

# Team 194: Aerodynamic Study of Airflow around an Airfoil in the EGI Cloud



*"CFD Support's  
OpenFOAM and  
UberCloud Containers  
enable efficient,  
effective, and easy*

## MEET THE TEAM

**End-User/CFD Expert:** Praveen Bhat, Technology Consultant, INDIA

**Software Provider:** Lubos PirkI, CFD Support with OpenFOAM, and UberCloud software container

**Resource Provider:** EGI Cloud: Sy Holsinger, Giuseppe La Rocca, with cloud resources from CESNET, supported by Boris Parak, CESNET in Prague

**UberCloud:** Application software container, Hilal Zitouni, UberCloud.

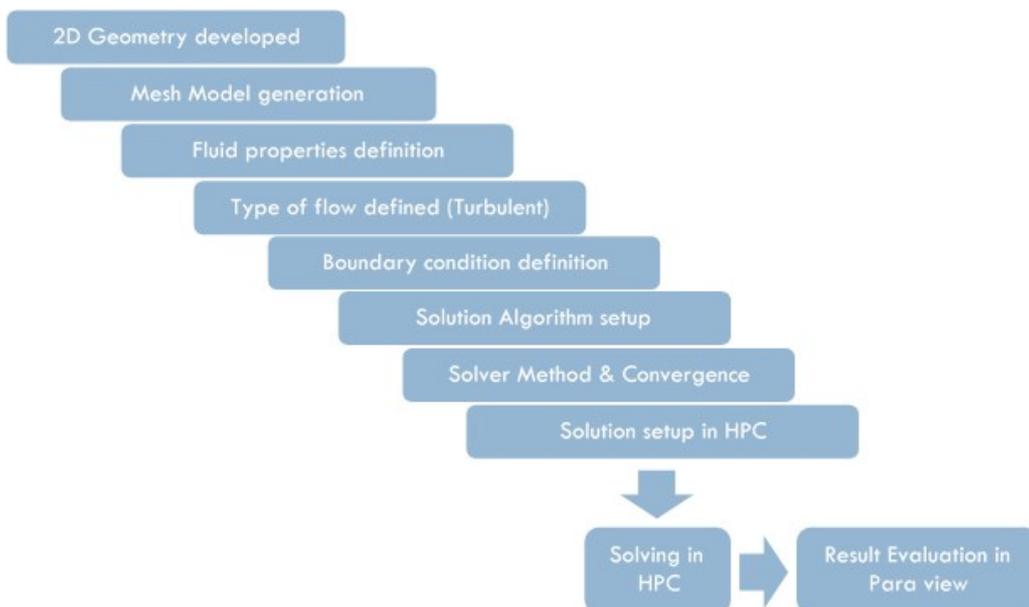
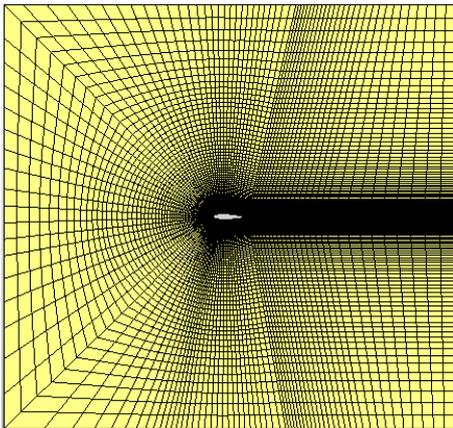


Figure 1: Model Setup flowchart

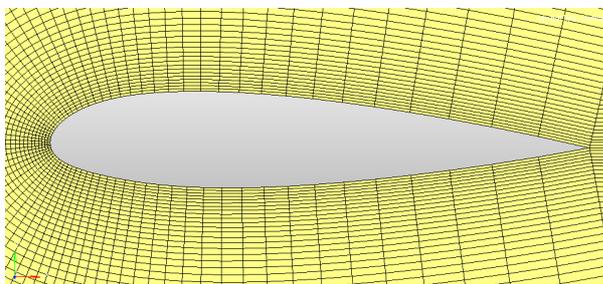
## USE CASE

The aerodynamic study on the 2D airfoil was performed with the incompressible air flow around a 2D airfoil. The model setup included the geometry preparation where a selected region was modelled to represent the surrounding air volume with the airfoil profile at the center. The airfoil profile needed to be accurately modelled to capture the variation in the airflow pattern around the airfoil. The model setup was done using CFD Support's professional OpenFOAM 3.0. The OpenFOAM software is embedded in an UberCloud Container sitting on EGI / CESNET cloud resources in Prague.

The main objective of this project was to experience the ease-of-use of the UberCloud OpenFOAM container and to evaluate HPC performance with respect to the accuracy of result prediction as well as the solution time and resource utilization.



**Figure 2: Mesh model for the airfoil**



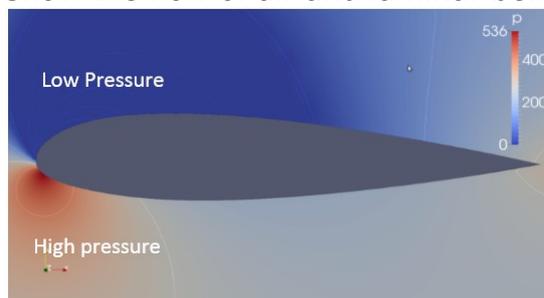
### **Process Overview**

The meshing density is very fine around the airfoil and also along the path of the trailing edge. The meshes were modelled coarser as they moved away from the airfoil region. The coarseness of the mesh increased near the air volume boundary (air inlet and outlet). The following details describe the steps in the simulation model setup using OpenFoam:

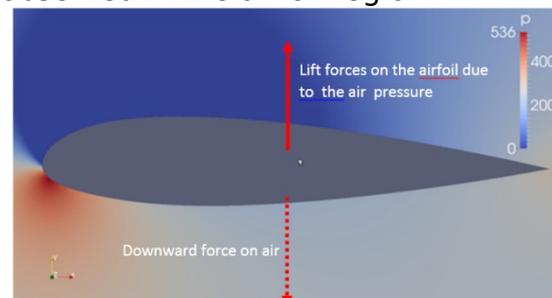
1. The Finite Element mesh model was generated followed by the fluid properties definition. The entire volume surrounding the airfoil is air which is considered incompressible in nature.
2. The fluid properties are defined as Newtonian fluids with a linear relationship between the shear stress (due to internal friction forces) and the rate of strain of the fluid.
3. Atmospheric air is turbulent in nature and there is a transition phase from turbulent to laminar in the region near the airfoil. Because of this transition, the mesh model needs to be refined accurately near the airfoil region along with defining the turbulence behavior of the air which is captured through a Spalart - Allmaras turbulence model.

4. The next section in the model setup was defining the model boundary conditions and assigning the pressure and velocity initial values. The boundary conditions were assigned where the airfoil edges were considered as a wall. The three sides of the air volume were considered as an inlet, and the edge following the trailing edge of airfoil was considered as an air outlet.
5. The next step in the model development was setting up the solution algorithm. The problem was solved as steady state. The OpenFOAM solver used for solving this problem was Simple FOAM. The following are the solution parameters for the SimpleFOAM solver: Start time: 0 sec; End time=500 sec; time step= 1sec. The SimpleFOAM solver uses the Gauss-Seidel method for solving. The pressure field is provided with a relaxation factor of 0.3 and the velocity field is assigned a relaxation factor of 0.7, along with the residual parameter which is set at  $1.0 \times 10^{-5}$ . The above parameters define the convergence criteria of the model.
6. The OpenFOAM model was then modified for parallel processing where the existing model was divided according to the number of HPC computing nodes.
7. The model was solved in parallel. Once the solution was converged, the solved model in the parallel processors was reconstructed to get the final simulation results. The final result is used to view the output of the airfoil simulation, and the respective result components are captured using the post-processing software tool ParaView.

The airfoil was surrounded by air volume and the variation in the flow velocity and air pressure near the airfoil section was reviewed. The different plots below show the flow of air and laminar behaviour observed in the airfoil region.



**Figure 3: Pressure distribution around airfoil with high & low pressure zone**

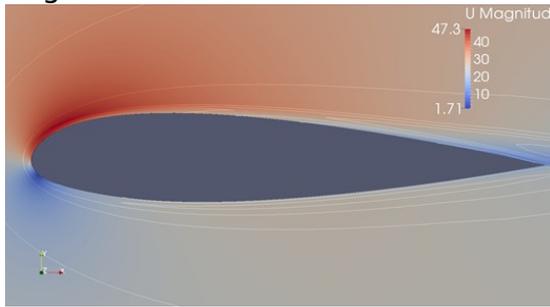


**Figure 4: Lift forces represented in the airfoil**

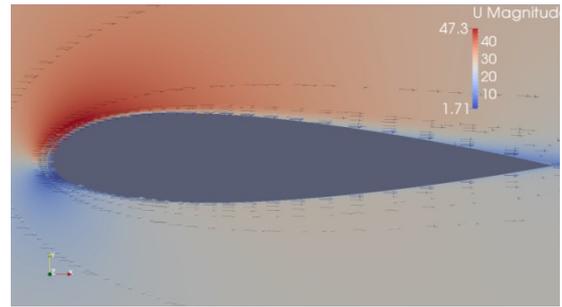
The pressure plot shows the air pressure distribution in the airfoil sections. The first diagram represents the pressure variation around the airfoil where we observe the low pressure region at the upper section of the leading edge of the airfoil and a higher pressure region in the lower section of the leading edge. The low pressure and high pressure variation section in the air volume is shown in the second diagram. The high pressure section near the airfoil creates a lift forces on the airfoil. The lift on the airplane wing follows Newton's third law - there will be a reaction force in the form of downward force on the air. The lift on the airplane wing should be consistent since it is governed by the conservation of the energy in the fluid.

Angle of attack is the orientation of the airfoil cord with respect to the travel direction. The state of stall can be analysed by determining the pressure coefficient distribution over the airfoil for various angles of attack and evaluating

how the pressure co-efficient value varies with the increase or decrease in the angle.



**Figure 5: Velocity contour of streamline of air flow around the airfoil**



**Figure 6: Velocity contour with air flow vectors around the airfoil**

The behavior of air flow was turbulent in the air volume, and the transition of the air behavior from turbulent to laminar was observed in the air volume nearing the airfoil section. The flow behavior of air was laminar around the wall of the airfoil. The airflow path near the wall boundary of the airfoil is laminar which is evident in Figures 5 and 6. The vector flow path of the air in the airfoil region was shown - the flow path represents individual air particle flow near the wall boundary of the airfoil.

### HPC Performance Benchmarking

The HPC Cloud system is a 32 core system with 32 GB RAM with Ubuntu 12.04. The software installed in the container is OpenFOAM version 3.0 with OpenFoam MPI and Paraview. The model was evaluated for the accuracy of the prediction of air flow around the airfoil, with both fine and coarse mesh. The time required for solving the model with different meshes was captured to benchmark the use of HPC performance in solving high density mesh models. The boundary conditions, solution algorithm, solver setup and convergence criteria remain the same for all models.

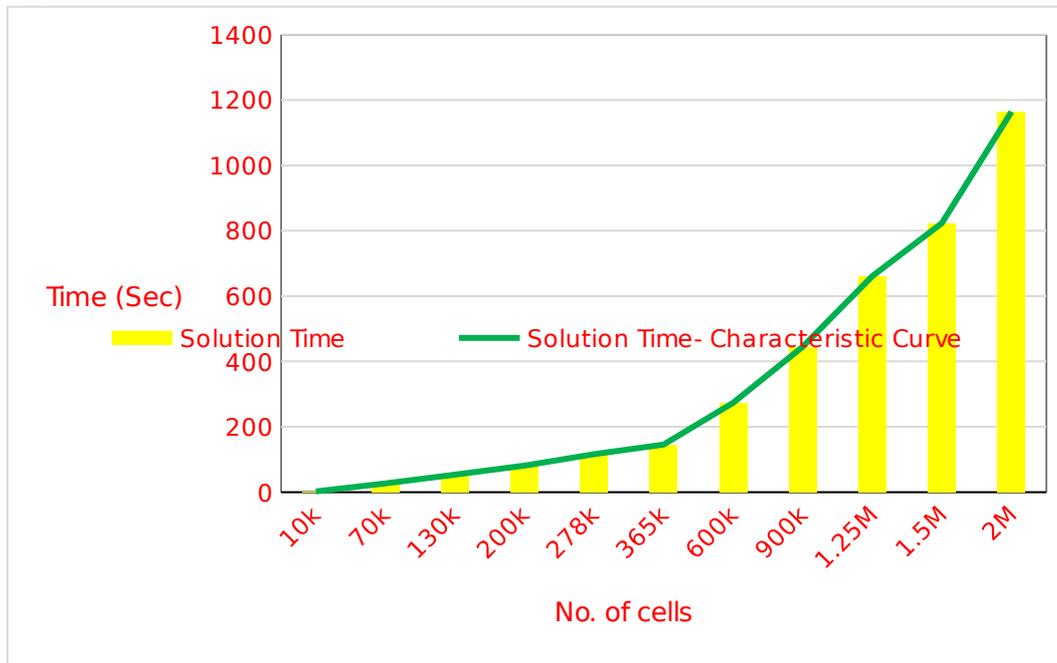
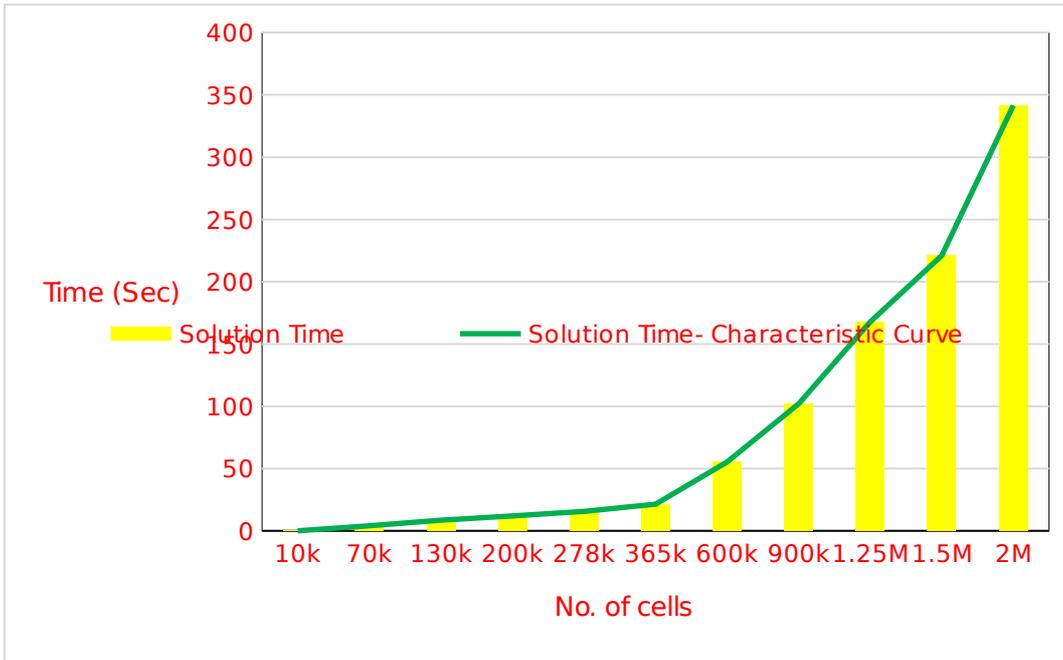
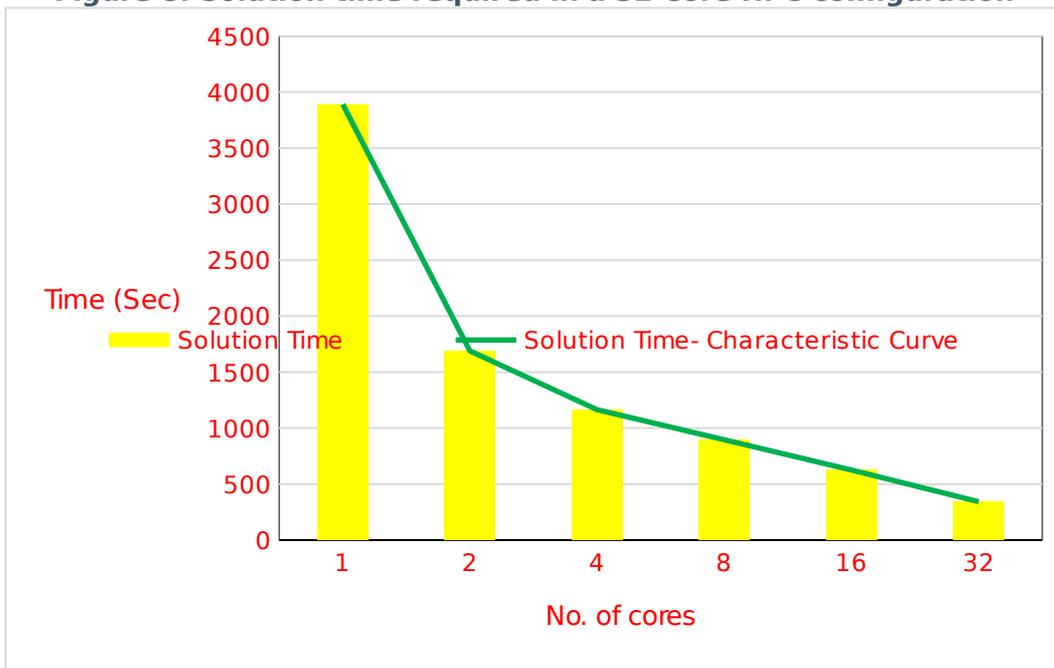


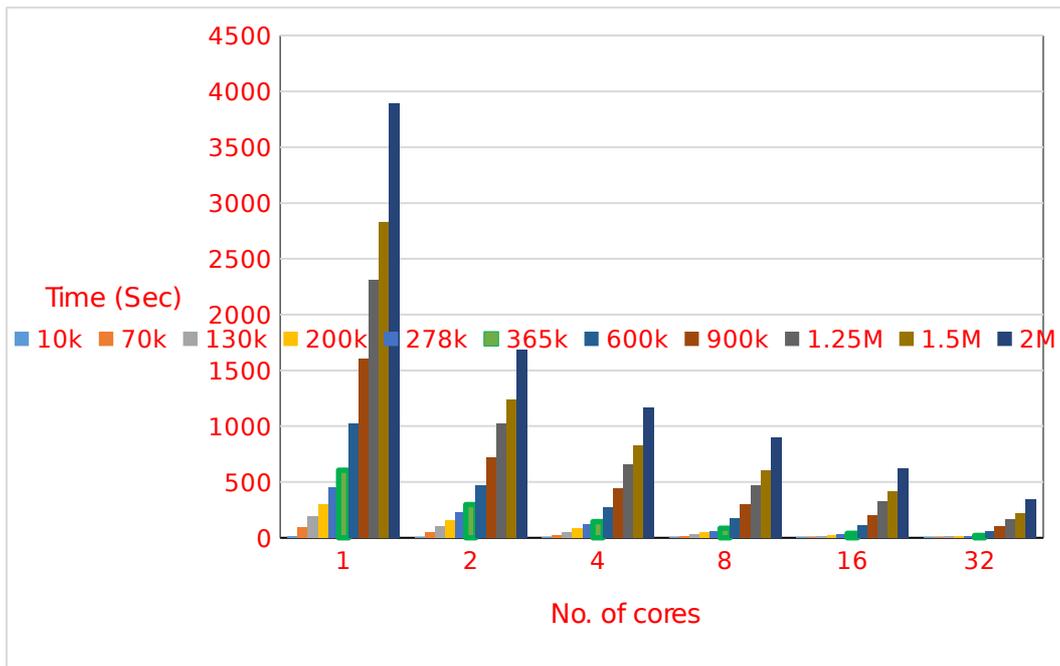
Figure 7: Solution time required in a 4 core configuration



**Figure 8: Solution time required in a 32 Core HPC configuration**



**Figure 9: Solution time for a model with 2M elements solved using different HPC core configurations**



**Figure 10: Comparison of solution time for different mesh densities models using different HPC core configurations**

### Effort Invested

**End user/Team expert:** 10 hours for simulation setup, technical support, reporting and overall management of the project.

**UberCloud support:** 3 hours for monitoring and administration of host servers and guest containers, managing container images (building and installing container images during any modifications/ bug fixes) and improvements (such as tuning memory parameters, configuring Linux libraries, usability enhancements). Most of the mentioned effort is one time and will benefit the future users.

**Resources:** ~200 core hours for performing various iterations in the simulation experiments.

### CHALLENGES

The project started by testing the installation of OpenFOAM 3.0 on the HPC server. Initial working of the application was evaluated and the challenges faced during the execution were highlighted. Once the server performance was enhanced, the next level of challenges faced was related to technical complexity. This involved accurate prediction of flow behaviour around the airfoil which was achieved by defining the appropriate element size to the mesh model. The finer the mesh the higher the simulation runtime required: therefore the challenge was to perform the simulation within the stipulated timeline.

### BENEFITS

1. The HPC cloud computing environment with OpenFOAM and ParaView made the process of model generation easier with process time reduced drastically along with result viewing and post-processing.
2. The mesh models were generated for different cell numbers where the experiments were performed using coarse-to-fine to highly-fine mesh models. The HPC computing resource helped in achieving smoother completion of the simulation runs without re-trials or resubmission of the same simulation runs.

3. The computation requirement for a very fine mesh (2 million cells) is high, which is next to impossible to achieve on a normal workstation. The HPC cloud made it feasible to solve very fine mesh models. The simulation time was drastically reduced providing the advantage of obtaining simulation results within an acceptable run time (~30 min).
4. The UberCloud experiments in the EGI / CESNET Cloud gave us extra confidence in the setup and the ability to run the simulations remotely in the cloud. The different simulation setup tools were installed in the UberCloud Container - this enabled the user to access the tool without any prior installations.
5. With the use of VNC Controls in the web browser, The UberCloud Container access was very easy with no installation of any pre-requisite software. The whole user experience was similar to accessing a website through the browser.
6. The UberCloud Containers helped with smoother execution of the project with easy access to the server resources. The regular UberCloud auto-update module via email provided huge advantage to the user that enabled continuous monitoring of the job in progress without any requirement to log-in to the server and check the status.

## **CONCLUSION & RECOMMENDATIONS**

1. The selected HPC Cloud environment with UberCloud containerized OpenFOAM 3.0 on EGI / CESNET cloud resources was a very good fit for performing advanced computational experiments that involve high technical challenges and require higher hardware resources to perform the simulation experiments.
2. Cloud resources were an excellent solution for performing advanced computational simulation experiments that involved high technical challenges and required higher hardware resources.
3. There are different high-end commercial software applications which can be used to perform virtual simulation. CFD Support's OpenFOAM with HPC UberCloud Containers helped us solve this problem with minimal effort in setting up the model and performing the simulation trials.
4. The combination of EGI/CESNET Cloud, UberCloud Containers, and CFD Support's OpenFOAM helped in speeding up the simulation trials and allowed us to complete the project within the stipulated time frame.

---

*Case Study Author - Praveen Bhat*