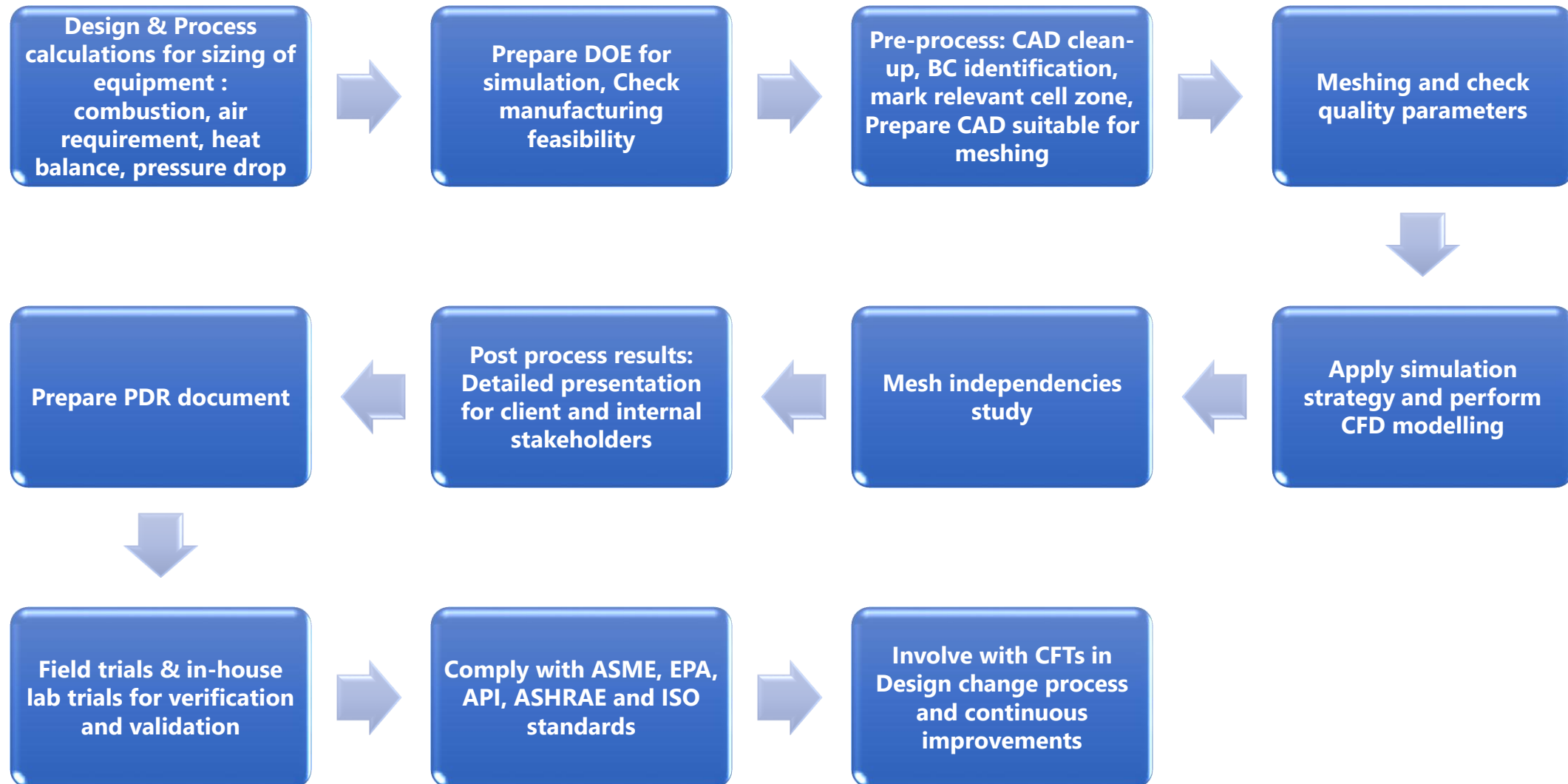


CFD Projects Overview

Presented by:
Sandeep Davare

Workflow



Thermal Incinerator

Objective

- Design thermal incinerator to heat the waste gases and recovery
- Involve in details calculations, Nozzle sizing its orientation, burner sizing calculation
- *Design Basis* : 3Ts, Exit temperature, residence time, Mixing

Operating Conditions

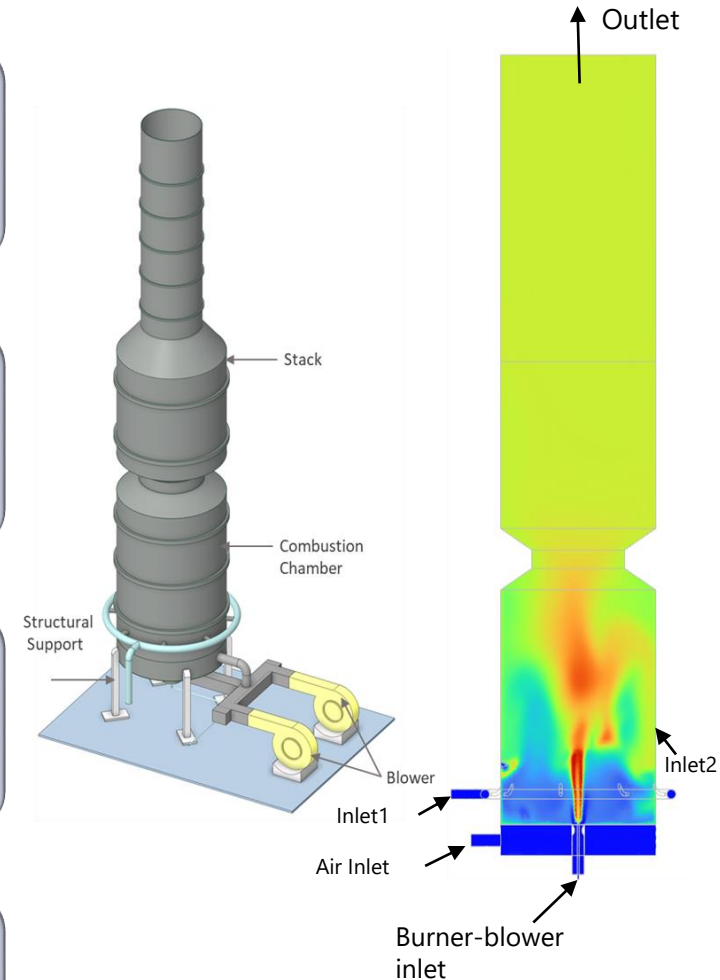
- Gas composition and respective mass flow rate of each stream, operating Pressure & temperature, Orientation, ambient conditions

CFD Methodology

- Steady State Realizable k- ϵ model with scalable wall function
- Energy equation & Radiation model (DO model) with ideal gas law
- Species transport model Eddy Dissipation & volumetric reactions
- Mesh count – 10M, y+ between 30-40

Outcomes

- Flue gas temperature, residence time, temperature distribution, Swirling motion of fluid flow, Flame profile & intermixing flue gases
- Check points: O₂ concentration at outlet (above 3% by mass), Species concentration at outlet to calculate DRE, Exit temperature, nozzle velocity, back pressure



TO-Design Basis

Steps in calculating the Sizing for Thermal Oxidizer

1. Calculate the average properties of each waste gas stream like LHV, density, MWT, Cp etc.
2. Calculate the air requirement for each waste gas stream by stoichiometric calculations (kg/s)
3. Calculate the air requirement for support gas/fuel gas (burner gas) (kg/s)
4. Calculate the total flue gas flow rate = sum of mass flow rate (*each waste gas stream + support gas/fuel gas + total air requirement for waste gas and burner fuel*)
5. Calculate the average properties of Flue gas stream like LHV, density, MWT, Cp etc.
6. Now as per literature we need to assume the temperature of flue gas (say 850°C) at the outlet and residence time (say 2 to 3sec)
7. Using the average density of flue gas at assumed temperature, calculate the volumetric flow rate of flue gas (mass flow rate of flue gas/density of flue gas)
8. Total volume required for TO= *volumetric flow rate of flue gas/ residence time (m3)*
9. Assume *Length: Diameter :: L:D* ratio of 2 to 3 (Therefore L=3D)
10. Equating total volume to $=\pi/4 * D^2 * L = (3/4) * \pi * D^3$, we will get Diameter and Length of Combustion chamber.
11. **Sizing for inlet and/or nozzle diameter** of Waste gas stream, Calculate the Volumetric flow rate of waste gas stream (mass flow rate of individual waste gas stream/density of waste gas)
12. Assume the velocity inside pipe as 20-25 m/s
13. Calculate the flow area to accommodate this velocity as (Volumetric flow rate of waste gas stream / Velocity)
14. Calculate Diameter from this area
15. Distribute the number of nozzles by selecting nozzle size to occupy this area & to achieve Ma <0.3
16. $DRE(\%) = \{(Ci - Cf) / Ci\} \times 100$ Where: Ci = Inlet concentration of the pollutant Cf = Outlet concentration of the pollutant

Enclosed ground flare

Objective

- Design & development of Enclosed ground as per API 521 & CPCB standard
- VBA tool-based design and process calculations
- *Design Basis:* flame profile, floor flux, burner exit velocity, window velocity, radiation & Species limit

Operating Conditions

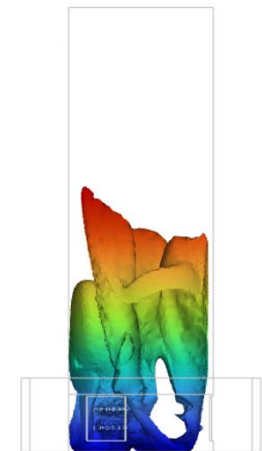
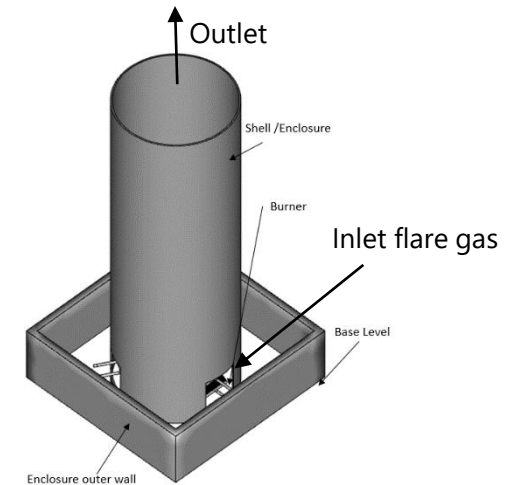
- Gas composition and respective mass flow, solid enclosure-ceramic properties, operating Pressure, temperature & ambient conditions for climate modeling

CFD Methodology

- Steady State Realizable k- ϵ model with scalable wall function
- Energy equation & Radiation model (DO model) with ideal gas law
- Species transport model Eddy Dissipation & volumetric reactions
- Mesh count – 12M, y^+ between 180-190

Outcomes

- Flare gas pressure, flame profile, floor flux, residence time, concentration at the outlet, flow distribution, draft pressure profile, HRR, DRE, surface incident radiation, wall temperature
- Studying multiple design configurations helped to select the best design to optimize overall weight of enclosure and consequently material saving



EGF Design Basis

- Assuming 80% of the pressure drop for sizing calculations (actual PD=0.8 * allowable PD)
- Verification of design calculations as per below conditions
- Exit tip velocity between 80 to 100 m/s
- Allowable floor flux should be below 300000 BTU/hr ft² (946 kW/m²) as per API 560
- Air ingress velocity through Window in the range of 4-6 m/s
- The window size is selected based on air entering velocity to achieve required natural draft and stoichiometric air flow to ensure complete combustion.
- To improve the mixing of fuel and air while maintaining higher EGF temperature for better combustion efficiency multiple trials were evaluated with respect to burner tip, EGF window & enclosure height.
- burner tip, Burner orientation, hole & slot sizes, number of holes were decided to achieve uniform flow distribution across the cross section of EGF.

EGF Burner Sizing

$$\rho = pm/rt$$

$$P_c = \left[\frac{2}{k+1} \right]^{k/(k-1)}$$

$$\frac{P_b}{P_t}$$

If $P_c < P_b/P_t$, then fuel exits orifice at subsonic conditions

If $P_c > P_b/P_t$, then fuel exits orifice at sonic conditions

$$M_e = \sqrt{\frac{2}{k-1} \left[\left(\frac{P_t}{P_b} \right)^{\frac{k-1}{k}} - 1 \right]} \quad c_e = \left[\frac{kT_e \bar{R}}{MW} \right]^{\frac{1}{2}}$$

$$T_e = \frac{T_t}{1 + \frac{k-1}{2} M_e^2} \quad \rho_e = \frac{P_b}{T_e \bar{R} / MW}$$

$$\dot{m} = c_d \rho_e A M_e c_e$$

formula from Wes bussman article of fluid flow
equation no.3.20

General Calculations

Air Requirement calculations											
Component	MW	Mass Fraction	Moles	Mole Fraction	Mole %	Wt%		Stoichiometric coefficient	Moles of O ₂ required	Stoichiometric moles of Air required	Stream LHV
Hydrocarbon	a	b	c=b/a	d=c/Σc	e=d*100	f=a*e	g=(f/Σf)*100	x	h=e*x/100	i=4.76*h	L=g*LHV of Hydrocarbon
			Σc			Σf				Σi	ΣL

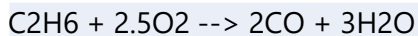
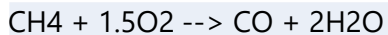
MW of Air	28.95	kg/kmol
MW of gas	Σf / 100	kg/kmol
Molar Flow of gas	m.Dot / MWW of gas	kmol/s
Actual Air required stoichiometrically = A	molar flow of gas * Σi	kmol/s
Mass flow rate = B	A * air MW	kg/s

Where m.dot = flare gas mass flow rate (kg/s)

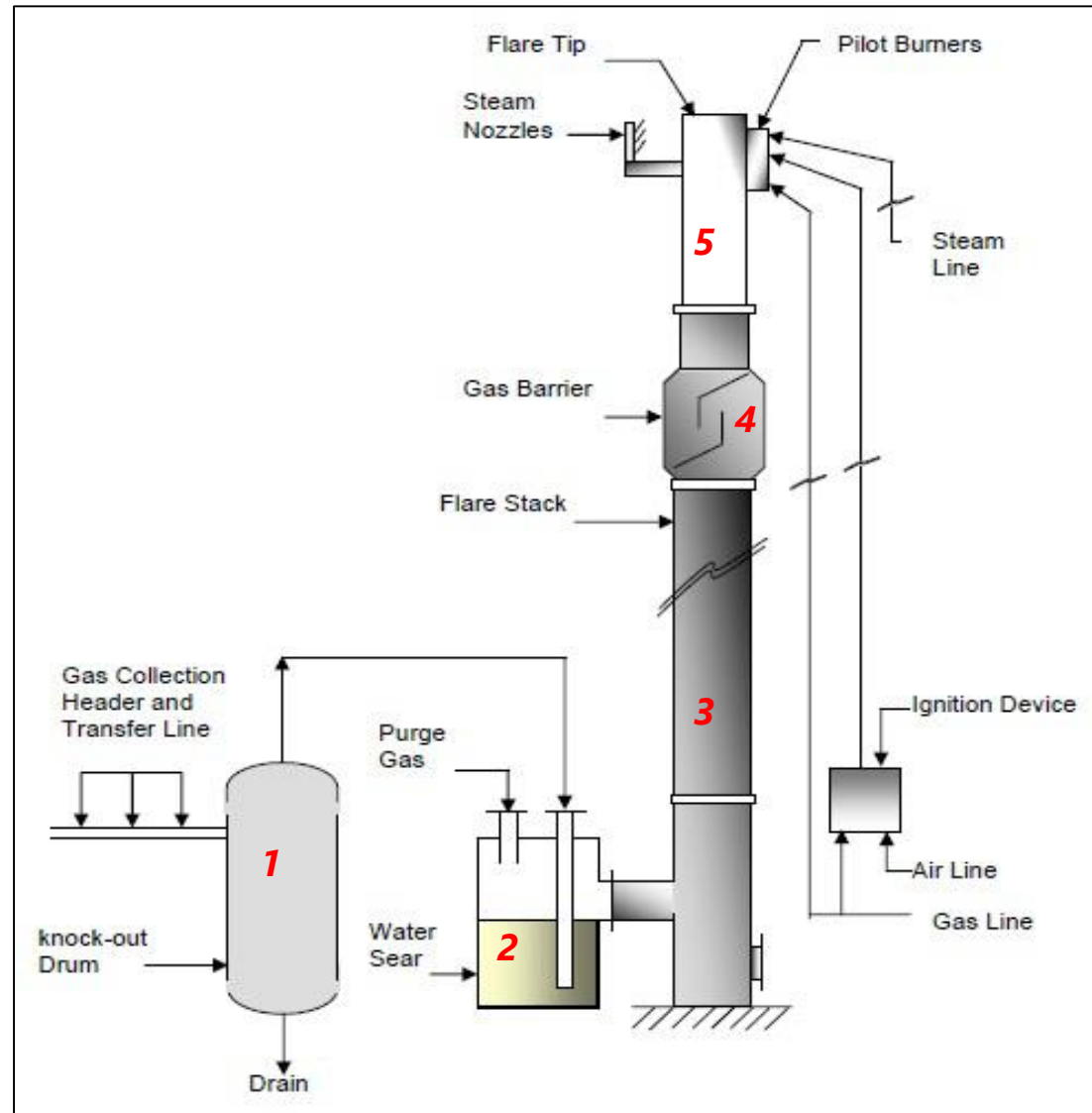
Draft pressure drop calculation

$$dh = 1000 * h(\rho_0 - \rho_r) / \rho_{h2o}$$

Stoichiometric calculations



Elevated Flare system components



Elevated Flare Tip

Objective

- Design & development of Elevated flare tip as per API 521 & CPCB standard- for high turndown ratio & low blower flow
- Detailed design and process calculations
- Develop VBA tool for design and process calculations
- **Design Basis:** flame profile , exit velocity, blower duct velocity

Operating Conditions

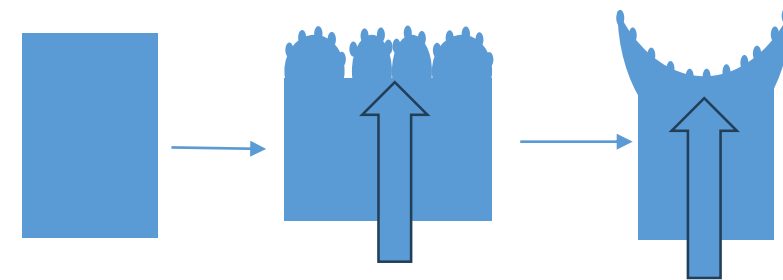
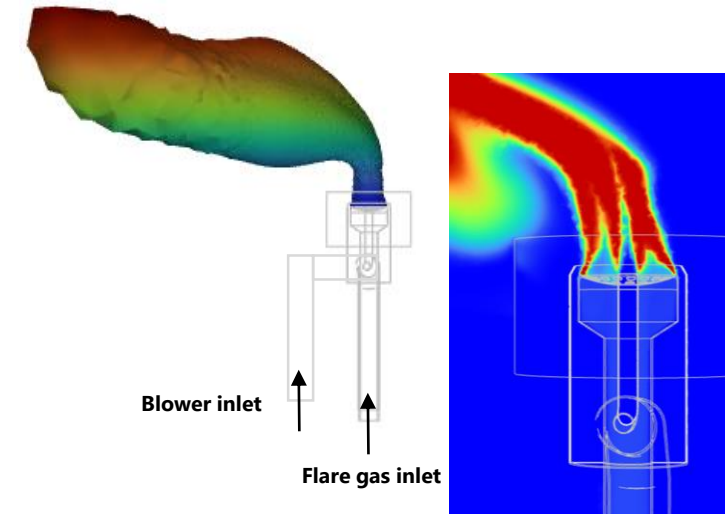
- streams composition and respective mass flow rate, operating temperature and pressure, ambient conditions for climate modeling

CFD Methodology

- Steady State Realizable k- ϵ model with scalable wall function
- Energy equation & Radiation model (DO model) with ideal gas law
- Species transport model Eddy Dissipation & volumetric reactions
- Mesh count – 8M, y+ between 60-70

Outcomes

- Flame profile, adiabatic flame temperature, tip temperature, concentration of species, exit tip velocity
- Multiple design evaluation with geometrical changes arrived at final design



Design Evolution : conventional to limpet to Concave tip

Elevated flare tip-Design evolution

- Exit tip velocity from flare tip should be below 80 m/s and blower duct velocity should be between 25-30 m/s
- VBA tool to calculate pressure drop , radiation , support fuel , noise, purge calculations.
- The total area of barrel is matched with the total area of holes on the new tip surface which doesn't add any pressure drop in tip.
- Assist air area modification, retention lugs design modifications to avoid reentering air from blower to flare tip.
- Multiple designs were evaluated by changing holes orientation, slant plate angle, flow distribution area for blower air to ensure proper flaring at low flow rates without any air entrainment into the flare tip.
- At the center the annular pipe for blower air is added to enhance mixing of fuel and air. Because of the high velocity from this central air pipe the negative pressure zone is created near the tip exit which helps in upward flow of fuel even for low fuel pressures.
- Elimination of retention lugs as compared to traditional design, where the high temperature zones were observed. As the mixing and turbulence is created by high velocity of air coming from annular space and central air pipe.

Pressure drop calculation in Elevated Flare System

Equipment	Process Sheet Del_P Calculation	Del_P Unit
Tip	$\text{Del_P} = \rho \cdot v^2 \cdot 0.5 / g;$	mmWC
Riser	$\text{Del_P} = 4fL \cdot \rho \cdot v^2 / 2gD$ Where $f = 0.079 / \text{Re}^{0.25}$	mmWC
Mol. Seal	$\text{Del_P} = C_d \cdot \rho \cdot v^2 \cdot 0.5 / g;$ Where $C_d = 3.5 - 0.035 \cdot (D)$	mmWC
WS	$\text{Del_P} = \rho \cdot v^2 \cdot 0.5 \cdot (1 + 0.25) / g$	mmWC
KOD	$\text{Del_P} = \rho \cdot v^2 \cdot 0.5 \cdot (1 + 0.5) / g$	mmWC
DPSC	$\text{Del_P} = X \cdot \text{Pressure factor (based on opening)}$ Where $X = 17500 \cdot ((v)/1000) / (\text{Dia, m}^2)$	mmWC

Multiphase analysis of Water Seal Drum

Objective

- Water seal drum design evaluation to overcome the field issue : water droplets observed at the tip outlet
- To examine the carryover of water droplets with flare gas flow
- design modifications in WSD to reevaluate the impact on performance as per OISD-106

Operating Conditions

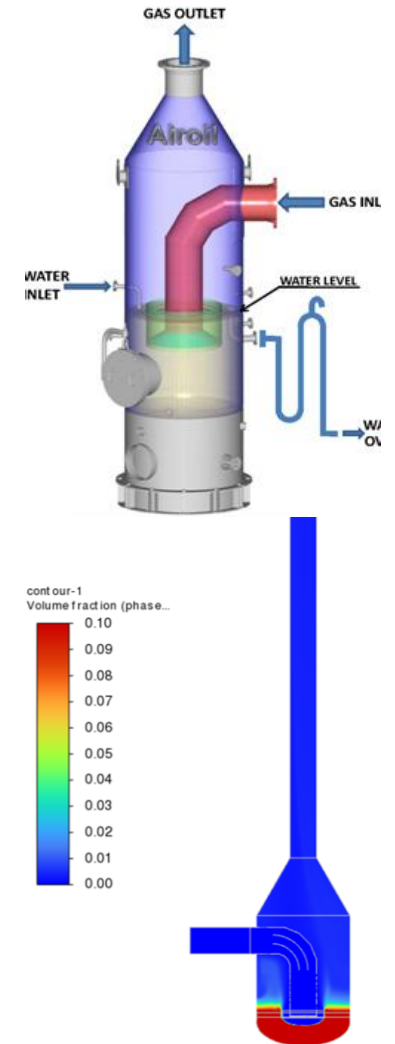
- Flare gas properties, water as secondary phase, fluid-water phase interaction coefficient, mass flow rate of flare gas, atmospheric pressure outlet

CFD Methodology

- Transient simulation with Realizable $k-\epsilon$ model with scalable wall functions, Multiphase with VOF- Sharp/dispersed interface model
- Mesh count – 20M and y^+ between 30-60

Outcomes

- Volume fraction distribution at the outlet of flare tip, velocity profile inside flare riser
- Water droplets reduced from 3% to 0.06% by changing miter bend length, water shield diameter, inserting turning vanes, perforation density of holes



Setup details

Numerical Settings:

- Implicit VOF with vof cut off $1e-10$ and Implicit body force turned off
- NITA (Non Iterative Time Advancement)
- P-V Coupling : PISO with Neighbor Correction 1
- Transient Formulation : Bounded second order time
- Discretization :
 - Gradient : Green-Gauss Node Based
 - Pressure : Body Force Weighted
 - VOF : Compressive
 - Momentum : Second Order Upwind
- URF
 - Pressure, Momentum and VOF = 0.5
 - Turbulent Kinetic Energy, Turbulent Dissipation Rate =1

time step = 0.0005 & 2000 Steps 20 iterations per step, CFL below 50

Solution Methods ⓘ

Pressure-Velocity Coupling

Scheme
SIMPLE

Spatial Discretization

Gradient
Least Squares Cell Based

Pressure
PRESTO!

Density
Second Order Upwind

Momentum
Second Order Upwind

Volume Fraction
First Order Upwind

Turbulent Kinetic Energy
First Order Upwind

Turbulent Dissipation Rate
First Order Upwind

Energy
Second Order Upwind

Pseudo Time Method
Off

Transient Formulation
First Order Implicit

☐ Non-Iterative Time Advancement

☐ Frozen Flux Formulation

☒ Warped-Face Gradient Correction

☐ High Order Term Relaxation

Options...

Centrifugal Compressor performance

Objective

- To evaluate flow characteristics of turbocharger compressor unit
- Compressor Maps - dP vs mass flow rate, Power & efficiency plots Surge and Choke Boundary regions

Operating Conditions

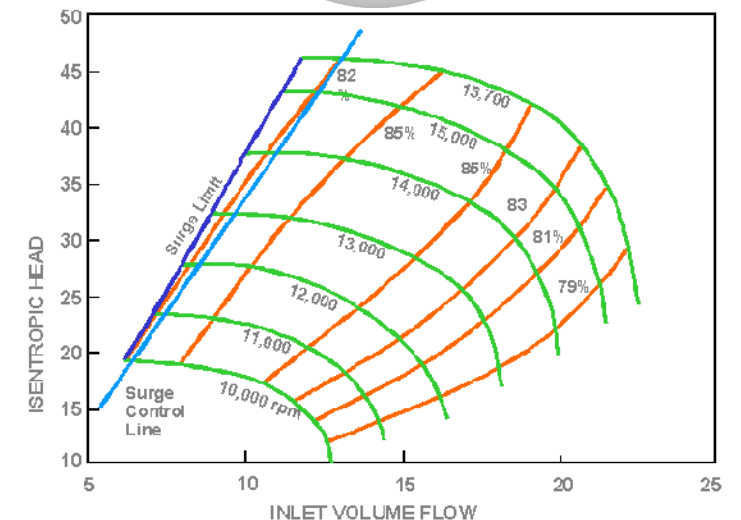
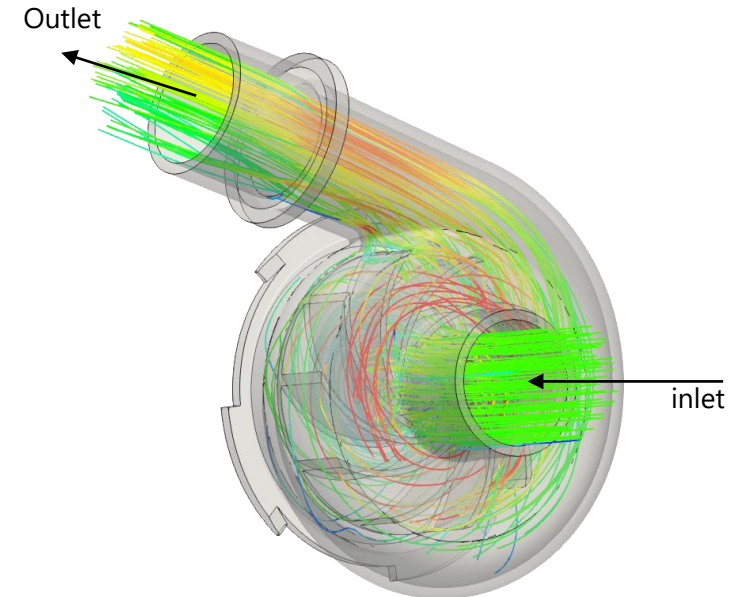
- Inlet pressure with temperature & Outlet pressure with temperature
- K & Cp as function of temperature, set of impeller rpm

CFD Methodology

- Steady state with k- ω SST model
- Non inertial frame of reference, energy equation, Sutherlands law and ideal gas law
- Mesh count – 12M with y^+ of 5-10

Outcomes

- Pressure ratio, mass flow outlet, Pressure & Velocity contours, temperature distribution on impeller



Important Formulae

$$\eta_s = \frac{\left(\frac{P_2}{P_1}\right)^{\frac{\gamma-1}{\gamma}} - 1}{\frac{T_2-T_1}{T_1}}$$

$$N_{\text{corr}} = N \sqrt{\frac{T_{\text{ref}}}{T_1}}$$

$$\dot{m}_{\text{corr}} = \dot{m} \sqrt{\frac{T_{\text{ref}}}{T_1}} \left(\frac{P_1}{P_{\text{ref}}}\right)$$

$$\sigma = 1 - (U/\Delta V_t)$$

$$w = C_p(T_2 - T_1)$$

$$P = \dot{m} \cdot w = \dot{m} \cdot C_p(T_2 - T_1)$$

$$T = P/\omega$$

Where

- N = Actual rotational speed
- T_1 = Inlet temperature
- T_{ref} = Reference temperature (Std atmospheric temperature, 288.15 K)
- \dot{m} = Actual mass flow rate
- P_{ref} = Reference pressure (Std atmospheric pressure, 101325 Pa)
- P_1 & P_2 = inlet and outlet pressure respectively
- C_p = Specific heat at constant pressure
- ΔV_t = *Tangential velocity loss*
- U = *Impeller tip speed*
- ω = *Angular velocity* = $2\pi N/60$

Fixed wing Aerial vehicle Aerodynamics

Objective

- To evaluate Aerodynamics performance of aerial vehicle
- Grid independency study to evaluate consistency of results

Operating Conditions

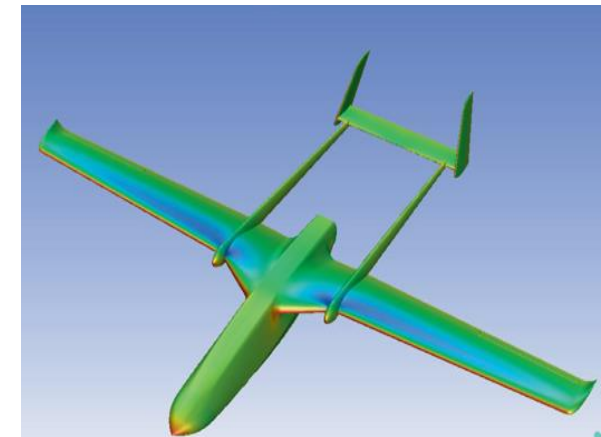
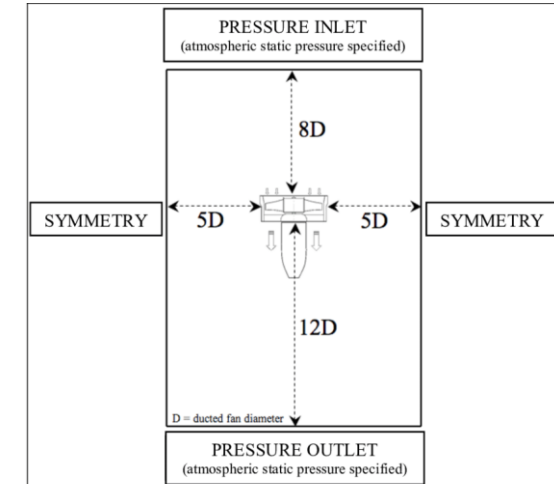
- Fluid properties AMSL for climate modelling
- Cruising speed as inlet velocity (23m/s)
- Atmospheric Pressure outlet

CFD Methodology

- Steady state with k- ω SST model
- Mesh Count 40M, 20M, 9M used, symmetry BC with y^+ of 2-3

Outcomes

- Pressure drop profile, velocity contours
- Drag and lift forces on body and wings (NACA aerofoil profile)
- Pressure forces at inputs to structural analysis
- Field trials correlation with CFD study



Important Formulae

$$L = \frac{1}{2} \rho V^2 S C_L$$

$$D = \frac{1}{2} \rho V^2 S C_D$$

$$P = D \cdot V$$

$$\frac{L}{D} = \frac{C_L}{C_D}$$

Where

L : Lift force (N)

D : Drag force (N)

ρ : Air density (kg/m³)

V : Velocity of the airflow relative to the wing (m/s)

S : Wing area (m²)

C_L , C_D : Lift & Drag coefficient (dimensionless)

P : Power (kW)

Thrust : Drag

Cold Plate-CHT in Power Electronic components

Objective

- To study thermal efficiency of DC-DC power inverter (NPD) cooling jacket
- Evaluate design changes using CFD during NPD stage

Operating Conditions

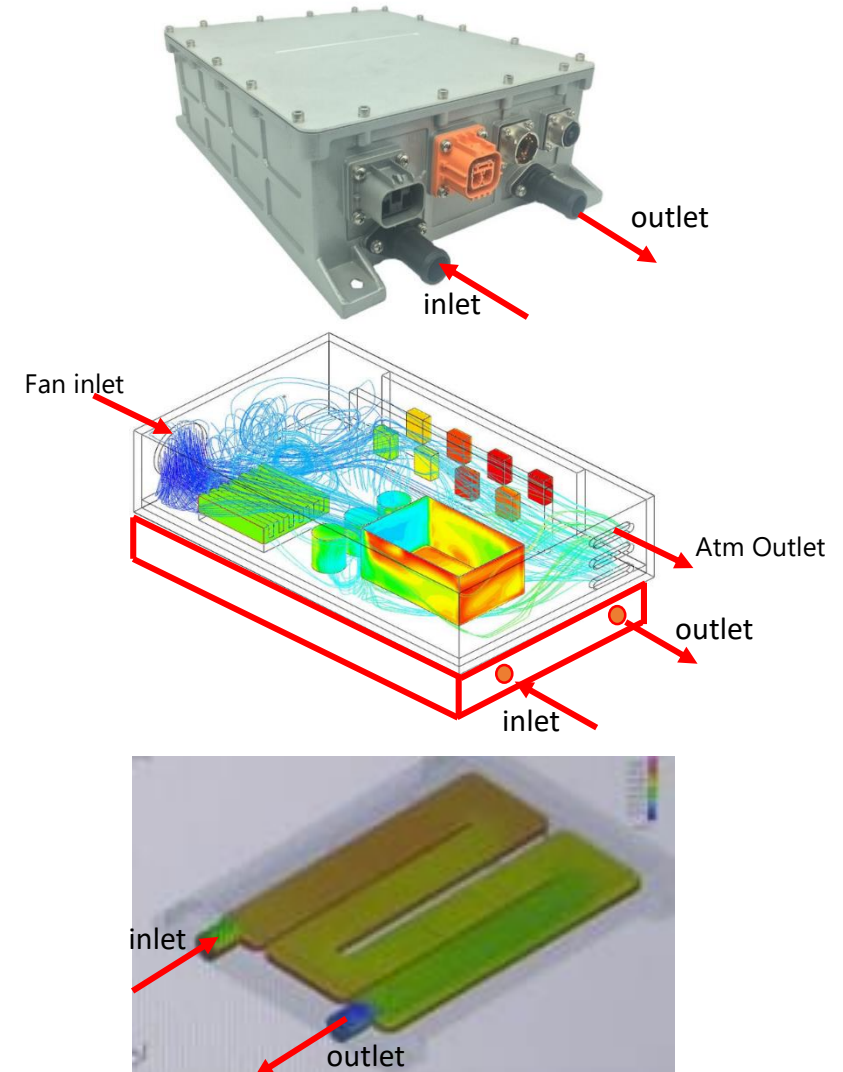
- Coolant : Ethylene Glycol & Water (50/50) with temperature : 26 & 45°C
- Volumetric flow inlet : 3, 5 & 10 LPM & pressure BC at outlet
- Heat sources : 400 W total
- Fan Curve (pressure jump vs Velocity)

CFD Methodology

- Steady state with Standard k- ϵ model & enhanced wall-treatment
- Mesh Count 8M with y^+ of 2-3

Outcomes

- Pressure drop profile, velocity contours, temperature distribution
- Design changes were effectively implemented
- Experimental correlation with CFD study to maintain the ΔP of 5 to 7°C



Battery Terminal cooling

Objective

- To address the hotspot issue in battery of EV truck application through CFD
- To evaluate thermal characteristics of Battery Terminal circuit to achieve uniform velocity distribution through the circuit

Operating Conditions

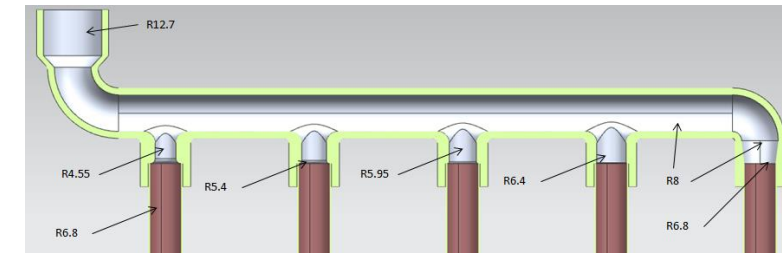
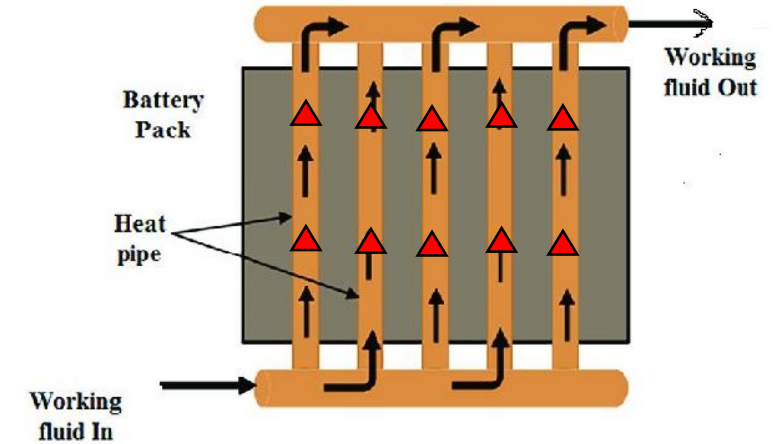
- Coolant temperature : 26°C & 45°C
- Pressure inlet of 2 bar & Volumetric flow outlet
- Fluid: Ethylene Glycol & Water (50/50)
- Heat-Spot temperature : 100 & 120 °C

CFD Methodology

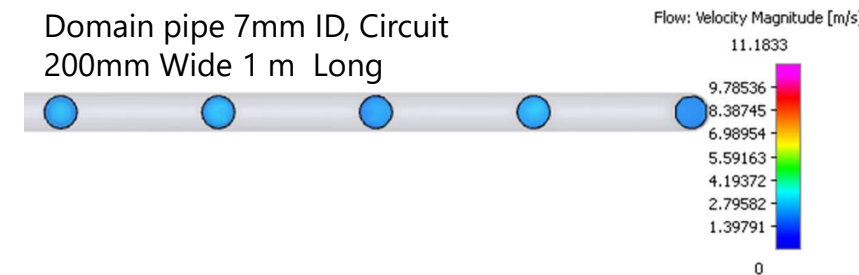
- Steady state with Standard k- ϵ model & enhanced Wall treatment with energy equation
- Mesh Count 4.8M with y^+ of 1-2

Outcomes

- pressure drop profile, velocity contours, Hotspot temperature distribution
- Experimental correlation with CFD study
- Pump selection criterion was evaluated from study

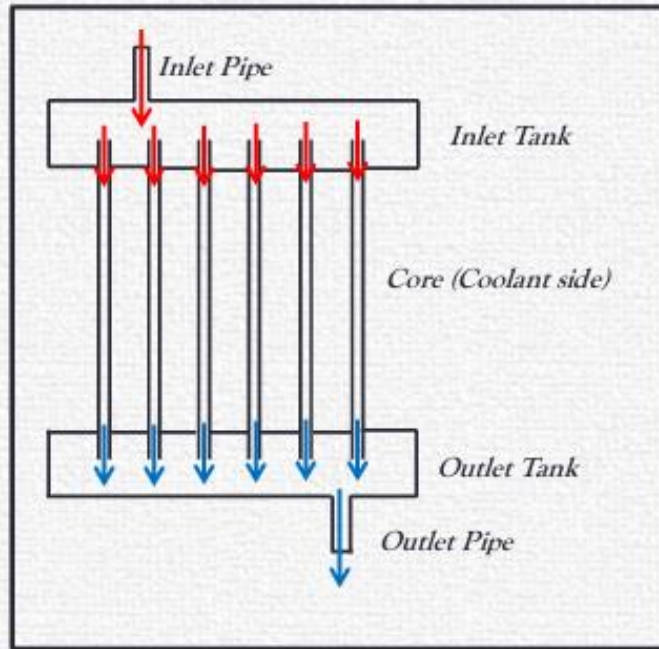


Domain pipe 7mm ID, Circuit
200mm Wide 1 m Long



Important Formulae

$$\begin{aligned}\Delta P_{Total} &= (\Delta P_f)_{inlet\ pipe} + (\Delta P_e)_{inlet\ pipe-inlet\ tank} + (\Delta P_c)_{inlet\ tank-core\ inlet} + (\Delta P_f)_{core} + (\Delta P_e)_{outlet\ tank-core\ outlet} \\ &+ (\Delta P_c)_{outlet\ tank-outlet\ pipe} + (\Delta P_f)_{outlet\ pipe}\end{aligned}$$



$$\Delta P_f = \frac{f \rho L V^2}{2D}$$

$$\Delta P_e = \frac{k_e \rho V^2}{2}$$

$$\Delta P_c = \frac{k_c \rho V^2}{2}$$

ΔP_f : Frictional Pressure Drop

ΔP_e : Expansion Pressure Drop

ΔP_c : Contraction Pressure Drop

k_e : Expansion Coefficient

k_c : Contraction Coefficient

Note: If there will be bend in inlet and/or outlet pipes, pressure drop due to bend will also be taken into consideration.

Thank You....!

<p>Turbulence Model – K-ω Shear Stress Transport (SST) Model</p>	<p>K-ω SST turbulence is a two equation model in which transport equations are solved for the turbulent kinetic energy (k) & specific dissipation rate (ω). It accounts for the transport of the turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients & rotating flows. It's has combination of K-ϵ & K-ω Model. Most suitable for boundary region or near wall problems</p>
<p>Turbulence Model – K-ϵ Model</p>	<p>It is general purpose industrial application model suitable for inviscid flow region which is away from the boundary (where viscous effects are not important to model) with suitable y^+ (30 to 300 for standard wall function and 1 to 5 for enhanced wall treatment) and wall functions</p>
<p>Multiphase with VOF model</p>	<p>VOF model is best suitable for sharp interface, two immiscible fluids with little or no mixing. The phase interaction is important parameter and described by surface tension modelling and wall adhesion to see effects of two or more phases</p>
<p>Turbulence Model – K-ϵ Realizable Model</p>	<p>Suitable for Swirling flows, flow separation and general purpose engineering problems. Good in converging</p>

Radiation Model – Discrete Transfer Model	For general cases, ranging from optically thin (transparent) to optically thick (diffusion) regions, like combustion, the Discrete Transfer model more accurately represent the solution of the radiative transfer equation.
Combustion Mechanism – 2 Step reaction	<p>The combustion is done as 2 step mechanism for CH₄, with carbon monoxide as intermediate product. Two step methane combustion, with CO as intermediate product is as follows,</p> $\text{CH}_4 + 1.5 \text{O}_2 \rightarrow \text{CO} + 2\text{H}_2\text{O}$ $\text{CO} + 0.5\text{O}_2 \rightarrow \text{CO}_2$ <p>Other chemical reactions can be incorporated with rate exponent and stoichiometric balance</p>

