The UberCloud Experiments

Technical Computing in the Cloud

Running your code on remote HPC computing resources

OpenFOAM

With Support From:

http://www.TheUberCloud.com

June 10, 2015
Welcome!

The UberCloud* Experiment started in July 2012, with a discussion about cloud adoption in technical computing and a list of technical and cloud computing challenges and potential solutions. We decided to explore these challenges further, hands-on, and the idea of the UberCloud Experiment was born, also due to the excellent support from INTEL generously sponsoring these experiments!

We found that especially small and medium enterprises in digital manufacturing would strongly benefit from technical computing in the cloud. By gaining access from their desktop workstations on demand to additional remote compute resources, their major benefits are: the agility gained by shortening product design cycles through shorter simulation run times; the superior quality achieved by simulating more sophisticated geometries or physics; and by running many more iterations to look for the best product design. These are benefits that increase a company’s competitiveness.

Tangible benefits like these make technical computing - and more specifically technical computing as a service in the cloud - very attractive. But how far away are we from an ideal cloud model for engineers and scientists? In the beginning, we didn’t know. We were just facing challenges like security, privacy, and trust; conservative software licensing models; slow data transfer; uncertain cost & ROI; availability of best suited resources; and lack of standardization, transparency, and cloud expertise.

However, in the course of this experiment, as we followed each of the 170+ teams closely and monitored their challenges and progress, we’ve got an excellent insight into these roadblocks, how our teams have tackled them, and how we are now able to reduce or even fully resolve them. In this Compendium we have selected five case studies solely dealing with the computational fluid dynamics software OpenFOAM**. OpenFOAM users have asked us to perform experiments together with cloud experts and cloud providers, and at the end of each experiment every team wrote a case study with lessons learned and recommendations which we have collected in this compendium.

Now, enjoy reading!

Wolfgang Gentzsch & Burak Yenier
The UberCloud, June 2015

*) UberCloud is the online community and marketplace where engineers and scientists discover, try, and buy Computing Power as a Service, on demand. Engineers and scientists can explore and discuss how to use this computing power to solve their demanding problems, and to identify the roadblocks and solutions, with a crowd-sourcing approach, jointly with our engineering and scientific community. Learn more about the UberCloud at: http://www.TheUberCloud.com.

**) None of the OpenFOAM® related products and services are approved or endorsed by OpenCFD Ltd. (ESI Group), producer of the OpenFOAM software and owner of the OpenFOAM® and OpenCFD® trade mark.

Please contact UberCloud help@theubercloud.com before distributing this material in part or in full. © Copyright 2015 UberCloud™. UberCloud is a trademark of TheUberCloud Inc.
The UberCloud Experiment Sponsors

We are very grateful to our Compendium sponsor Intel, and to our sponsors ANSYS, Autodesk, Bull, CloudFlow, Microsoft Azure, Nephoscale, and NICE DCV. Their sponsorship allows for building a sustainable and reliable UberCloud community platform:

Big Thanks also to our media sponsors HPCwire, Desktop Engineering, Bio-IT World, HPC Today, scientific computing world, insideHPC, Primeur Magazine, BRC, and Engineering.com for their the widest distribution of this UberCloud Compendium of case studies in the cloud:
Case Studies: OpenFOAM in the Cloud

Team 4

Simulation of Jet Mixing in the Supersonic Flow with Shock

“It (remote visualization) provides exciting possibilities for remote, very large data management, limiting the unpleasant (and unavoidable) remote-rendering delay effect.”

MEET THE TEAM

End-user: Science and Technology Corporation (S&T) in The Netherlands specializes in project implementations and product developments where science and technology plays an important role. Services include: Delivering short-term and long-term staff for your projects; delivering reliability software solutions; and developing software products. The S&T team of professionals consists almost exclusively of PhD and MSc engineers who combine a high level of systems engineering with their science skills.

Software: We used an in-house code developed in Fortran, the open source library OpenFOAM, and the commercial code StarCCM+.

Cloud resource provider: Claudio Arlandini is with CILEA (Consorzio Interuniversitario Lombardo per l'Elaborazione Automatica).

HPC/CAE expert: Ferry Tap, founder of Dacolt, an organization providing solutions and consulting in combustion and CFD.

USE CASE

The hardware platform consisted of a single desktop node, Ubuntu-10.04 64 bit, 8 GB RAM, 5.7 TB RAID storage. Currently available expertise: Two PhD research scientists with industrial-level CFD expertise, and a professor in fluid dynamics.

Benchmarking the OpenFOAM solver against an in-house FORTRAN code for supersonic CFD applications on remote HPC resources included the following steps:

1. Test OpenFOAM solver (sonicFoam) on a 2D case at the same conditions as in an in-house simulation
2. Test OpenFOAM solver with dynamic mesh refinement (sonicDyMFoam) on the same 2D case – a series of simulation to be performed to find suitable refinement parameters for acceptable accuracy and mesh size
3. Production simulations with dynamic mesh refinement, which could be a 3D case or a series of 2D simulations with different parameters.
The total estimate for resources was 1,120 CPU hours and 320 GB of disk space.

**CHALLENGES**

Generally, the web interface provided by CILEA was pretty convenient to run the jobs, although some extra efforts were required to download the results. Both the traditional approach (secure shell access) and web interface were used to handle simulations. The complete workflow included:

- Create test case on the end-user desktop
- Upload it to CILEA computing resource through ssh
- Run the case using the web interface
- Receive email notification when the case is finished
- Download the results through ssh
- Post-process the results on the end-user desktop

Direct access is beneficial for transferring large amount of data, providing a noticeable advantage when using the "rsync" utility. In fact, it might be desirable to run the jobs from the command line as well, although this may just be a matter of habit.

An alternative means of accessing remote data offered by CILEA is remote visualization. This approach receives maximum benefit from the relevant HPC facilities where the remote visualization system is sitting on top of a 512 GB RAM node plus video-image compression tools. It provides exciting possibilities for remote, very large data management, limiting the unpleasant (and unavoidable) remote-rendering delay effect.

**CONCLUSIONS**

The simulations were not completed beyond step 1 due to unexpected numerical problems and the time spent investigating these problems. Approximately 4 CPU hours were used in total.

Because the initial test program runs were not completed during this round of the experiment, both the end-user and the resource provider indicated they would like to participate in the second round. Also, the end-user was interested in evaluating the GPU capabilities of OpenFOAM.

**Case Study Authors:** Ferry Tap and Claudio Arlandini
Team 52

High-Resolution Computer Simulations of Blow-off in Combustion Systems

“Remote clusters allow small companies to conduct simulations that were previously only possible for large companies and government labs.”

MEET THE TEAM

End user: Combustion Science & Engineering, Inc. (CSE, USA)
For more than fourteen years, Combustion Science & Engineering, Inc. (CSE) has been dedicated to the study, advancement, and application of combustion and fire science.
Cloud resource provider: Bull extreme factory (Bull XF, France)
Software provider: ESI (with OpenFOAM)
HPC expert: Dacolt (Netherlands)

USE CASE

The undesired blow-off of turbulent flames in combustion devices can be a very serious safety hazard. Hence, it is of interest to study how flames blow off. Simulations offer an attractive way to do this. However, due to the multi-scale nature of turbulent flames, and the fact that the simulations are unsteady, these simulations require significant computer resources. making the use of large, remote computational resources extremely useful. In this project, a canonical test problem of a turbulent premixed flame is simulated with OpenFOAM and run in extremefactory.com.

Application software requirements
OpenFOAM can be downloaded from: http://www.openfoam.org/download/. OpenFOAM can handle this problem very well.

Custom code or configuration of end-user
OpenFOAM input files are available at http://dl.dropbox.com/u/103340324/3dCoarse_125.tar.gz. These files were used in a 3D simulation that ran OpenFOAM (reactingFOAM to be precise) in 40
cores. To get an idea of how to run these files, have a look at the section "Run in parallel" in: https://sites.google.com/site/estebandgj/openfoam-training

**Computing resource requirements:** At least 40 cores.

![Figure 1](image1.png)

**Figure 1** – Schematic of the bluff-body flame holder experiment. Sketch of the Volvo case: A premixed mixture of air and propane enters the left of a plane rectangular channel. A triangular cylinder is located at the center of the channel and serves as a flame holder.

![Figure 2](image2.png)

**Figure 2** – Predicted temperature contours field for the Volvo case using OpenFOAM.

**CHALLENGES**

The current challenges for the end-user (with just his in-house resources is that the computational resources needed for these simulations are significant (i.e. more than 100 cpus and 1-3 days of continuous running).

**BENEFITS**

Remote clusters allow small companies to conduct simulations that were previously only possible for large companies and government labs.

**CONCLUSIONS**

Running reactingFOAM for a simulation of a bluff-body-stabilized premixed flame requires a mesh of less than 1/4 million cells. This is not much, but the simulations need to run for a long time, and are part of a parametric study that needs more than 100 combinations of parameters. Running one or two huge simulations is not the goal here.

The web interface was easy to use – so much easier than running in Amazon's EC2, that I did not even read the instructions and was able to properly run OpenFOAM. Nonetheless, it was not very
clear how to download all the data once the simulation ended. In addition the simulation ran satisfactorily. There were some errors at the end, but these were expected.

The team has one suggestion: A key advantage of using OpenFOAM is that it allows us to tailor OpenFOAM applications to different problems. This requires making some changes in the code and compiling with wmake, something that can be done in Amazon's EC2. It is not clear how this can done with the present interface. A future test might be to run myReactingFOAM instead of reactingFOAM.

Case Study Author: Ferry Tap
Simulating Wind Tunnel Flow Around Bicycle and Rider

“Being able to quickly adapt a solution to a certain environment is a key competitiveness factor in the cloud-based CAE arena.”

MEET THE TEAM

End User: Mio Suzuki, Analysis Engineer (CFD, Wind Tunnel analysis) with Trek Bicycle Corporation. Trek is a bicycle manufacturer with approximately 1,800 employees worldwide, whose mission is to build the best bikes in the world.

Software provider and HPC expert: Mihai Pruna, Software Engineer with CADNexus, developing CAD to CAE interoperability tools based on the CAPRI platform and connecting to OpenFOAM. CADNexus is a global provider of interoperability software for collaborative multi-CAD & CAE environments.

Cloud resource provider: Kevin Van Workum, PhD, Chief Technical Officer at Sabalcore Computing Inc. Sabalcore Computing has been an industry leader in HPC On-Demand services since 2000.

USE CASE

The CAPRI to OpenFOAM Connector and the Sabalcore HPC Cloud Infrastructure were used to analyze the airflow around bicycle design iterations from Trek Bicycle. The goal was to establish a great synergy between iterative CAD design, CFD analysis, and HPC Cloud Environments.

Trek has been heavily invested in engineering R&D, and does extensive prototyping before producing a final production design. CAE has been an integral part of design process in accelerating the pace of R&D and rapidly increasing the number of design iterations. Advanced CAE capabilities have helped Trek reduce cost and keep up with the demanding product development time necessary to stay competitive.

Automating iterative design changes in Computer Aided Design (CAD) models coupled with Computational Fluid Dynamics (CFD) simulations can significantly enhance the productivity of engineers and enable them to make better decisions in order to achieve optimal product designs. Using a cloud-based or On-Demand solution to meet the HPC requirements of computationally
intensive applications decreases the turn-around time in iterative design scenarios and reduces the overall cost of the design.

With most of the software available today, the process of importing CAD models into CAE tools, and executing a simulation workflow requires years of experience and remains, for the most part, a human-intensive task. Coupling parametric CAD systems with analysis tools to ensure reliable automation also presents significant interoperability challenges.

The upfront and ongoing costs of purchasing a high performance computing system are often underestimated. As most companies’ HPC needs fluctuate, it’s difficult to adequately size a system. Inevitably, this means resources will be idle for many hours and, at other times, will be inadequate for a project’s requirements. In addition, as servers age and more advanced hardware becomes available, companies may recognize a performance gap between themselves and their competitors.

Beyond the price of the hardware itself, a large computer cluster demands specialized power resources, consumes vast amounts of electrical power, and requires specialized cooling systems, valuable floor space and experienced experts to maintain and manage them. Using a HPC provider in the cloud overcomes these challenges in a cost effective, pay-per-use model.

**Experiment Development**

The experiment was defined as an iterative analysis of the performance of a bike. Mio Suzuki at Trek, the end user, supplied the CAD model. The analysis was performed on two Sabalcore provided cluster accounts. The CADNexus CFD connector, an iterative preprocessor, was used to generate OpenFOAM cases using the SolidWorks CAD model as geometry. A custom version of the CAPRI-CAE interface, in the form of an Excel spreadsheet, was delivered to the end user by the team expert Mihai Pruna, who represented the Software Provider, CADNexus.

The CAE interface was modified to allow for the deployment and execution of OpenFOAM® cases on Sabalcore cluster machines. Mihai Pruna also ran test simulations and provided advice in setting up the CAD model for tessellation, that is, the generation of an STL file suitable for meshing (Figure 1: CAD Model Tessellation prior to Meshing).

The cluster environment was set up by Kevin Van Workum with Sabalcore, allowing for rapid and frequent access to the cluster accounts via SSH as needed by the automation involved in copying and executing the OpenFOAM cases.

![CAD Model Tessellation prior to Meshing](image)
The provided bicycle was tested at two speeds: 10 and 15 mph. The CADNexus CFD connector was used to generate cutting planes and wake velocity linear plots. In addition, the full simulation results were archived and provided to the end user for review using ParaView, a free tool (see top figure).

ParaView or other graphical post-processing applications can also be run directly on Sabalcore using their accelerated Remote Graphical Display capability.

Thanks to the modular design of the CAPRI powered OpenFOAM Connector and the flexible environment provided by Sabalcore Computing, integration of the software and HPC provider resources was quite simple.

**CHALLENGES**

**General**
Considering the interoperability required between several technologies, the set up went fairly smoothly. The CAPRI-CAE interface had to be modified to work with an HPC cluster. The production version was designed to work with discrete local or cloud based Ubuntu Linux machines. For the cluster environment, some programmatically generated scripts had to be changed to send jobs to a solver queue rather than execute the OpenFOAM utilities directly.

The CAD model was not a native SolidWorks project but rather a series of imported bodies, and surfaces exhibited topological errors that were picked up by the CAPRI middleware. Defeaturing in SolidWorks, as well as turning off certain consistency checks in CAPRI, helped alleviate these issues and produce quality tessellations.

**Data Transfer Issues**
Sometimes, a certain OpenFOAM dictionary would fail to copy to the client, causing the OpenFOAM scripts to fail. This issue has not been resolved at this time, but it seems to occur only with large geometry files, although it is not the geometry file that fails to copy. Possible solutions include zipping up each case and sending it as a single file.

Retrieving the full results can take a long time. Solutions already developed involve doing some of the post processing on the client and retrieving only simulation results data specified by the user, as implemented by CADNexus in the Excel based CAPRI-CAE interface (Figure 2: Z=0 Velocity Color Plot Generated with CADNExus Visualizer Lightweight Postprocessor), or running ParaView directly on the cluster, as implemented by Sabalcore.

**End User's Perspective**
Capri is a fantastic tool to connect the end user desktop environment directly to a remote cluster. As an end user, the first challenge I faced was thoroughly understanding the formatting of the Excel sheet. As soon as I was able to identify what was wrong with my Excel entries, the rest of the workflow went relatively smoothly and as exactly specified in the templates’ workflow. I also experienced slowness in building up the cases and running the cases. If there is a way to increase the speed at each step (synchronizing the CAD, generating cases on the server, and running), that would enhance the user experience.

**BENEFITS**

The CAPRI-CAE Connector and the CAPRI-FOAM connector dramatically simplify the generation of design-analysis iterations. The user has a lot fewer inputs to fill in, and the rest are generated automatically. The end user does not need to be proficient in OpenFOAM or Linux.
With respect to the HPC resource provider, the environment provided to the user by Sabalcore was already setup to run OpenFOAM, which helped speedup the process of integrating the CADNexus OpenFOAM connector with Sabalcore’s services. The only required modification to the HPC environment made by Sabalcore was to allow a greater than normal number of SSH connections from the user, which was required by the software. With Sabalcore's flexible environment, these changes were easily realized.

CONCLUSIONS AND RECOMMENDATIONS

Among the lessons learned in the course of this project were:

- Being able to quickly adapt a solution to a certain environment is a key competitiveness factor in the cloud-based CAE arena.
- A modular approach when developing your CAE solution for HPC / Cloud deployment helps speed up the process of adapting your solution to a new provider.
- Selecting an HPC resource provider that has a flexible environment is also vital to quickly deploying a custom CAE solution.

From an end user perspective, we observed that each cluster provider has a unique way of bringing the cloud HPC option to the end user. Many of them seem to be very flexible with respect to the services and interface they provide based on the user’s preference. When choosing a cloud cluster service, we suggest that a CAE engineer investigate and select the service that is most suitable for the organization’s particular engineering needs.

Case Study Authors: Mihai Pruna, Mio Suzuki, and Kevin Van Workum
MEET THE TEAM

End-user and software provider: Ries Bouwman, ESI. ESI provided input for a typical building HVAC (heating, ventilation, and air conditioning) simulation. Software used is the open source CFD toolbox OpenFOAM 2.2.1.

Cloud resource provider: Jerry Dixon, OCF. OCF’s enCORE HPC on-demand service utilizes the Hartree Centre’s Blue Wonder IBM iDataPlex Linux cluster.

HPC expert: Ehsan Eslamian, PhD candidate at Anglia Ruskin University.

USE CASE

With ever greater building projects underway worldwide, having a reliable HVAC (heating, ventilation, and air conditioning) simulation tool becomes increasingly important, as does the mesh and therefore the (transient) simulation. We decided to evaluate how well OpenFOAM and HPC can help with the simulation of an internal building flow. To start, we chose to investigate just a transient buoyant flow. Perhaps later we will look into CHT (cooling through cold water in concrete) and radiation (sun). However, the real goal of the project was to find out how easy using HPC through the cloud is for SMEs in terms of accessibility, usability, user friendliness and other factors.

“Having access to an on-demand HPC capability might overcome the barrier of investing heavily in CFD resources...”
The use case is a simple room with an adiabatic floor and four walls. The ceiling is a so-called “active” ceiling, with three pipes running through it. The pipes can contain warm or cold water for active heating or cooling of the room.

**CHALLENGES**

The built environment is only just beginning to appreciate the advantages of CFD over classical “rule-of-thumb” approaches. The ever more complex and larger buildings require a detailed in-depth knowledge of airflow around and inside the building.

Having access to an on-demand HPC capability might overcome the barrier of investing heavily in CFD resources, including expert personnel, hardware and software.

**End user challenges**

- Setup of Conjugate Heat Transfer (CHT) problem for CFD novices. In its current form, a CHT simulation setup in OpenFOAM is very complex. It starts with creating a multi-region mesh (fluid and solid zones) with uniform interfaces and goes on to define all boundary conditions. ESI-OpenCFD is working on improving the usability for smooth setup and running of CHT cases.

**Resource provider challenges**

- A key challenge is to meet the requirements of a diverse set of users who are using the same system, often in quite different ways. This particular project has presented no real issues, the environment for the end user worked perfectly from day 1 and there have been no problems with upload and download of data files etc.

- Delivering excellent service and meeting user expectations in a multi-tenanted environment is a challenge: the expectations of the user on this project have been met and job scheduling has not been a problem. This project has run very smoothly.

- Providing the service at an economically attractive rate while also ensuring there is a return on both the investment and the operating costs.

**BENEFITS**

**For the resource provider**

- Permits us to increase utilisation of a valuable capital asset and gain a financial return that will support renewal of the HPC cluster at a later date.

- Provides an excellent case study for the use of a remote HPC cluster which can be used in marketing of the service.

- Provides exposure to both OCF and the Hartree Centre. The aims of the Hartree Centre are primarily to promote the adoption of HPC within UK industry, so simplified access to the technology via an on-demand model perfectly meets this objective. Both OCF and the Hartree Centre are engaged with organisations across a range of industry sectors who have seen the benefits of HPC and are either embarking on a plan to adopt its use, or are expanding the use of HPC to develop products and services more quickly, to more exacting standards and of course to enhance profits. The Hartree Centre is working closely with a number of businesses in the areas of R&D collaboration, software development and general technical support in relation to HPC projects.
Case Studies: OpenFOAM in the Cloud

**For the end user**
- Flexible cost control.
- Environment that does not require a CFD expert.

**For the software provider**
- Shift from B2B to B2C business and access to new verticals.

**For the team expert**
- Expanding my knowledge of HPC in cloud and CFD.
- Learning to work with high end organisations.
- Learning time management.

**CONCLUSION AND RECOMMENDATIONS**

We have encountered some delays in creating a smoother process in OpenFOAM. We are continuing our work in Round 4 and will provide conclusions and recommendations when the project is finished.

Case Study Author: Ehsan Eslamian
MEET THE TEAM

End User: Mio Suzuki, Trek Bicycle Corporation
Team Expert: Matthew Lycke, Ciespace
Software provider: Ciespace CAE Cloud Platform with OpenFOAM
Cloud hardware provider: Ciespace

USE CASE

In this study, Ciespace CFD (with OpenFOAM) was used to simulate flow around a stationary bicycle in a wind tunnel setting. The bicycle CAD was generated by the end user (Mio) using Solidworks. The exported Parasolid geometry was then uploaded to Ciespace’s Cloud.

Ciespace’s CAE Cloud Platform has an Open source CFD application that is an end-to-end cloud-based. Ciespace CFD is a unique cloud-based CFD software in which the traditional CAE environment runs in the cloud. While the production version runs on AWS, the entire software stack is cloud neutral and can run on private clouds within an enterprise as well. Since all geometry, meshing, solving and results visualization is done in the cloud, no large file upload/download is necessary. Such CAE structure is convenient not only for streamlining the problem setup and iterations, but also its high portability brings additional value to the customers. The web-based portability eliminates the need for CAE engineers to remotely log-in to their office computer via VPN. For engineers with frequent remote assignments, this could be a great asset.

One of the biggest hurdles that deter many interests of end users from using HPC Cloud is software licensing. Ciespace CFD cloud integrated structure removes the problem of purchasing an additional, special license for off-site and distributed computing. Under the Ciespace model, CAE engineers are also spared from keeping up with routine software version updates and hardware maintenance.

Project workflow/end user perspective:
Once the user logs in to the online Ciespace CFD account, they are directed to an initial page called Dashboard. Dashboard is where users can view the existing projects and tasks at the macro level. The dashboard also hosts series of example cases for beginners that serve as tutorials.
The dashboard houses a space/folder where all uploaded CAD models are stored. When a user chooses to create a new project, they can pull the geometry into the project from this model folder. After importing the part, Import Model node is automatically created for viewing the geometry. The rendering speed is quite fast. The software runs smoothly over the web such that it gives the user a sense of using an application as if it were on their local machine. Ciespace CFD’s GUI is attractive and practically laid out. The model set up, mesh, solving, and post-processing processes are arranged in the horizontal manner at the bottom of the screen. Feature control panel is located on the left side of the screen on which one can input necessary parameters to complete the simulation setup.

Progressing from the part set up to mesh creation and solver set up is intuitive. For each step, associated visual is created in the main screen. These visuals are reassuring way to confirm that the intended operations are producing the desired effect. What is particularly useful in the software is the ability to copy and create a secondary (and tertiary and so on) study trees in the workflow. This is a convenient way to evaluate various scenarios within one project.
CHALLENGES

While many features were intuitive to use, there were some challenges. At the time of this project, the process of establishing and assigning boundaries suffered from the software not saving the specified setting, thus requiring the user to keep entering the same information repetitively. The meshing also took longer than anticipated. Both these problems are related and attributed to the Surface Wrapper capabilities being newly implemented in that particular version of the software at that time as well as to the end user’s lack of experience.

The only other challenge was encountered when a solver stopped responding. Matt (team expert) quickly responded to resolve the task. The download option for Solver output does exist but being an Open source software, OpenFOAM output is not very clear to the new user.

CONCLUSIONS

Benefit of using this software:
- Portability gives complete freedom from operating the software on a location-bound machine(s)
- Ciespace CFD’s structure eliminates the need to deal with licensing structure in order to solve parallel on a large scale cloud HPC
- Modern and intuitively laid out GUI encourages efficient simulation workflow
- HPC ready – the integration of the software with the cloud HPC cluster eliminates the need for transferring the files, which often is a time consuming step in the traditional CAE environment

Desired features:
- More robustness in the surface wrapper to improve the downstream meshing time
- Options for higher physics model. At the time of using the software (January, 2014), the choices for physics model with OpenFOAM seemed to be rather limited.
- Expanded post processing options /capabilities comparable to ANSYS/StarCCM+ are desired.

Case Study Authors: Mio Suzuki and Matthew Lycke
Aerodynamic Study of Airfoil

“The combination of OpenFOAM & UberCloud Containers enables efficient, effective, and easy performance of complex engineering simulations.”

MEET THE TEAM
End-user/CFD expert: Praveen Bhat, Technology Consultant, INDIA
Software provider: ESI Group / OpenFOAM
Cloud resource provider: NephoScale
HPC expert: Burak Yenier, Co-Founder, CEO, UberCloud.

USE CASE

The aerodynamic study on the 2D airfoil is performed with the incompressible air flow around a 2D airfoil. The model setup includes the geometry preparation where a selected region is model that represents the surrounding air volume with the airfoil profile at the center. The airfoil profile needs to be accurately modeled to capture the variation in the airflow pattern around the airfoil. The model setup is done using the open source software OpenFOAM. The OpenFOAM software is embedded in an UberCloud Container located in the NephoScale cloud facility. The main objective of this project is to experience the ease-of-use of the UberCloud OpenFOAM container and to evaluate
the HPC performance with respect to the accuracy of result prediction and also with respect to the solution time and resource utilization.

![Mesh model for the aerofoil](image)

**Figure 3: Mesh model for the aerofoil**

**Process Overview**

Meshing density is very fine around the airfoil and along the path of the trailing edge. The meshes were modeled coarser as it moves away from the airfoil and the coarseness of the mesh increases near the air volume boundary (air inlet and outlet). The following describe the setup steps:

1. The Finite Element mesh model is generated followed by the fluid properties definition. The entire volume surrounding the airfoil is air which is considered as 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 will be 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 is defining the model boundary conditions and assigning the pressure and velocity initial values. The boundary conditions are assigned where in the airfoil edges are considered as wall. The three sides of the air volume are considered as inlet and the edge following the trailing edge of airfoil is considered as air outlet.
5. The next step in the model development is setting up of the solution algorithm. The problem is solved as steady state and the OpenFOAM solver used for solving this problem is 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 1.0x10^-5. The above parameters define the convergence criteria of the model.
6. The OpenFOAM model developed is then modified for parallel processing where the existing model is divided according to the number of HPC computing nodes.
7. The model is solved in parallel and once the solution is converged, the solved model in the parallel processors is 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 is surrounded by air volume and the variation in the flow velocity and air pressure near the airfoil section is reviewed. The different plots below show the flow of air and laminar behaviour observed in the airfoil region.
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 and the high pressure section near the airfoil creates a lift forces on the airfoil. The lift on the air plane wing can be considered to be Newton’s third law where in there will be a reaction force in the form of downward force on the air. The life on the airplane wing should be consistent since it is 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 analyzed by determining the pressure co-efficient 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 of attack.

The behavior of air flow will be turbulent in the air volume, and the transition of the air behavior from turbulent to laminar is observed in the air volume nearing the airfoil section and the flow behavior of air will be laminar around the wall of the airfoil. The airflow path it follows near the wall boundary of the airfoil is laminar which is evident from Figures 5&6. The vector flow path of the air in the airfoil region is also represented where 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 2.2 with OpenFoam MPI and Paraview. The model is evaluated for the accuracy of prediction of air flow around the airfoil, with both fine and coarse mesh. The time required for solving the model with different meshes is 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 the models.
Figure 8: Solution time required in a 4 core configuration

Figure 9: Solution time required in a 32 Core HPC configuration
Figure 10: Solution time for a model with 2M elements solved using different HPC core configurations

Figure 11: 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 monitoring/administration of host servers & 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 effort and will benefit the future users.

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

**CHALLENGES**

The project started testing the installation of OpenFOAM 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 were related to technical complexity. This involved accurate prediction of flow behaviour around airfoil which is achieved through defining appropriate element size to the mesh model. The finer the mesh the higher is the simulation runtime required and hence the challenge was to perform the simulation within the stipulated timeline.

**BENEFITS**

1. The HPC cloud computing environment with OpenFOAM & Paraview made the process of model generation easier with process time reduced drastically along with result viewing & 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-trails or resubmission of the same simulation runs.
3. The computation requirement for a very fine mesh (2 million cells) is high, which is near to impossible to achieve on a normal workstation. The HPC cloud provided this feasibility to solve very fine mesh models and the simulation time drastically reduced thereby providing an advantage of getting the simulation results within acceptable run time (~30 min).
4. The UberCloud experiments in the HPC Cloud showed the possibility and gave extra confidence in the setup and run of the simulations remotely in the cloud. The different simulation setup tools were installed in the UberCloud Container and 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, and the regular UberCloud auto-update module through 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 on Nephoscale 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 a very good fit for performing advanced computational experiments that involve high technical challenges and require higher hardware resources to perform the simulation experiments.
3. There are different high-end commercial software applications which can be used to perform virtual simulation. Open source OpenFOAM with HPC UberCloud Containers helped us to solve this problem with minimal effort in setting up the model and performing the simulation trials.

4. The combination of HPC Cloud, UberCloud Containers, and OpenFOAM helped in speeding up the simulation trials and also completed the project within the stipulated time frame.

APPENDIX: UberCloud Containers: Brief Introduction

UberCloud Containers are ready-to-execute packages of software. These packages are designed to deliver the tools that an engineer needs to complete his task in hand. The ISV or Open Source tools are pre-installed, configured, and tested, and are running on bare metal, without loss of performance. They are ready to execute, literally in an instant with no need to install software, deal with complex OS commands, or configure. The UberCloud Container technology allows wide variety and selection for the engineers because they are portable from server to server, Cloud to Cloud. The Cloud operators or IT departments no longer need to limit the variety, since they no longer have to install, tune and maintain the underlying software. They can rely on the UberCloud Containers to cut through this complexity. This technology also provides hardware abstraction, where the container is not tightly coupled with the server (the container and the software inside isn’t installed on the server in the traditional sense). Abstraction between the hardware and software stacks provides the ease of use and agility that bare metal environments lack.

Case Study Author: Praveen Bhat

Disclosure: None of the OpenFOAM® related products and services offered by UberCloud are approved or endorsed by OpenCFD Ltd. (ESI Group), producer of the OpenFOAM software and owner of the OpenFOAM® and OpenCFD® trademarks.
Thank you for your interest in the free and voluntary UberCloud Experiment.

If you, as an end-user, would like to participate in this Experiment to explore hands-on the end-to-end process of on-demand Technical Computing as a Service, in the Cloud, for your business then please register at: http://www.theubercloud.com/hpc-experiment/

If you, as a service provider, are interested in promoting your service/product at the UberCloud Exhibit then please register at http://exhibit.hpcexperiment.com/about/join/

1st Compendium of case studies, 2013: https://www.theubercloud.com/ubercloud-compendium-2013/


In any case, if you wish to be informed about the latest developments in technical computing in the cloud, then please register at http://www.theubercloud.com/. It’s free.

Please contact UberCloud at help@theubercloud.com before distributing this material in part or in full.
© Copyright 2015 UberCloud™. UberCloud is a trademark of TheUberCloud Inc.