GANESH BORDE

MS in AEROSPACE ENGINEER| UNIVERSITY OF TEXAS AT AUSTIN

BS in AERONAUTICAL ENGINEER| VELTECH UNIVERSITY

ganeshyng789@gmail.com

+1 737-326-0702

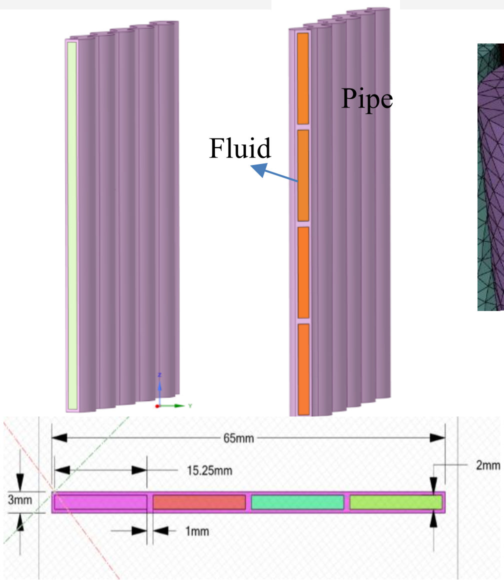
Austin, TX, USA

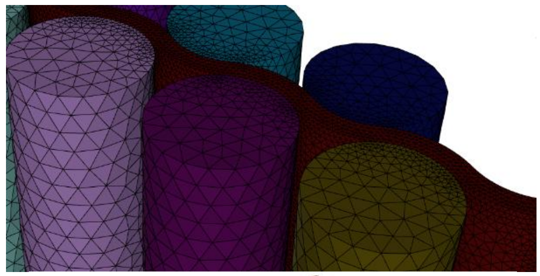
linkedin.com/in/ganesh-borde-928701217

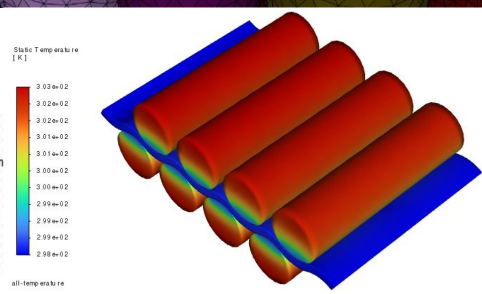
ganesh-borde.github.io/

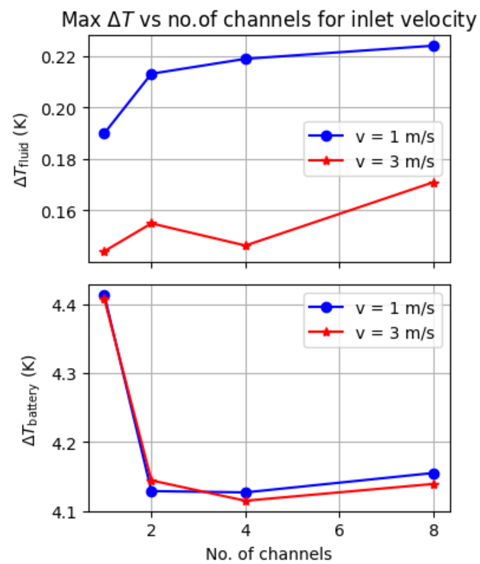
github.com/Ganesh-Borde

18650 Battery Pack Cooling Using Serpentine Pipe









What?

- EV Battery pack cooling using serpentine pipe –used in Tesla
- Heat released @ 4C =74163 W/m^3
- Study effect of single channel and multi channel 2,4, and **8**

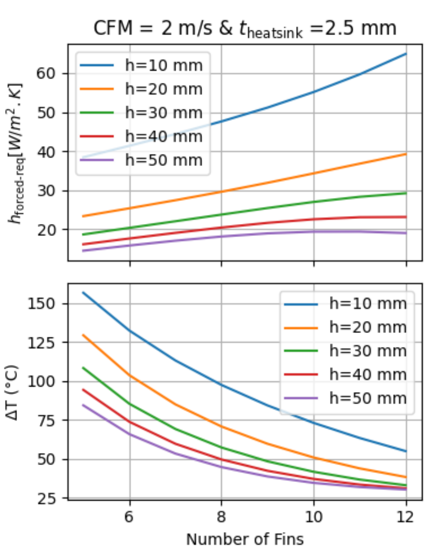
How?

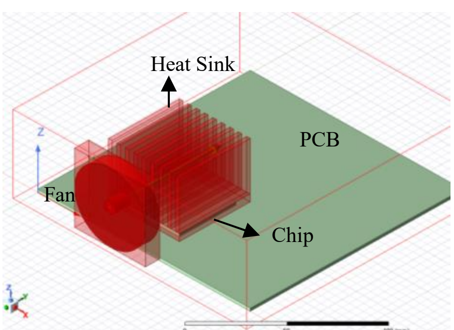
- ANSYS water tight geometry meshing
- Ansys Fluent- Steady state, Conjugate heat transfer, and Multi zone
- Flow is laminar and solving energy
- Multiple Iteration

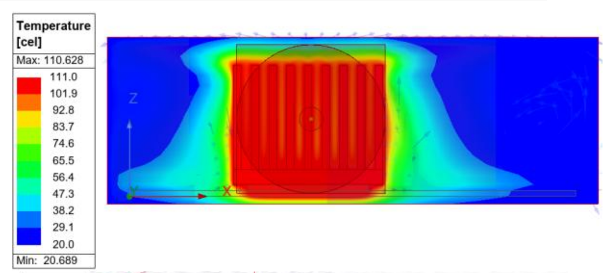
Results

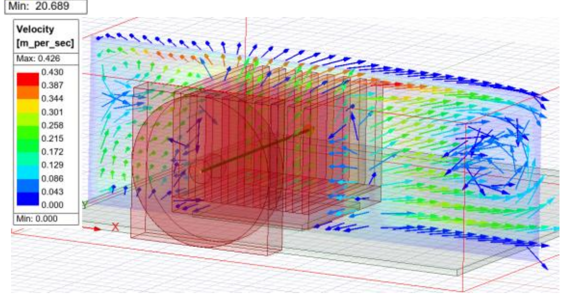
- Using multiple channels decreased MTD (Maximum temperature diff)
- But in the cost of increasing fluid temperature
- Adding more channels reduces overall fluid mass rate, so no change in MTD

Analytical & CFD-FEA Study of Chip Cooling with Optimized Heatsink Design.









What?

- Analytical Thermal Resistance Model (RM) for a chip-heatsink-fan system by using DOE
- $Q_{chip} = 30W, T_{amb} = 20$℃.
- DOE varying thickness and no.of fins to maintain $T_{jun} = 85$℃

How?

- Ansys Icepak –uses Fluent for flow and FEA for structure
- Inbuilt automatic fine structured mesh
- Turbulent, steady state
- PyAEDT for DOE by varying the location of chip and heat sink
- Grille with free area ratio of 0.9

Results

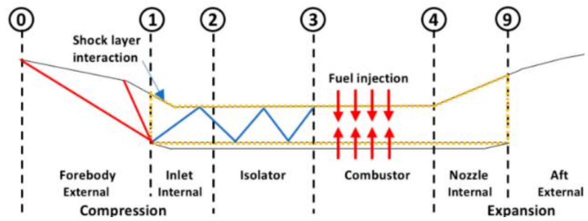
- Thickness and no.of fins have influence on required cooling from analytical model.
- Influence on location of chip-heatsink using PyAEDT Icepak results
- Results from simulations & RM had small differences in temperatures and velocities, due to simplicity in RM.

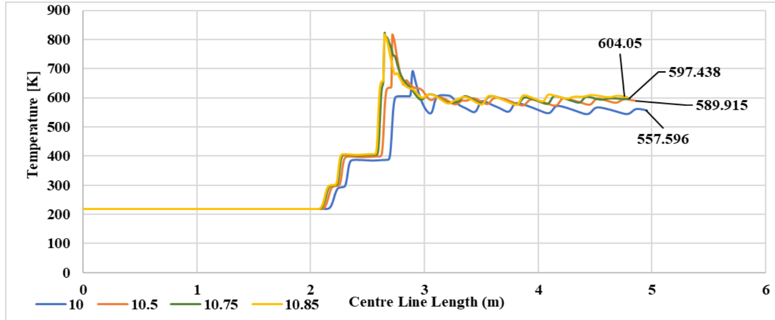
GANESH BORDE

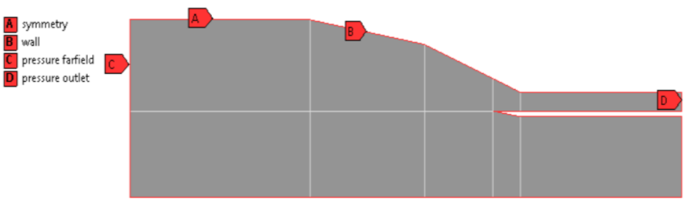
MS in AEROSPACE ENGINEER| UNIVERSITY OF TEXAS AT AUSTIN
BS in AERONAUTICAL ENGINEER| VELTECH UNIVERSITY

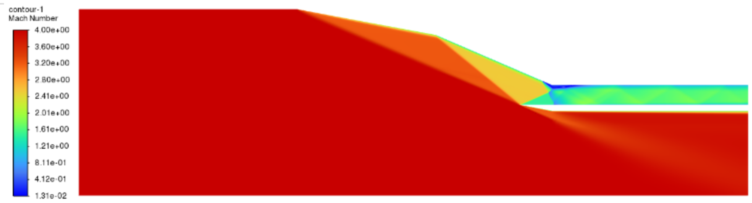
ganeshyng789@gmail.com
+1 737-326-0702
Austin, TX, USA
linkedin.com/in/ganesh-borde-928701217
ganesh-borde.github.io/
github.com/Ganesh-Borde

Design of Scramjet Inlet for Shock on Lip Condition at Low Mach Number by Using CFD









What?

- Engine for high speed flights with only 2 engines, Turbo-Ram and Scramjet
- At low Mach number 4&5 achieving minimum ignition temperature (MIT) by compression is challenging
- Using shock on lip for flow spillage and 2 ramps by varying angles for compression.

How?

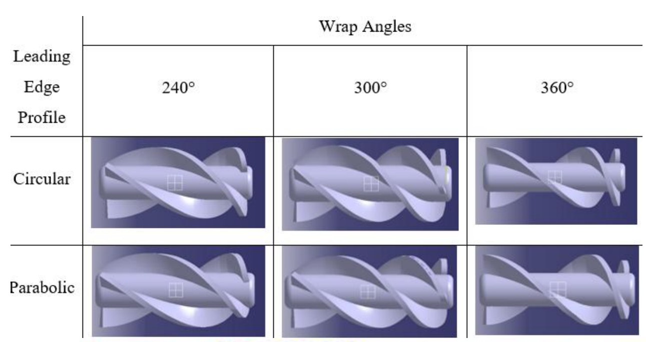
- Using Hypersonic aerodynamics, designed analytical geometry and modeled in Catia.
- Block-Structured mesh using Ansys meshing
- Ansys Fluent- RANS, Steady state, Density-based solver and ideal gas
- The Advection Upstream Splitting Method (AUSM) for shock capture

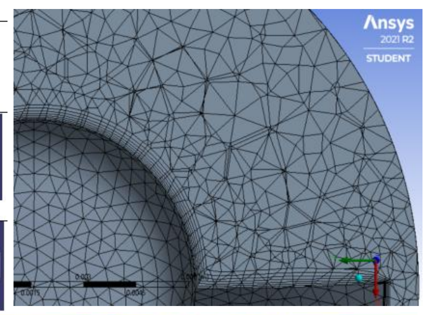
Results

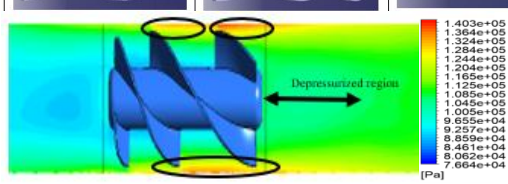
- Initial geometry doesn't give required MIT for JP-7 fuel.
- Varying angles of ramps by different combination got desired MIT for combustion in engine.
- Shock-on-lip observed in all simulations

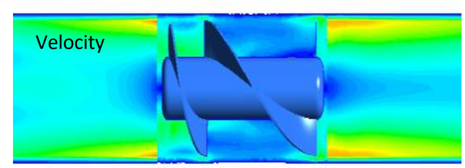
Design and analysis of small-scale axial flow pump impeller using CFD (Ansys CFX)

	Specification	
Pump	Revolution per Minute (RPM)	10000
	Discharge	5[L/Min]
	Working Fluid	Water
	Desired Head	1000[Kg/m³]
	Density of working fluid	1[m]
Blade	Type	Helical
	Total length	43.5[mm]
	Tip Diameter	20[mm]
	Hub Diameter	10[mm]
	Blade Height	5[mm]
	Blade Thickness	1[mm]









What?

- Mini axial-flow pump for small-scale use: chemical dosing, blood pumps, dispensers
- Analyzed 6 impeller designs with:Wrap angles: **240°, 300°, 360°**
- Leading edge shapes: **circular** and **parabolic**

How?

- Designed in **CATIA**, domains split into suction, pump, and delivery.
- **Meshed in ANSYS** with inflation near impeller walls for boundary layer accuracy
- Simulated in **ANSYS CFX**, steady-state, 10000 rpm, 5 L/min flow
- Used **k-ε turbulence model**, sliding interfaces between rotating and stationary domains

Results

- **300° wrap angle + parabolic edge** gave best total head and smooth flow
- Parabolic edges minimized flow separation at leading corners
- **Head increased as discharge decreased**, confirming expected trend
- Improved pressure and velocity distribution observed near blade surfaces

GANESH BORDE

MS in AEROSPACE ENGINEER| UNIVERSITY OF TEXAS AT AUSTIN

BS in AERONAUTICAL ENGINEER| VELTECH UNIVERSITY

ganeshyng789@gmail.com

+1 737-326-0702

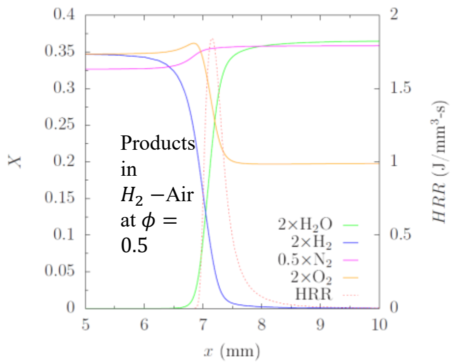
Austin, TX, USA

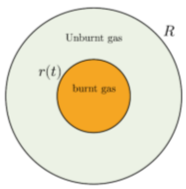
linkedin.com/in/ganesh-borde-928701217

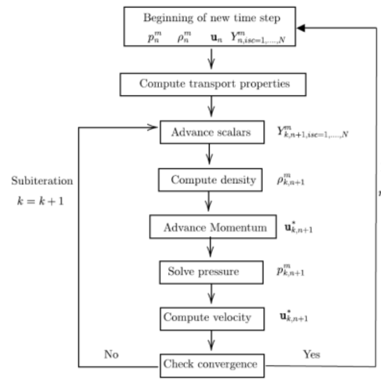
ganesh-borde.github.io/

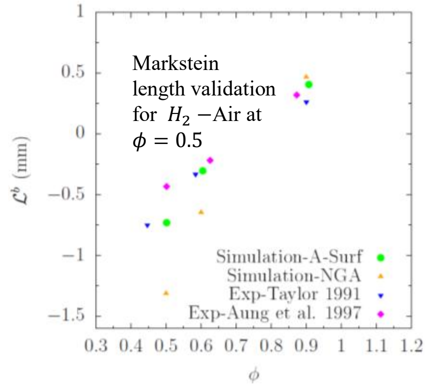
github.com/Ganesh-Borde

Measurement of Flame Properties in CFD Simulated Premixed Cylindrically Expanding Flames









$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0, \qquad \frac{\partial (\rho \mathbf{u})}{\partial t} + \nabla \cdot (\rho \mathbf{u} \otimes \mathbf{u}) = -\nabla p + \nabla \cdot \boldsymbol{\sigma},$$

$$\frac{\partial \rho h}{\partial t} + \nabla \cdot (\rho h \mathbf{u}) = -\nabla \cdot \mathbf{q}, \quad \frac{\partial \rho Z}{\partial t} + \nabla \cdot (\rho u Z) = \nabla \cdot (\rho D_Z \nabla Z) + \dot{\omega},$$

$$\mathbb{K} = \frac{1}{r}\frac{dr}{dt}. \qquad S_f = S_L - \mathcal{L}\mathbb{K},$$

Mixture		MA		MA-MC		MA-MA	
Name	p_0 (atm)	S_L (m/s)	$\mathcal{M}^b$	S_L (m/s)	$\mathcal{M}^b$	S_L (m/s)	$\mathcal{M}^b$
D80E95	1	3.6894	2.7282	3.772	4.4887	3.6990	3.0011
D82E90	1	3.2013	2.6941	3.2797	4.2494	3.2156	2.9735
D84E90	1	2.8169	2.6861	2.8856	4.2689	2.8318	3.0155
D89E65	1	1.0863	0.9635	1.0986	1.2127	1.0986	1.2127
	2	1.0273	0.9010	1.0427	1.1986	1.0381	0.9708
	3	0.9626	0.8340	0.9744	1.1514	0.9714	0.9051

What?

- Combustion research on alternate premixed fuel needs, a lot of Numerical studies
- Hydrogen-Air Combustion in Von Karman Combustion Chamber (VKCC)
- Creating a framework to calculate flame properties efficiently.

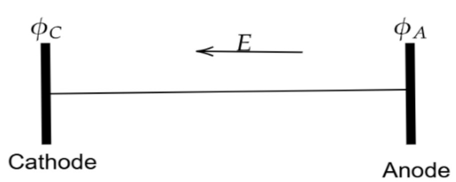
How?

- Usually 3D spherical solver are used- expensive. Here 1D cylindrical mapped solver is used, computationally efficient.
- Pre analysis using Cantera 1D-Planar
- Using Mixture-average model for diffusion velocity. Inhouse code NGA-HPC, Parallel, Soret effect included

Results

- Parameters interested – Flame speed, Markstein number and length
- NGA results are validated with experimental results.
- NGA is applied for premixed Hydrogen study.

Plasma Fluid Models for DC Glow Discharges using the Drift-Diffusion Model (FVM)



$$\frac{\partial n_s}{\partial t} + \nabla \cdot \Gamma_s = S$$

$$\Gamma_e = -\mu_e n_e E - D_e \nabla n_e, \Gamma_i = \mu_i n_i E - D_i \nabla n_i$$

$$S = \bar{\alpha}\left|\Gamma^{\text{drift}}\right| = \bar{\alpha}\mu_e |E| n_e$$

μ_e, D_e and $\bar{\alpha}$ are function of $|E|$

$$\nabla^2 \phi = -\frac{\rho}{\varepsilon_0}$$

$$E = -\nabla \phi \text{, where } \rho = (n_i - n_e)e$$

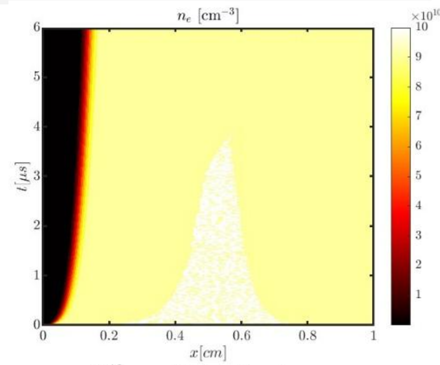
Cathode

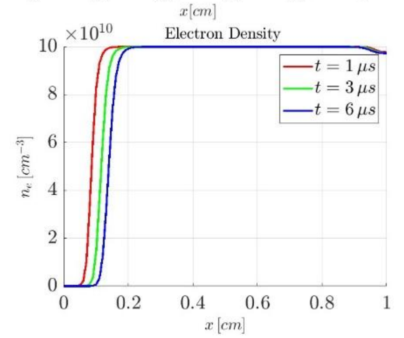
- $\nabla n_i = 0, \nabla n_e = 0$
- $\phi = \phi_C$
- $\Gamma_i = \frac{1}{4} v_i n_i + n_i \mu_i E$

  where $v_j = \sqrt{8k_b T_j / \pi m_j}$
- $\Gamma_e = \frac{1}{4} v_e n_e \pm \gamma \Gamma_i$

Anode

- $\nabla n_i = 0, \nabla n_e = 0$
- $\phi = \phi_A$
- $\Gamma_i = \frac{1}{4} v_i n_i$
- $\Gamma_e = \frac{1}{4} v_e n_e$





What?

- Drift diffusion model, very common model in plasma physics to study number density of electron and ions.
- **Applications:** Plasma Confinement, Glow Discharges and Ionization Fronts, Electric Probe Diagnostics, Semiconductor Plasmas, Plasma Thrusters

How?

- FVM
- Explicit time discretization ODE45
- Convective term Γ^{drift} - using Lax-Friedrich Flux Splitting and WENO5 for discretization
- Dirichlet B.C to solve the Electric Potential results a sparse matrix
- $\Delta t = 1$ ns, t = 6 μs,
- $\Delta x = 1$ μm, L = 1 cm

Results

- Captured the sheaths and number densities