

Politechnika Warszawska
Wydział Mechaniczny Energetyki i Lotnictwa

CFD CALCULATIONS OF FORMULA STUDENT CAR IN OPENFOAM ON AWS CLUSTER

Obliczenia inżynierskie w chmurze

Adam Cisowski

January 19, 2019

Contents

| | | |
|---|------------------------------------------------------|---|
| 1 | Introduction | 2 |
| 2 | Model | 2 |
| 3 | Mesh | 3 |
| 4 | Solver | 3 |
| 5 | Step by step | 3 |
| | 5.1 Generate mesh | 3 |
| | 5.2 fluent3DMeshToFoam | 3 |
| | 5.3 Boundary condition | 3 |
| | 5.4 Forces and Coefficients | 4 |
| | 5.5 Convert mesh to polyhedral | 4 |
| | 5.6 Parallel calculations | 5 |
| | 5.7 Calculations | 5 |
| 6 | Create an OpenFoam cluster on AWS | 5 |
| | 6.1 Ssh Agent | 5 |
| | 6.2 Sharing the Master Instance Volume | 5 |
| | 6.3 Mounting the Master Volume from Slaves | 6 |
| | 6.4 Run calculation | 6 |
| 7 | Results | 7 |
| | 7.1 Contours | 7 |
| | 7.2 Comparison with Fluent | 8 |
| 8 | Working on AWS Amazon | 9 |
| 9 | Summarize | 9 |

1 Introduction

Formula Student is a student engineering competition in which participate over 800 teams from around the world. WUT Racing Team from Warsaw University of Technology has taken part in the competitions from six years. Two constructions has been built and now third construction is being designed. Conception of aerodynamic package is to generate as big down force as it is possible irrespective of drag forces. It makes sens because of low maximum speed during the competitions (it is max 90kmph). The down force increase friction between road and tires. Thanks for that the car rides faster in turns.

In this report will be presented OpenFoam's calculations of entire Formula Student car in comparison to Ansys Fluent's calculations on the same mesh.

2 Model

The model was created in Siemens NX. Each aerodynamic element has been optimized firstly in 2D, next in 3D.

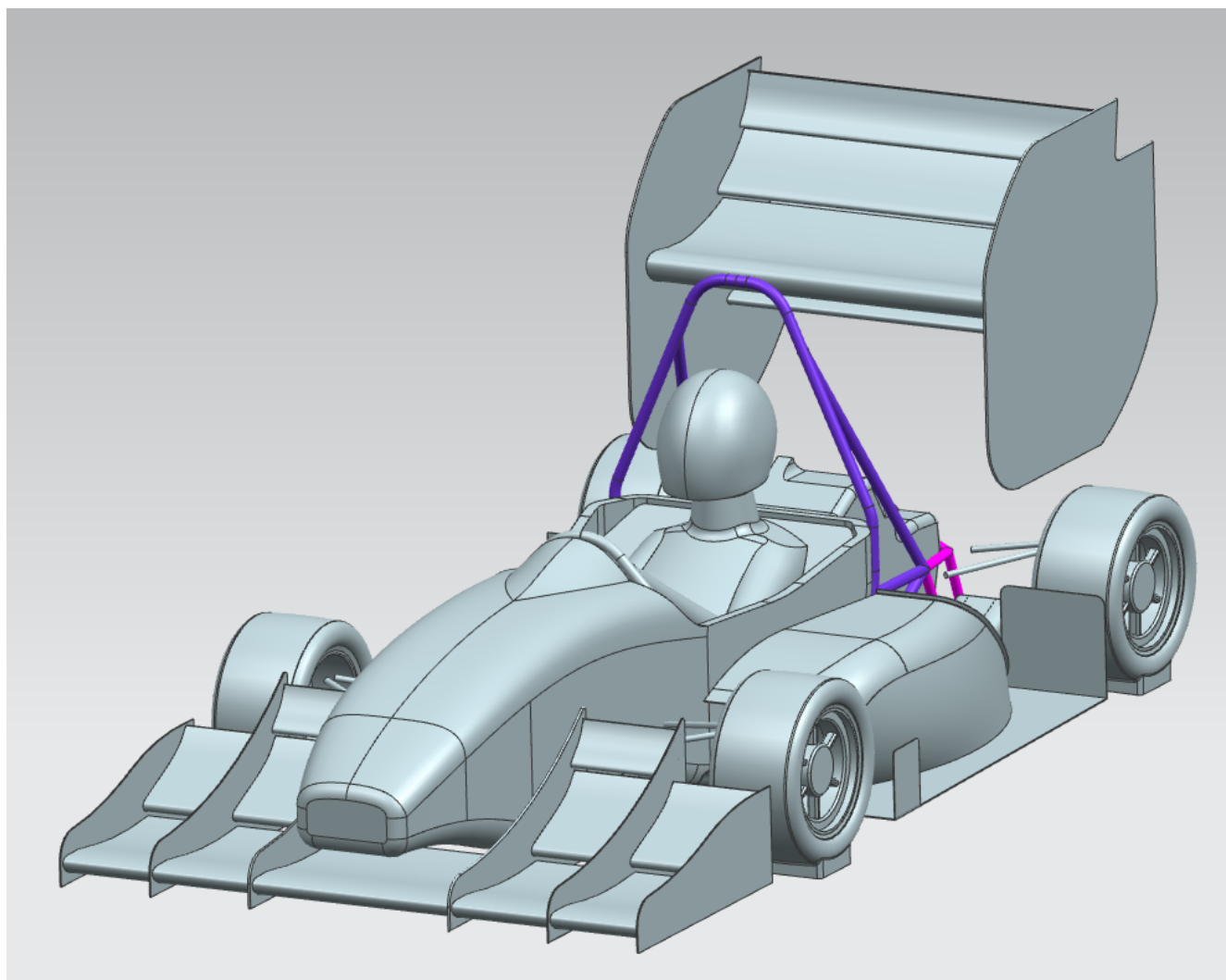


Figure 1: WUT3 aerodynamics model

3 Mesh

The mesh was created in Ansys Mesher. It was very difficult to generate good quality mesh, because of complex geometry. Minimum Orthogonal Quality of the mesh is about $7e-02$.

It is worth to mention that the mesh has to be export in ASCII format. OpenFoam is working on ASCII and it does not read the mesh wrote in binary.

Firstly, the mesh was created with boundary layer, which improve calculations precision near walls. Unfortunately, quality of the mesh getting worse with boundary layer and OpenFoam could not calculate this. The residuals and forces reached large values. In this case Fluent was better than OpenFoam, because it made calculations and results was quite good.

In external flow calculations very good option is polyhedral mesh, which can be made in OpenFoam with command `polyDualMesh`.

4 Solver

For this calculation MotorBike tutorial was adopted. It use incompressible `simpleFoam` solver and `k-omega SST` turbulence model which is the best option for external flows around cars.

5 Step by step

5.1 Generate mesh

As mentioned, the mesh was export in ASCII Fluent Mesh format (.msh) and copied to case folder.

5.2 `fluent3DMeshToFoam`

This command converts Fulent mesh to OpenFoam's format and creates `polyMesh` folder in `constant`. There are all parameters of mesh, for example boundaries, which are import with mesh's named selection.

5.3 Boundary condition

This step is to adapt boundary conditions in time 0 to right case. In the case, inlet velocity magnitude is $15 \frac{m}{s}$. Of course on road is moving wall condition and on wheels are rotating wall condition.

5.4 Forces and Coefficients

Very important for car aerodynamics are forces and coefficients (C_x and C_l). OpenFoam can calculate them. It needs special file named forces in system folder. It looks like this:

```
forces_ns_rear_wing
{
    type            forces;

    libs ( "libforces.so" );

    writeControl    timeStep;
    timeInterval    1.2;

    log            yes;

    patches        ( ns_rear_wing );
    rho            rhoInf;      // Indicates incompressible
    log            true;
    rhoInf         1.2;        // Redundant for incompressible

    CofR           (0 0 0);    // Rotation around centre line of propeller
    pitchAxis      (0 0 1);
}
```

Figure 2: A part of code of forces file

The forces are collected from all car's patches. In result OpenFoam create file with forces and moments in all directions.

5.5 Convert mesh to polyhedral

Instead of tetrahedral mesh will be used polyhedral mesh, which has less elements, and calculations are more stable. From 15mln elements it was made 3mln.

polyDualMesh is using to convert mesh to polyhedral. There is one argument - minimum angle between normals to cell's faces. In this calculation it is 30 degrees. Also it should be used options: -overwrite and -doNotPreserveFaceZones. First one (optional), overwrite new mesh in time 0, which is comfortable. Second delete old faces from mesh. It is necessarily to correct work.

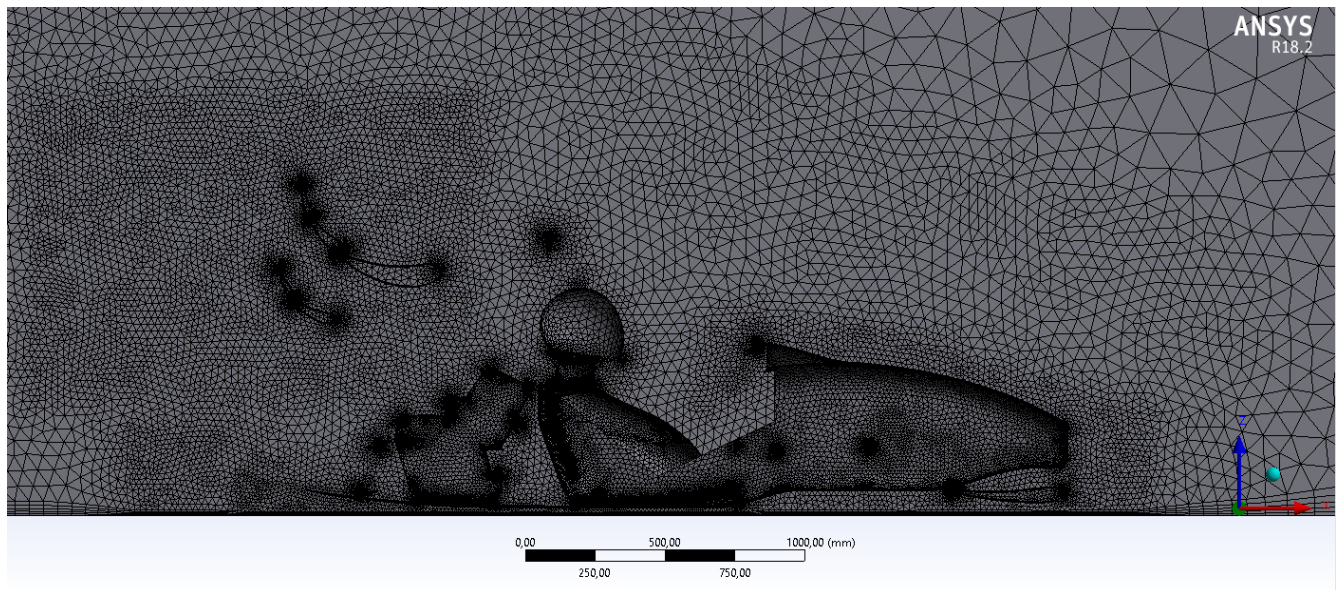


Figure 3: Mesh used in calculations has over 19 mln elements

5.6 Parallel calculations

Calculation was carried out on 32 cores, on four EC2 instances. The instances have been merged to cluster. Scotch decompose method has been chosen.

5.7 Calculations

There was 360 time steps and it was enough to stabilize forces and getting good accuracy.

6 Create an OpenFoam cluster on AWS

Large calculations require a lot of RAM memory and processor's cores. Without required memory the calculations cannot be carried out. With a small number of processors it would take a lot of time. Sometimes, a single instance is insufficient to make CFD calculations. It is possible to merge a few instances to make a larger one.

Create the OpenFoam cluster will be presented. The bash script has been created to automate calculations and it has been placed on the author's git repo.

6.1 Ssh Agent

A premission from master instance to slave instances is given by publikey (.pem) through ssh agent. To enable connection to instances without -i parameter this script should be run:

```
#!/bin/bash
cd ~/.ssh
eval "$(ssh-agent)"
exec ssh-agent bash
ssh-add <klucz>.pem
```

Figure 4: SSH Agent script

Lines begin with "eval" and "exec" are necessarily when something is going wrong.

6.2 Sharing the Master Instance Volume

Next script shares the master instance volume to slave instances. All data is storage in the master instance, slave instances share memory and processors' cores, but all data are save in master instance.

```
sudo sh -c "echo '/home/ubuntu/OpenFOAM *(rw,sync,no_subtree_check)' >> /etc/exports"
sudo exportfs -ra
sudo service nfs-kernel-server start
```

Figure 5: Share volume script

6.3 Mounting the Master Volume from Slaves

Next step merge instances to work together. While calling script, it is required to input 4 argument - private IPs of master and slave instances.

```
SPIPS="$2" "$3" "$4"  
for IP in $SPIPS ; do ssh $IP 'rm -rf ${HOME}/OpenFOAM/*' ; done  
for IP in $SPIPS ; do ssh $IP 'sudo mount "$1":${HOME}/OpenFOAM ${HOME}/OpenFOAM' ; done
```

Figure 6: Connect instances

6.4 Run calculation

Before run calculation it is required to create text file with private IPs of master and slave instances, named "machines". Now case is ready to run by command "foamJob -p -screen simpleFoam". Log file is being created and all logs are being shown on the screen. Log file has been placed in github repository, there are the information about number of cores used in calculations, convergence and all iterations.

7 Results

7.1 Contours

Images show contours of pressure and velocity around the car. It is able to see high pressure on the top of aerodynamic elements and negative pressure - under the elements. This difference generate down force which is useful in turns.

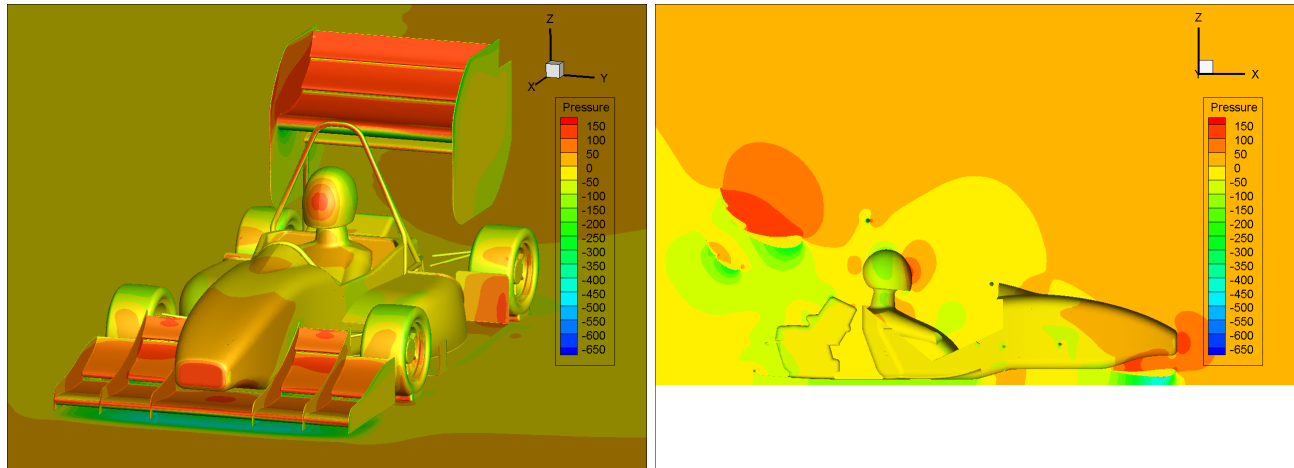


Figure 7: Contours of pressure

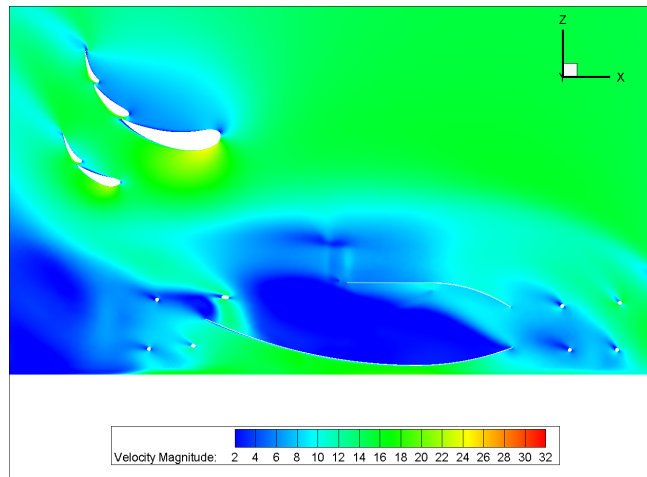
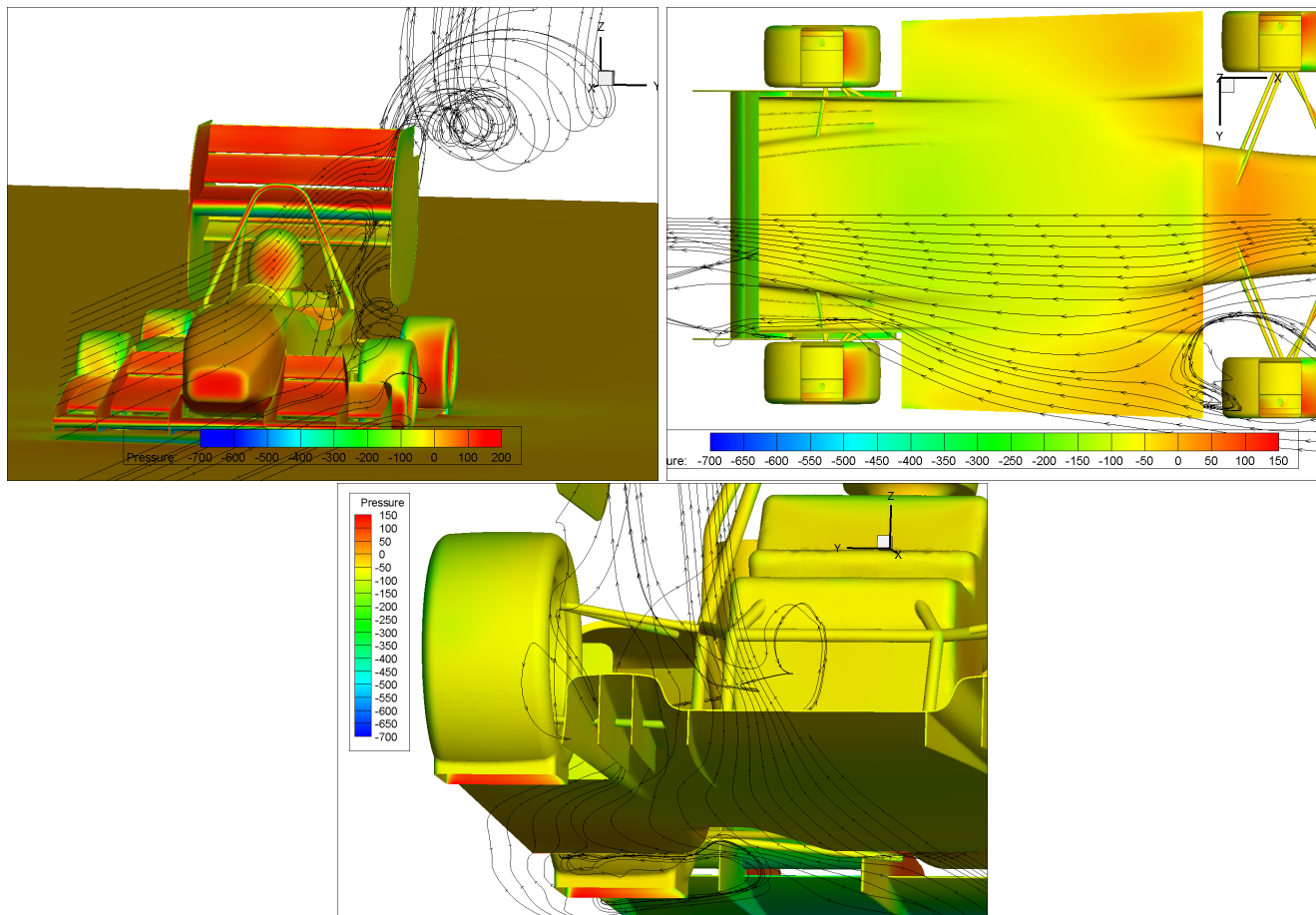


Figure 8: Contours of velocity

Good visualization of the flow are streamlines which show, how air flow over (and under) the car. They give many information and they help improving the construction.



(a) Visualizations

7.2 Comparison with Fluent

The same calculation has been carried out in Fluent, one of the best CFD software. Is OpenFoam, as open source, free software, as good as commercial software? The down forces and coefficients will be compared:

| Patch | OpenFoam | Fluent | Difference |
|------------|----------|--------|------------|
| Rear Wing | -178N | -164N | -8,5% |
| Front Wing | -220N | -258N | 15% |
| Diffuser | -190N | -132N | -44% |
| Others | 80N | 52N | 54% |
| Sum | -508N | -502N | 1% |
| Cl | -2,53 | -2,5 | — |
| Cx | 1,03 | 1,00 | — |

Differences are quite big between OpenFoam and Fluent. The reason may be in low mesh quality (especially near the road) and not enough elements.

8 Working on AWS Amazon

The case has been create on PC and coarse mesh has been converted to OpenFoam format to check entire case. When the case was correct, it has been sent to S3 storage with fine mesh.

EC2 instances was launched from OpenFoam AMI, avalible in AWS Marketplace. Then the files have been download from S3 to EC2 instance by command "aws s3 cp s3://bucket-name/file ./". Next, the bash script has been launched.

The results has been sent to S3 by command "aws s3 sync <folder> s3://bucket-name/folder" or it was download directly by FileZilla to PC.

9 Summarize

The OpenFoam is very good free software, but in more complex calculations it might fail. Create the 4 instances cluster makes calculations faster. The biggest problem in creating cluster was to use ssh agent, because of many errors. Automatization of this process was problematic too, but finally it works.