied .foam file within your simulation directories, and direct ParaView to it when opening results.



ParaView post-processing

Tip (continued)

- To open .foam files in ParaView, select the “Open” button.

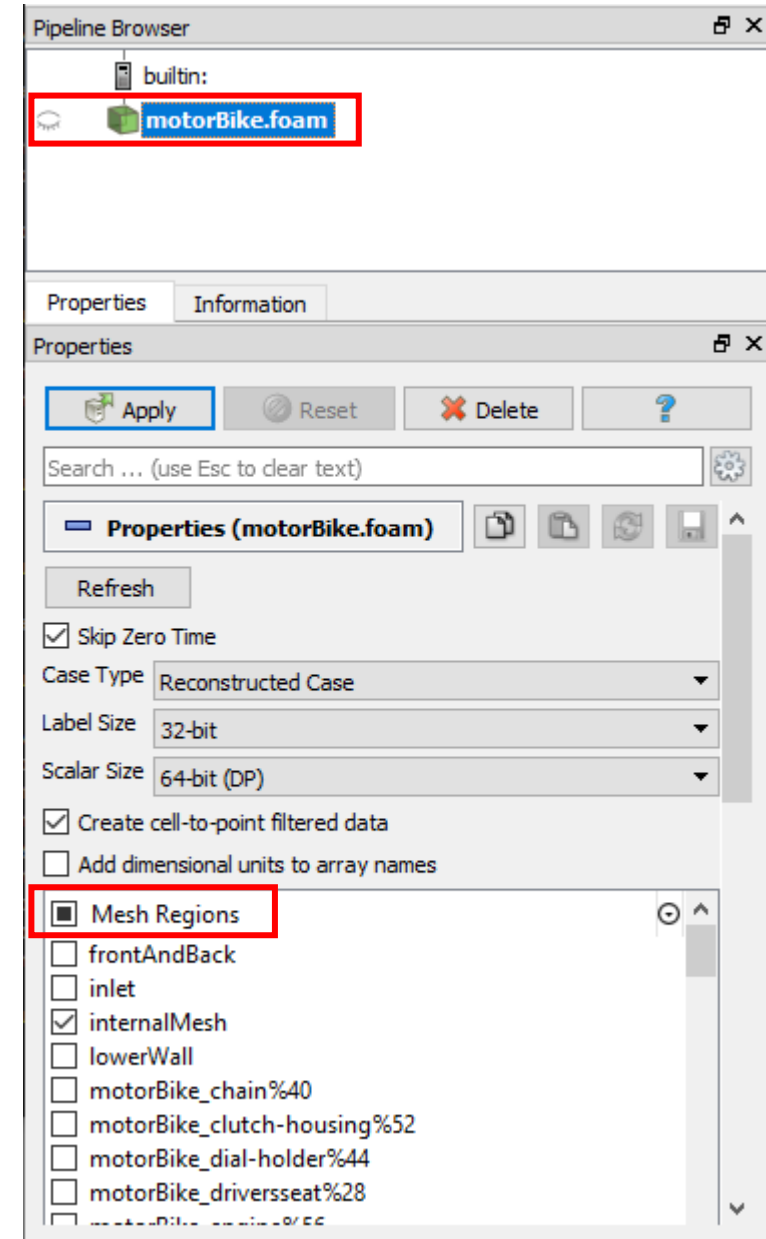


- In the “motorbike” folder, select the file “motorBike.foam”. This file was created after running the “paraFoam” command.

| Filename | Type |
|----------------|-----------|
| 0 | Folder |
| 500 | Folder |
| 501 | Folder |
| 502 | Folder |
| 503 | Folder |
| constant | Folder |
| postProcessing | Folder |
| > processor.. | Group |
| system | Folder |
| motorBike.foam | foam File |

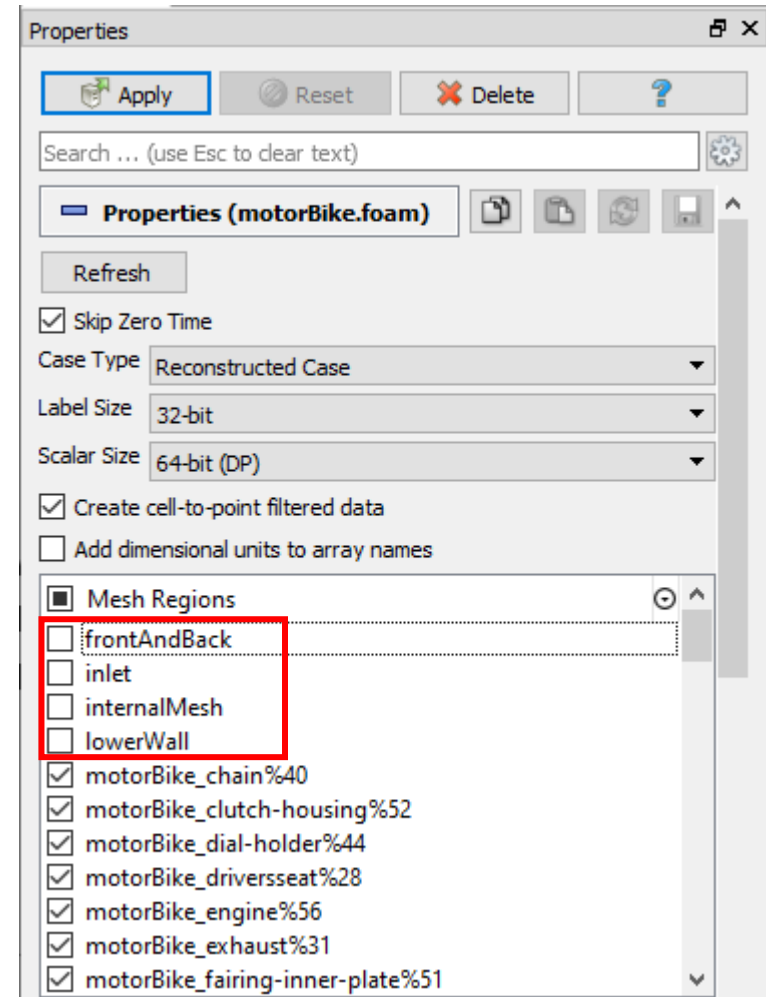
ParaView post-processing

- The “motorbike.foam” file has been loaded into ParaView.
- In the “Properties” tab, deselect “Mesh Regions” to select all of the components.



ParaView post-processing

- All of the motorbike geometry parts and mesh sections are now selected.
- Deselect the following:
 - frontAndBack
 - Inlet
 - internalMesh
 - lowerWall
 - Outlet
 - upperWall
- We do this so that we can view the motorbike and not the surrounding domain (inlet, outlet, and side walls).
- Select the “Apply” button.

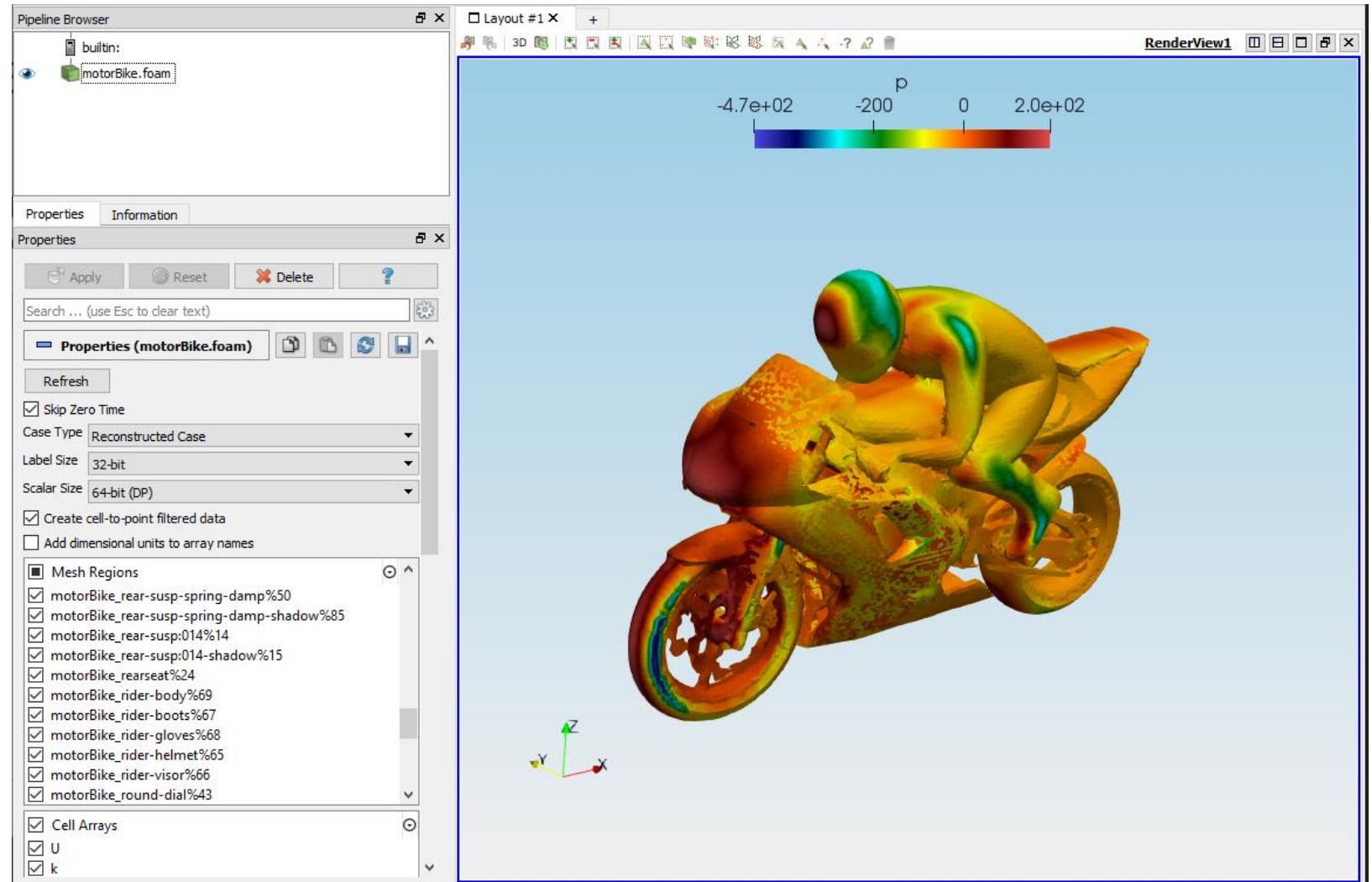


ParaView post-processing

The results for the motorbike simulation are now on screen.

Tip

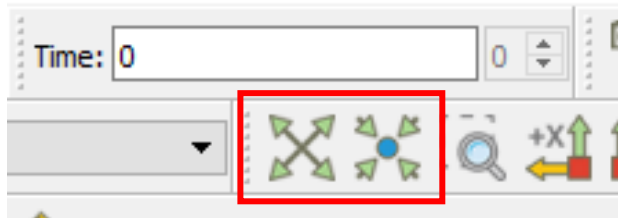
- Use the left mouse button to rotate the view.
- Use the right mouse key to zoom in and out.



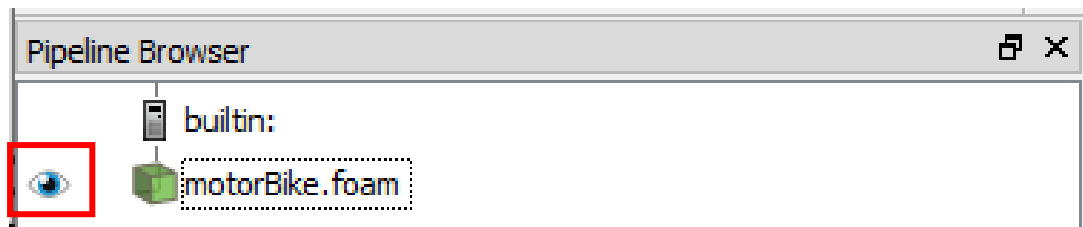
ParaView post-processing

Tip

- If the results move out of view, select the “Reset” or “Zoom to data” button to reset the view.

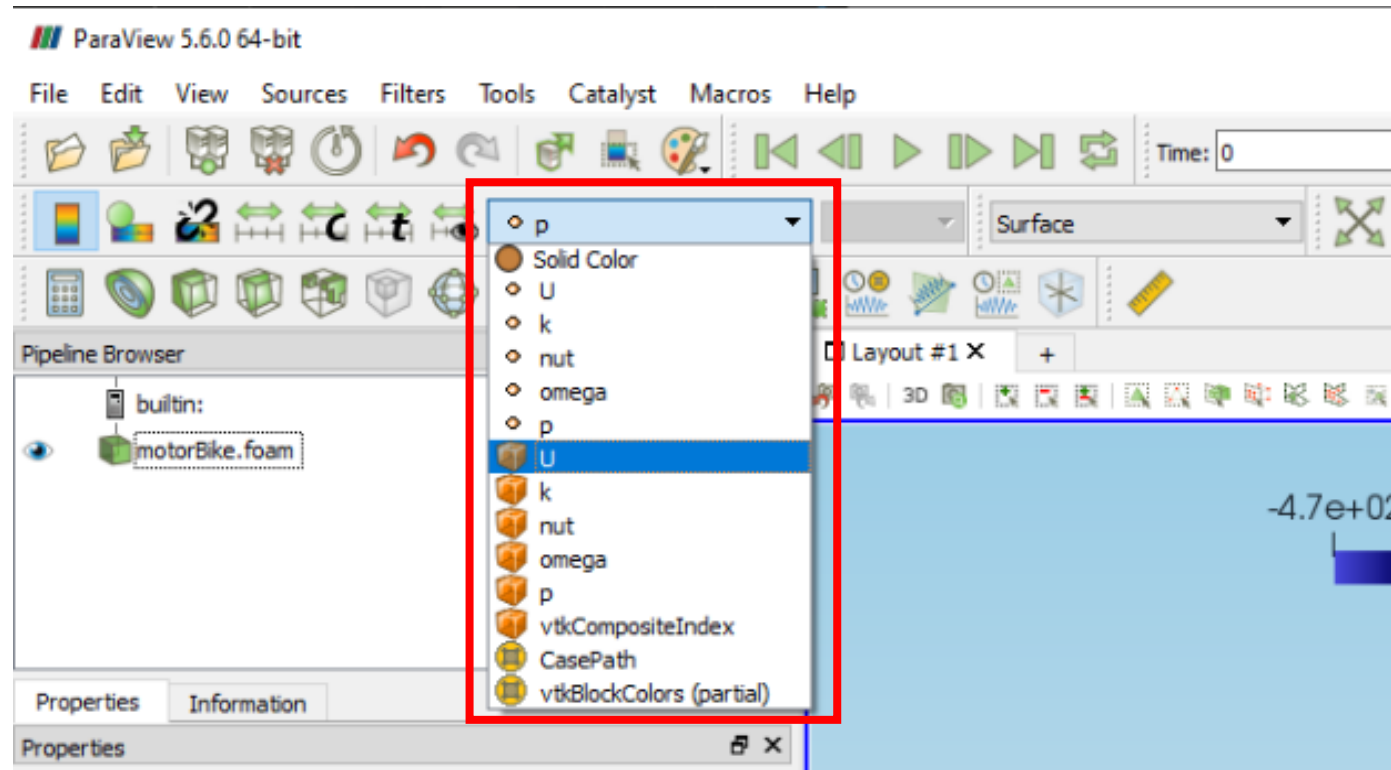


- Ensure the results are visible by checking that the “eye” button is selected.



ParaView post-processing

- The results which are displayed in ParaView are for the final time step.
- The initial results shown are for pressure distribution.
- To view other results, select the dropdown menu, as shown below.

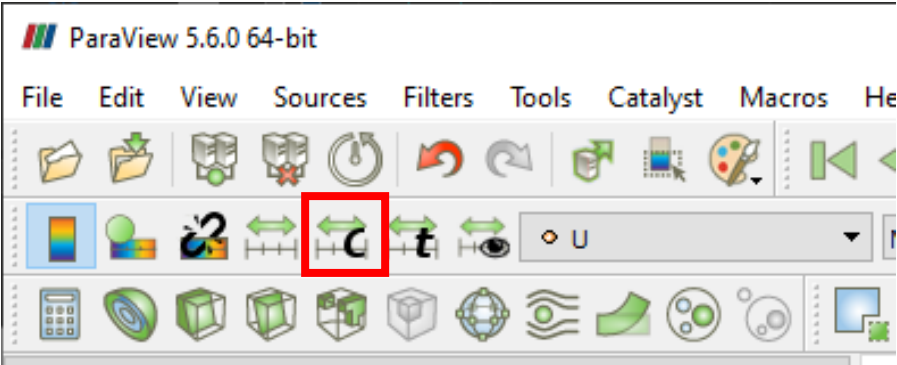


ParaView post-processing

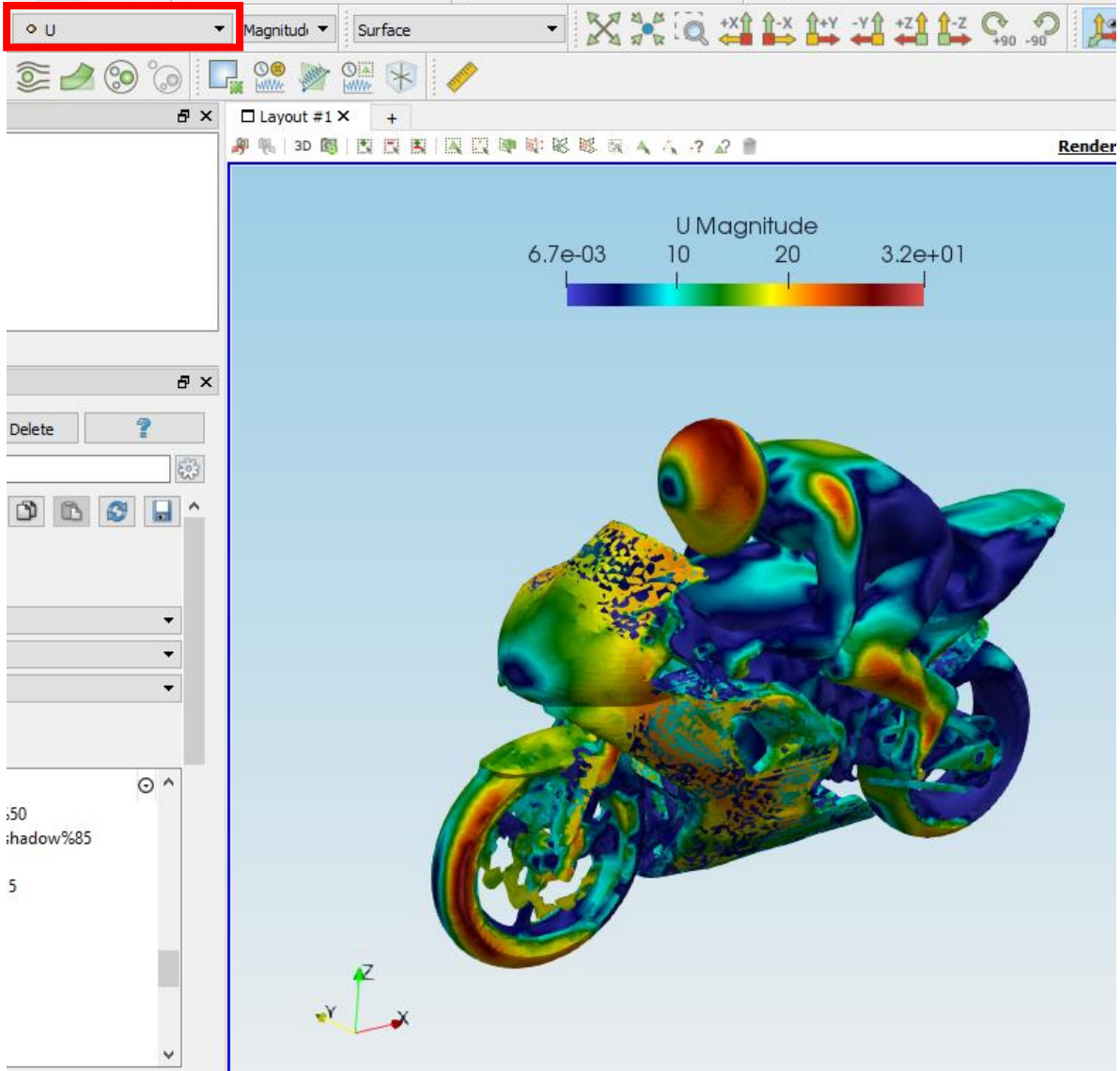
- Here are the results for velocity.

Tip

- You can rescale the scales using the “Rescale to custom range” button.



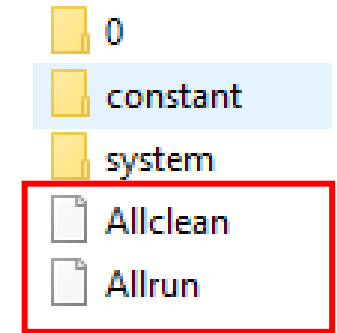
- The motorbike test case is now complete.



Allclean & Allrun

Tip

Within the motorbike directory, you will find two script files. “Allclean” and “Allrun”



Allclean

- This is a script to clean your simulation directory. It will delete all simulation data and mesh files created. It is recommended to use this script when starting a new simulation, so that no previously generated data can cause any interference with your new simulation. Another good use for this script is if you have tried to generate a mesh several times without cleaning the directory first and are running into errors.
- Run this script by typing “Allclean” into the terminal and hitting enter on your keyboard.

Allrun

- Allrun is a script that automates the simulation process. It will run all of the commands required to build your mesh and submit your simulation. Open in Notepad ++ to view the terminal commands it uses. Run this script by typing “Allrun” into the terminal and hitting enter.

Run directory for new simulation

- To set up a new directory for your simulation project, create a copy of the “motorbike” folder in the “run” folder.
- Rename this new directory to “RearWingSimulation”
- Within your terminal, enter this new directory (`cd ../RearWingSimulation`).
- Run the “allClean” command within the terminal. This will run a script that will delete all mesh and simulation data from the directory.
- Your new simulation directory and OpenFOAM files are now ready to be worked on. You should be left with three subdirectories; “0”, “constant”, and “system”, and any other files/scripts that were within the simulation directory outside of these directories.

```

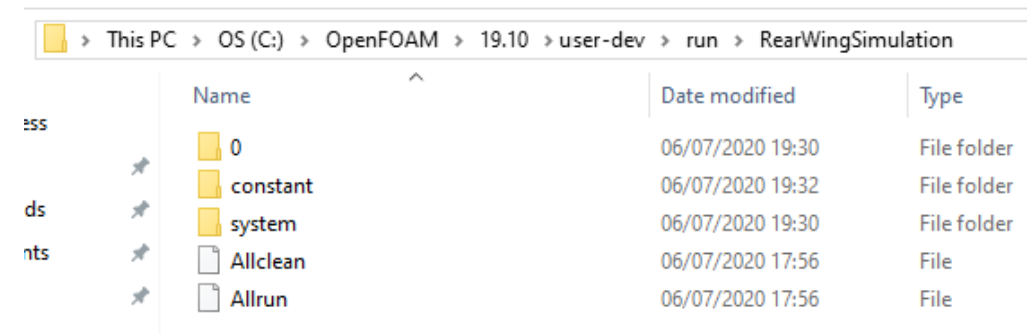
user@DESKTOP/cygdrive/c/OpenFOAM/19.10/user-dev/run
$ cd ..

user@DESKTOP/cygdrive/c/OpenFOAM/19.10/user-dev/run
$ cd RearWingSimulation/

user@DESKTOP/cygdrive/c/OpenFOAM/19.10/user-dev/run/RearWingSimulation
$ allClean
Cleaning /cygdrive/c/OpenFOAM/19.10/maryrose-dev/run/RearWingSimulation case

user@DESKTOP/cygdrive/c/OpenFOAM/19.10/user-dev/run/RearWingSimulation
$ |

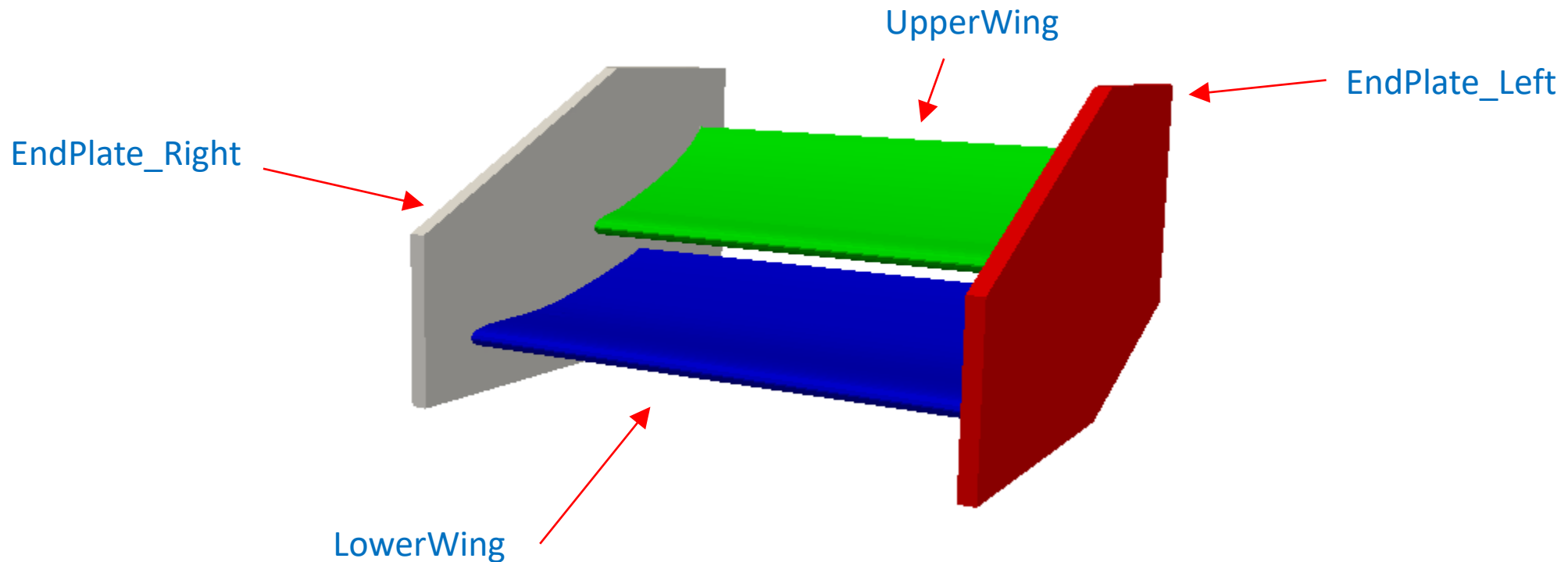
```



| Name | Date modified | Type |
|----------|------------------|-------------|
| 0 | 06/07/2020 19:30 | File folder |
| constant | 06/07/2020 19:32 | File folder |
| system | 06/07/2020 19:30 | File folder |
| Allclean | 06/07/2020 17:56 | File |
| Allrun | 06/07/2020 17:56 | File |

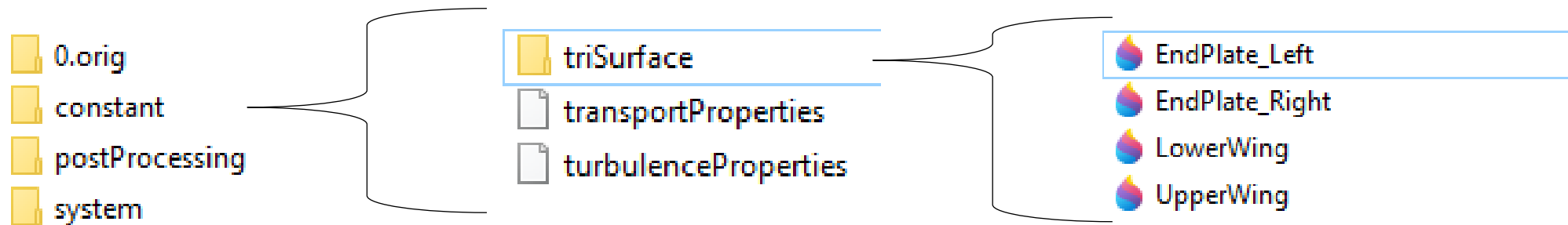
Geometry

- The geometry prepared for this project is a simplified formula student rear wing.
- Each of the component names are provided below.



Geometry location

- A surface mesh file (.stl) of a simplified formula student rear wing is provided for this project.
- Place the “.stl” files within the “constant/triSurface” directory.



C++

OpenFOAM is written in the C++ programming language. OpenFOAM is technically a C++ library, that creates executables as applications we use within OpenFOAM. Solvers (designed to solve specific problems) and utilities (tools to perform tasks and functions) are the two general categories that OpenFOAM applications fall into.

The syntax for C++ must be followed when editing OpenFOAM files. This can generally be followed correctly by following how existing code is written and following the same format.

The following is a quick introduction to C++ with OpenFOAM

http://www.tfd.chalmers.se/~hani/kurser/OS_CFD_2014/basicsOfC++.pdf

For more in-depth information, there is further free material available on Youtube and across the internet:

<https://www.youtube.com/watch?v=vLnPwxZdW4Y>

This course will not cover how to write C++ code from scratch for OpenFOAM, as this is a more advanced topic. Instead, we will show you how to adapt existing function files for our purposes.

Week 1 complete

In week 1, we have:

- Installed OpenFOAM and post-processing software.
- Ran the motorbike tutorial and viewed the results on ParaView.
- Became familiar with the OpenFOAM commands and the ParaView software.
- Added the .stl files for the rear wing simulation.

In week 2, we will create a volume mesh for our simulation to solve our fluid equations on.