**Shravankumar Nagapuri**

[**https://projects.skill-lync.com/profiles/Shravankumar-Nagapuri-507**](https://projects.skill-lync.com/profiles/Shravankumar-Nagapuri-507)

**2-4-1137/1, Gokulnagar, Hanamkonda, Warangal, Telangana, India. Pin - 506001 | 9573777508 | nagapurishravan301@gmail.com**

**Objective**

Mechanical Engineer seeking full-time opportunities in CFD firm with special interest in scripting and product simulation applications**.**

**Education**

**Masters certification program in CFD, SKILL-LYNC** (Jul 2019 -Present)

**M.E in Mechanical Engineering,** University of Engineering, Osmania University, Percentage - 70.33(May 2013)

**B.Tech in Mechanical Engineering,** KITS, Huzurabad, India, Percentage - 71.33 (Jun 2009)

**Course Projects**

**Combustion Simulation on the Combustor model using ANSYS Fluent, SKILL-LYNC** (Oct 2019)

* Explained about the possible types of combustion simulations in FLUENT
* A 2D Asymmetry Combustion Model is considered and performed a combustion simulation on the model using ANSYS Fluent and the variation of the mass fraction of the different species (like CO2, H2O, CH4, N2, O2 and NOx) using line probes at different locations are plotted.

**Analysis of Cyclone Separator using ANSYS Fluent, SKILL-LYNC (**Oct 2019)

* A cyclone separator model is considered and Performed an analysis by applying four different boundary condition types at the inlet i.e. reflect, trap, escape and wall-jet and submitted the images for the different conditions
* The number of particles through the inlet is varied and executed the simulation for 3 cases by using the “Reflect” option as the boundary condition for the Discrete Phase, Reported the number of particles that pass through or remain in the separator.

**Rayleigh Taylor instability modelling with air and water using ANSYS Fluent, SKILL-LYNC** (Sep 2019)

* **Performed the Rayleigh Taylor instability simulation for 3 different mesh sizes with the base mesh being 0.5 mm.**
* **Compared the results by showing the animations, and explained why a steady state approach might not be suitable for these type of simulations and calculated theAtwood number**
* **Explained how it affects the behavior of the Stability**

**Conjugate Heat Transfer Analysis on an Exhaust manifold using ANSYS Fluent, SKILL-LYNC** (Sep 2019)

* Exhaust Manifold model considered and performed a CHT analysis on a model of an exhaust manifold.
* Compared the solutions for two sizes of meshes, the first one should be the baseline mesh and the second one should have a finer mesh on the solid.

**Analysis of Fluid Flow over Ahmed Body using ANSYS Fluent, SKILL-LYNC** (Sep 2019)

* Explained the importance of the Ahmed Body
* 3D Ahmed Body consideredand performed a steady state analysison model with a baseline mesh.
* Performed the Grid dependency test on the Ahmed body Model by refining the mesh.
* Compared the velocity profiles for the three setups using a middle cut plane.

**Simulation of a 1D Super-sonic nozzle flow simulation using McCormack Method Using MATLAB, SKILL-LYNC** (Aug 2019)

* Wrote the code to solve 1D supersonic nozzle flow equations using the McCormack method by implementing both conservative and non-conservative forms.
* Wroteseparate functions for conservative and non-conservative forms
* Figured out the minimum number of cycles for which the simulation should be run in order for convergence
* Compared the normalized mass flow rate between the conservative forms and non-conservative forms.

**Steady and Unsteady 2D Heat conduction using MATLAB, SKILL-LYNC** (Aug 2019)

* Wrote a MATLAB code to solve the 2D Transient state Heat conduction equation explicitly.
* Wrote a MATLAB code to solve the 2D Transient state Heat conduction equation implicitly using Iterative techniques (Jacobi,Gauss Seidal,SOR**)**
* Wrote a MATLAB code to solve the 2D Steady state Heat conduction equation implicitly using Iterative techniques (Jacobi,Gauss Seidal,SOR**)**

**Block Mesh Drill down challenge using OpenFOAM, SKILL-LYNC** (Sep 2019)

* Generated the Block Mesh file for the given Geometry using OpenFOAM and use the icoFoam solver to simulate the flow through a backward-facing step.
* Created multiple meshes and compared the results obtained from each mesh.
* The velocity magnitude, pattern change as a function of mesh grading factor shown. Used factors are 0.2, 0.5, 0.8
* Measured the velocity profile at 0.085 m from the inlet of the geometry

# Simulation of Flow through a pipe using OpenFOAM (Sep 2019)

* Wrote a program in MATLAB that generated the computational Block mesh file automatically for any wedge angle and grading schemes, where wedge angle is less than 5 degrees(i.e., 3 degrees) and used front and back BC as wedge
* For a baseline mesh, showed that the velocity profile matches with the Hagen poiseuille's equation
* Showed that the velocity profiles from entry to full developed
* Post process of velocity and shear stress are shown

**Symmetry Vs Wedge Vs HP using OpenFOAM** (Oct 2019)

* Wrote a program in MATLAB that generated the computational Block mesh file automatically for any wedge angle and grading schemes, where wedge angle is less than 5 degrees (i.e., 3 degrees) and used front and back BC as symmetry.
* For a baseline mesh, showed that the velocity profiles and compared it with the simulation of flow through a pipe

challenge results

* Wrote a program in MATLAB that generated the computational Block mesh file automatically for different wedge

angles (i.e., 10, 25,45) and grading schemes and used front and back BC as symmetry

* Showed that the velocity profiles from entry to full developed
* Post process results of velocity and shear stress shown and compared these results with the Hagen poiseuille's equation results.

**Other Projects**

**CFD Analysis of Flow and Heat Transfer through Preswirl System,** Osmania University (May 2013)

* Axisymmetric and three-dimensional (3D) sector CFD model of preswirl system are considered.
* In the 3D sector models, the preswirl nozzles or receiver holes represented as axisymmetric slots so that steady state solutions assumed.
* A number of commonly used turbulence models tested in CFD compared and predicted the results of different models.

**Design of Formula-1 Car & Analysis ofAerodynamic Forces**,KITS, Huzurabad, JNT University (April 2009)

* According to Formula-1 car 2009 Technical Regulations, a scale model of original design is prepared and performed wind tunnel test and analyzed the aerodynamic forces.

**Other Experience**

* **Assistant professor,** Anurag college of Engineering, Aushapur, Hyderabad, Telangana, India (Sep 2014 – Jun 2019)
* **Assistant Professor**, MLR Institute of Technology, Gandimaisamma, Dundigal, Hyderabad (Sep 2013 – Apr 2014)
* **Teaching Assistant,** SR Engineering college(Jan 2009 – Oct 2010)

**FDP/Workshops attended**

* Attended the 2 days workshop on “Advances in Mechanical Engineering” Vidya Jyothi Institute of Technology, Hyderabad.
* Attended the 2 days workshop on “Challenges & Advances in Renewable Energy sources” at Kamala Institute of Technology & science, Huzurabad, Karimnagar.
* Attended the 2 days workshop on “Trends in Mechanical Engineering” at SR Engineering College, Ananthsagar, warangal.
* Attended the 3 days FDP on "Advances in Computer Aided Engineering for Complex Machine Components" at Anurag college of Engineering, Hyderabad.

**Publications**

**Analysis of exhaust gas flow in a Three way Catalytic Converter of a Diesel Automobile using CFD Software,** *Abhilash Udhari, Shravan kumar Nagapuri, June-2017, Global Journal for Research Analysis, International.*

**Software Packages**

* Modelling: Pro-E/Creo, SolidWorks, CATIA V5
* Computational Analysis: ANSYS, MATLAB, OPENFOAM
* Statistical Data Analysis: MS-Excel.