RAJA SELVAMANI

SUMMARY :

7 years industrial experience with expertise in solving practical problems involving heat transfer and fluid mechanics using analytical skills and CFD tools. Passion for understanding underlying physics, addressing design challenges and identifying the best possible engineering solution.

Industrial Experience :

- Identifying the basic heat transfer and fluid flow physics behind the engineering problem under consideration. Using simulation techniques to capture the relevant physics and identSimulation of heat transfer and fluid flow for industrial and automotive power train system applications
- Understanding physics, geometry creation, flow domain extraction, discretizing domain, solver setting and post processing results
- Thermodynamic models for engine simulation and combustion problems
- Numerical coding and application of CFD solver algorithms (SIMPLE) to solve fluid flows problems (academic)
- Creating high quality Hexahedral/Tetrahedral mesh for complex geometries using HyperMesh, Ansys Workbench, ICEM CFD (Tetra and blocking) (experienced in external aerodynamics simulation)
- Flexibility in working with different software for simulation
- Responsible for quality checks, final report preparation and final deliverables
- Organized internal training sessions in thermodynamics and heat transfer
- Strong verbal and written communication skills

EDUCATION

- Masters of Engineering 2010 from Texas A&M University, College Station, TX, U.S.A.
- Bachelor of Engineering (Mechanical), 2007 from PRRM Engineering College, affiliated to J.N.T.U. Hyderabad

SOFTWARE KNOWLEDGE

- Meshing Software : HyperMesh, ICEMCFD, ANSYS WORKBENCH
- CFD Software : ANSYS FLUENT , Star CCM+ (beginner), CFD Post (post processing)
- Design Software : SolidWorks
- Programming : C, Fortran, Matlab

WORK EXPERIENCE

- CFD Senior Engineer, Delphi Technologies, Jan 2018 to till date
- CFD Senior Engineer, CFD – Larsen & Toubro Technology Services, Bangalore, India Nov 2011 to till date
- Assistant Professor Vasavi College of Engineering, Hyderabad, India, from Aug 2010 to Nov 2011

PROJECT MANAGEMENT

- Customer requirements understanding & study, project execution
- Interacting with clients of multi-cultural background to collect input data required to execute the CFD job
- Interpreting the given input geometry and visualizing the fluid domain
- Study the physics of the problem and to generate appropriate mesh, model selection, material properties and boundary conditions
- Responsible for preparation of final deliverable reports and maintain project files and documents

RAJA SELVAMANI

PROJECT HIGHLIGHTS

PROJECTS EXECUTED @ Delphi Technologies (since 2018)

PROJECT: Optimization of Injector (SCR and Fuel injector application)

Software : ANSYS FLUENT, ICEMCFD

Project Description

- Fuel injectors are required to provide high atomization and its performance is highly sensitive to the design parameters.
- Actual spray formation simulation is time consuming as it requires detailed LES modeling and stringent mesh requirements. Team developed a simplified methodology using URANS approach to correlate atomiation efficiency with flow variables. Shorter turnaround time for studying design optimization and manufacturing tolerance effects.
- Meshing was performed using ICEMCFD blocking. Mesh consistency along with necessary quality were produced within quick turnaround time
- Multiphase mixture model used and scheme files for capturing contours of fluid properties at important sectional planes.
- Post proessing of energies that are indicative of atomization performance and validation using experimental results were documented with observations and design suggestions.
- Q criterion, velocity contours overlapped with vectors and streamlines used to describe the swirl and tumble effects and understand the results.

PROJECT: Acoustics noise evaluation of a centrifugal pump

Software : ANSYS FLUENT, ICEMCFD

Project Description

- Project is on going. Meshing and priliminary literature search completed. Initially RANS basaed turbulance model Acoustics models have been studied and appropriate assumptions and their implications gauged. Methodology has been set by the team to evaluate the noise spectrum.
- Preliminary study includes careful understanding on various physics behing the acoustics problem. Turbulence model to be used is scale resolved models which are not as expensive as LES models.
- Sliding mesh approach to be used in order to capture stator and rotor effects accurately.
- Meshing using Ansys workbench and ICEMCFD, interfaces created in Fluent, boundary layers in mesh provided in order to ensure accurate range of y+. Mesh size ensured to be less than 1/10th of the minmum wavelength of interest. And also accounted for y+ on walls of impeller blades.
- Acoustics model to be used is CAA for the source and FWH method for propagation into the far field.

PROJECTS EXECUTED @ L&T TS (2011 – 2017)

PROJECT: HVAC Analysis of buggy floor of a process industry

Software : ANSYS FLUENT, ICEMCFD

Project Description

- Domain consists of buggy floor with flow obstacles like buggies, buggy filling station, HVAC supply and exhaust grills.
- It is required to verify health and safety criteria (HS&E) for the plant by observing the velocity contours and vectors across important sectional planes.
- Two iterations are performed. Baseline to evaluate the existing HVAC layout. And Iteration 01 to implement design changes for the HVAC such that the HS&E criteria are fulfilled.
- Particle tracking performed to evaluate the particle extinction time and to ensure that 90% of it gets removed before 300 seconds.
- Streamlines used to observe flow patterns inside the domain.

PROJECT: Metal spillage simulation inside laboratory

Software : ANSYS FLUENT, ICEMCFD

Project Description

- Domain consists of a laboratory containing cabinets that prepare metal powder
- It is required to design a HVAC system such that the concentration of metal powder due to spillage does not exceed a threshhold value at any point inside the room. This would result in combustion of the metal powder.
- Proposed ventilation system was evaluated using DPM model (as volume fraction is low) Particle interaction with continuous phase along with gravitational effects were enabled.
- After checking results with steady particle injection, it was decided to consider unsteady particle injection for this problem.
- Results showed that the convergence was a problem as the continuity equation for continuous phase was showing diverging behavior.
- Problem was shared with ANSYS technical suppor and final status report shared with customer. (This project is on-going)

PROJECT: Condensation Analysis of Room

Software : ANSYS FLUENT, ICEMCFD

Project Description

- Domain consists of hot water jet flow through nozzles for sanitization
- Water vapour condenses on walls and becomes potential microorganism source
- Hand calculations performed to quantify flashing of water vapour through jet to determine the water vapour evaporation rate

- Extraction fan used to remove water vapour
- CFD simulation performed to study the above problem and evaluate various options for ventilation systems
- Design suggestions provided to increase extraction fan capacity, or decrease nozzle flow rate to avoid flashing and change of location of extraction fan

PROJECT: Engine pressure crank angle simulation

Software : Excel

Project Description

- Excel based calculation of pressure crank angle diagram using the specifications of a given engine
- Air standard cycle approach was used to simplify the physics
- Suction and exhaust pressure were assumed to be equal to charge air system and atmospheric pressure respectively
- Compression and expansion were assumed to be isentropic processes following pV^{γ} = constant
- Combustion process was simulated using Weibe's law for mass fraction burnt
- Thermodynamic first law was applied to conserve heat energy added due to fuel being burnt and energy used by piston
- Inlet valve open and close, Exhaust valve open and close were varied iteratively so that the results match the peak pressure provided in input
- Energy lost due to friction and unburnt fuel was simulated by assuming a factor C (<1), and calculating heat added $Q = m_f$ (LCV)C, C is again varied so that the area under the p-θ curve matches work done calculated by brake power, engine speed and mean effective pressure

PROJECT: Peak Pressure Calculation – CA System of Turbocharger

Software : Excel, ICEMCFD, ANSYS FLUENT

Project Description

- IMO certification for the marine engine requires that the CA system of the diesel engine can withstand the peak pressure generated due to a combustion event in the CA system.
- Combustion problem inside the CA system of turbocharger studied. CA system is closed on engine and turbocharger side for constant volume case. CFD predicted peak pressure used in FEA analysis to determine structural integrity of CA system.
- Thermodynamic hand calculations for calculating adiabatic flame temperature and peak pressure for constant volume premixed combustion (excel sheet created)
- Transient calculations for pressure in constant pressure case. CA system assumed to be a cube with a small outlet. Choked and Non- Choked flow taken into account. Bernoulli's equation used for non choked flow. Speed of sound is assumed for choked flow.
- CFD Simulation methodology established for combustion problems that are turbulence driven (flame propagation and fluid properties contours created

with respect to time – results are in agreement with analytical calculations)
- Species model used with volumetric reactions
- 2 step Arrhenius equation parameters used for the combustion of CH_4 into CO_2 and H_2O
- Point of ignition simulated with high temperature in a small region near spark
- Lamniar finite rate and Eddy dissipation models studied for sample cases and their applications were understood
- Laminar and turbulent flame speed, chemical kinetics and other combustion models for non-premixed combustion were studied

PROJECT: Steady/Transient analysis for pulsed hair removing device

Software : HYPERMESH, ANSYS FLUENT

Project Description

- Device contains a time based pulsed heat source to generate high intensity light which removes skin hair
- Detailed model was created and discretized in Hypermesh
- For steady case, average and r.m.s power was used for the heat source term in the energy equation
- For transient case, a user profile was used for the heat source
- Heat source consists of two flashes for a time period in milli seconds each and zero flash for remaining of the time period. The total cycle repeats in periodic fashion.
- It was observed that the r.m.s power calculated closely matched the transient case.
- High temperatures were noticed in the model, the reason for which was restrictions of flow of cooling air near the filament. Suitable design suggestions were provided to improve the flow
- Temperature plots of various solid components, pathlines for air-flow, velocity and pressure field plots were provided using CFD Post

PROJECT: HVAC analysis for ship

Software : HYPERMESH, ICEMCFD, ANSYS FLUENT

Project Description

- 3D geometry created using 2D drawings. Heat loads like lighting, electronic devices and human occupancy were modelled
- HVAC components like AC inlets, exhaust louvers etc. were modelled
- Humidity was modelled using water vapour as species
- Solar heat gain was taken into account using solar ray tracing method
- ICEMCFD TETRA was used to create a tetrahedral mesh with good quality and required refinment at crucial locations
- Initial solution through P1 Radiation model was calculated and then Discrete Ordinate method used for capturing radiation effects
- Radiation properties were applied according to the material of different surfaces from ASHRAE standards
- Natural convection plumes were examined and optimised by appropriately

changing the opening in ceiling

- PMV, temperature and humidity distribution were plotted inside the domain for different outside atmospheric conditions
- Design suggestions made based on the results

PROJECT: Bottle Assembly Dust cover Analysis

Software : ICEMCFD, ANSYS FLUENT

Project Description

- Dust cover used for bottle assembly line prevent dust from entering into empty bottles. Different models of dust cover were simulated using CFD
- 2D and 3D models were created, discretized and solved. Post processing involves generating velocity , pressure contours and air flow pathlines
- Recirculations were observed in the bottle, volume of fluid flowing into the bottle was calculated and used as a prameter to compare the efficiency of different dust covers
- Additional design iterations were performed using additional components used to ensure positive pressure in the dust cover enclosed region

PROJECTS EXECUTED @ TAMU (2007 – 2010)

NUMERICAL SIMULATION OF SIMPLE ALGORITHM FOR FLOWS OVER BACKWARD FACING STEP AND LID DRIVEN CAVITY PROBLEMS

INTERNSHIP @ CUSP TECHNOLOGY, HYDERABAD INDIA (2007)

NUMERICAL SIMULATION OF SUBSONIC FLOW USING PANEL METHOD ON 2D NACA 4412

ACADEMIC ACHIEVEMENTS

- **First Prize in Ansys Convergence 2014 for CFD "Thermal comfort evaluation of ship cruise"**
- Best paper selected during National level technical symposium **"PRAGNYA 2007" GRIET, Hyderabad, India**
- College Topper 2007, PRRM Engineering College, JNTU

PERSONAL INFORMATION

- Contact: Cell No. +91 8123 561917, 9538 7000 66
- Email - rajaselv@gmail.com

- Languages known : Tamil, English, Telugu, Hindi, Kannada (basic spoken)