MECHWELL

# MECHWELL

DRIVEN BY INNOVATION

# Capacity Building of Officers from Petroleum Refining Sector on Efficient Use of Energy

07th July, 2017 IOCL, Vadodara

Presented By : Akshay Shah

# Introduction

MECHWELL

**Established in 1983, Mechwell is a multi-discipline applied R&D oriented engineering firm.**

**Mechwell was established to provide solutions for Cement, Power & Allied industries for flow, emissions & energy consumption problems.**

**Business:**

- **Turnkey Solutions to Cement, Power & Allied Industries including custom engineering, specialty systems design (EFIP), manufacturing and implementation, and support to industry in emissions control to achieve SPM<30mg, heat and mass transfer, and applied energy systems.**
- **Design, Manufacture & Supply of Expansion Joints, Dampers & Gates and EFIP's**
- **CFD & FEA Consultancy Services.**

**Domains :**

**Research & Development , Testing , Manufacturing & Supply , Air Pollution Control**

# Mechwell Industries Ltd..

R&D – Center For Innovation

Testing & commissioning

Manufacturing

Air pollution control

# R&D – CENTER FOR INNOVATION

- The use of a range of advanced tools including Computational Fluid Dynamics (CFD),

  Finite Element Analysis (FEA) Computer Aided Design (CAD) and other simulation technologies.

- Mechwell provides Computer Aided Engineering services to a host of industries, most notably the **Power, Cement, Aviation, Oil and Gas, Renewable Energies and Automotive sectors.**

# CAE- R&D CENTER FOR INNOVATION

- **Why CFD – Computational Fluid Dynamics ?**

- It's the numerical analysis method involving large computerized iterations for predicting fluid flow patterns .
- Time efficient & Optimized Solutions

- **Process for Design Optimization**

- Computerized 3D CAD model is developed – with Actual Site Drawings
- High Quality Unstructured Hex Mesh is prepared for the model
- After confirmation from client fluid flow simulation is proceeded to Analyze flow patterns
- Optimum of design Changes for Ideal fluid flow across the system

# R&D – Center For Innovation

- The use of a range of advanced tools including Computational Fluid Dynamics (CFD), Finite Element Analysis (FEA) Computer Aided Design (CAD) and other simulation technologies.

## Why CFD – Computational Fluid Dynamics ?

- It's the numerical analysis method involving large computerized iterations for predicting fluid flow patterns .
- Time efficient & Optimized Solutions

$$\left\{\frac{\partial}{\partial x}(\overline{u^2}) + \frac{\partial}{\partial y}(\overline{uv}) + \frac{\partial}{\partial z}(\overline{uw})\right\} = -\left(\frac{\partial \overline{p}}{\partial x}\right) + \left\{\frac{\partial}{\partial x}\left(\mu\frac{\partial \overline{u}}{\partial x} - \rho\overline{u'^2}\right) + \frac{\partial}{\partial y}\left(\mu\frac{\partial \overline{u}}{\partial y} - \rho\overline{u'v'}\right) + \frac{\partial}{\partial z}\left(\mu\frac{\partial \overline{u}}{\partial z} - \rho\overline{u'w'}\right)\right\}$$

# CAE - Application Domains & Focus Areas

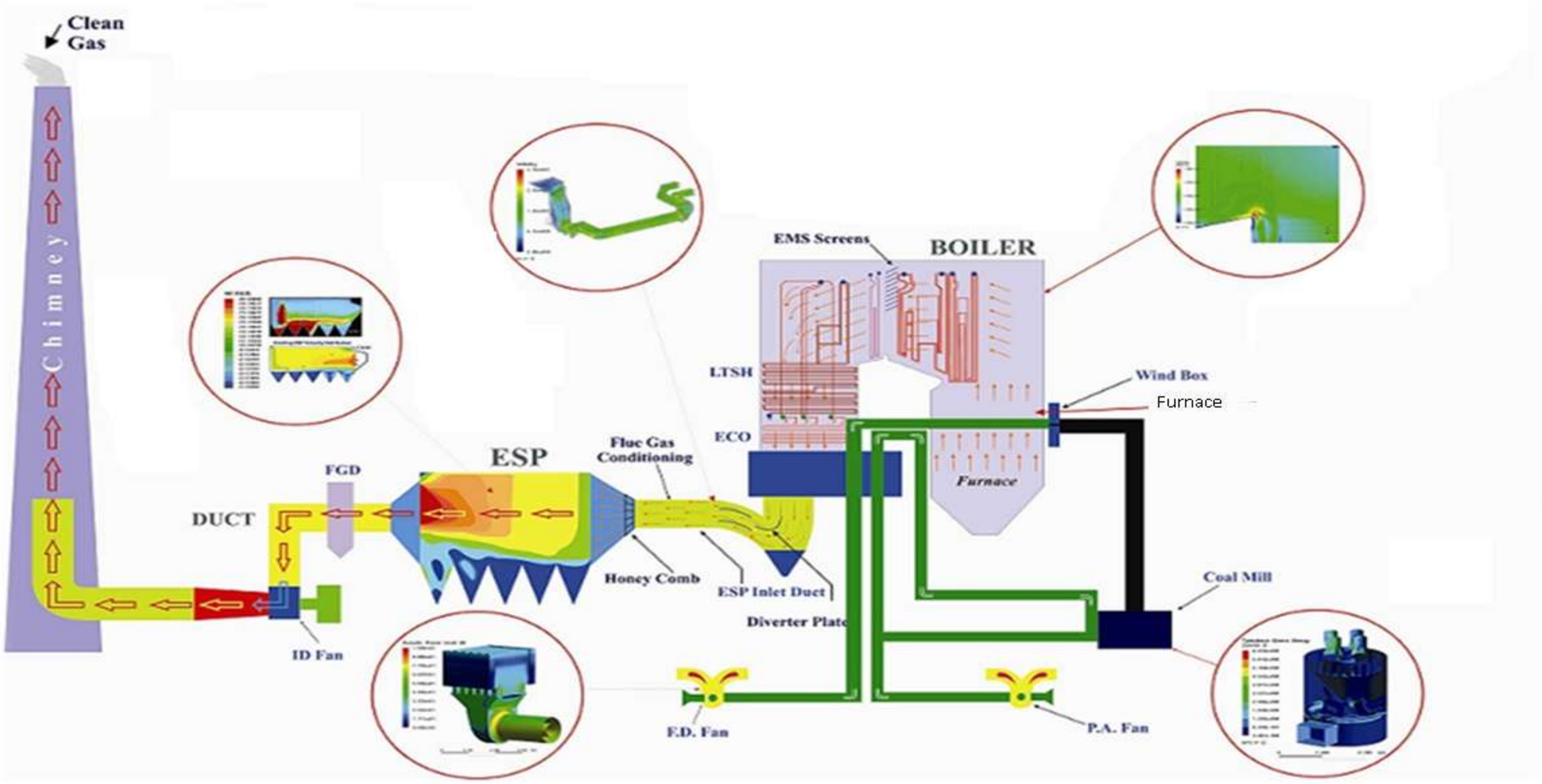
MECHWELL





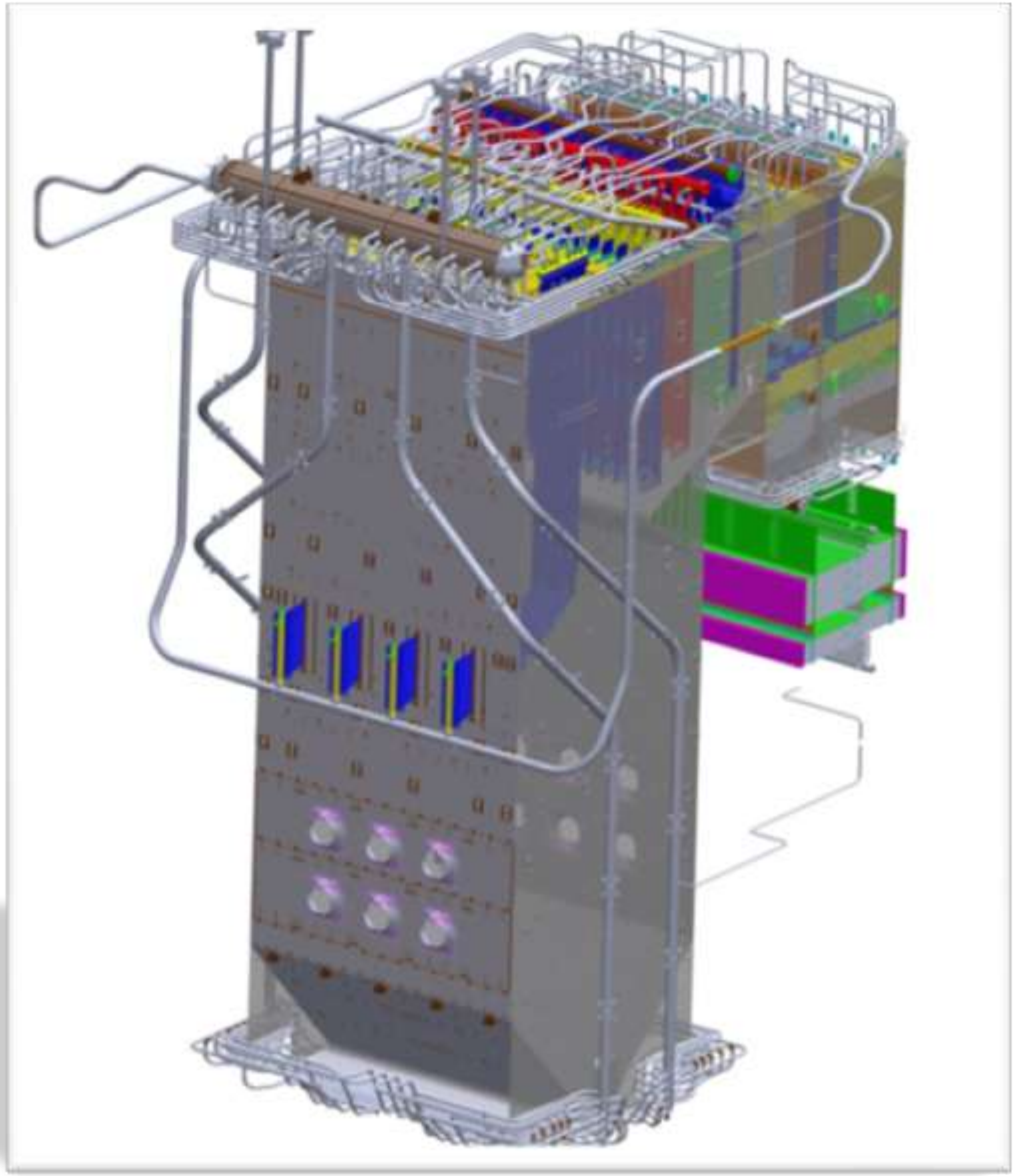


# CFD In Power Plants

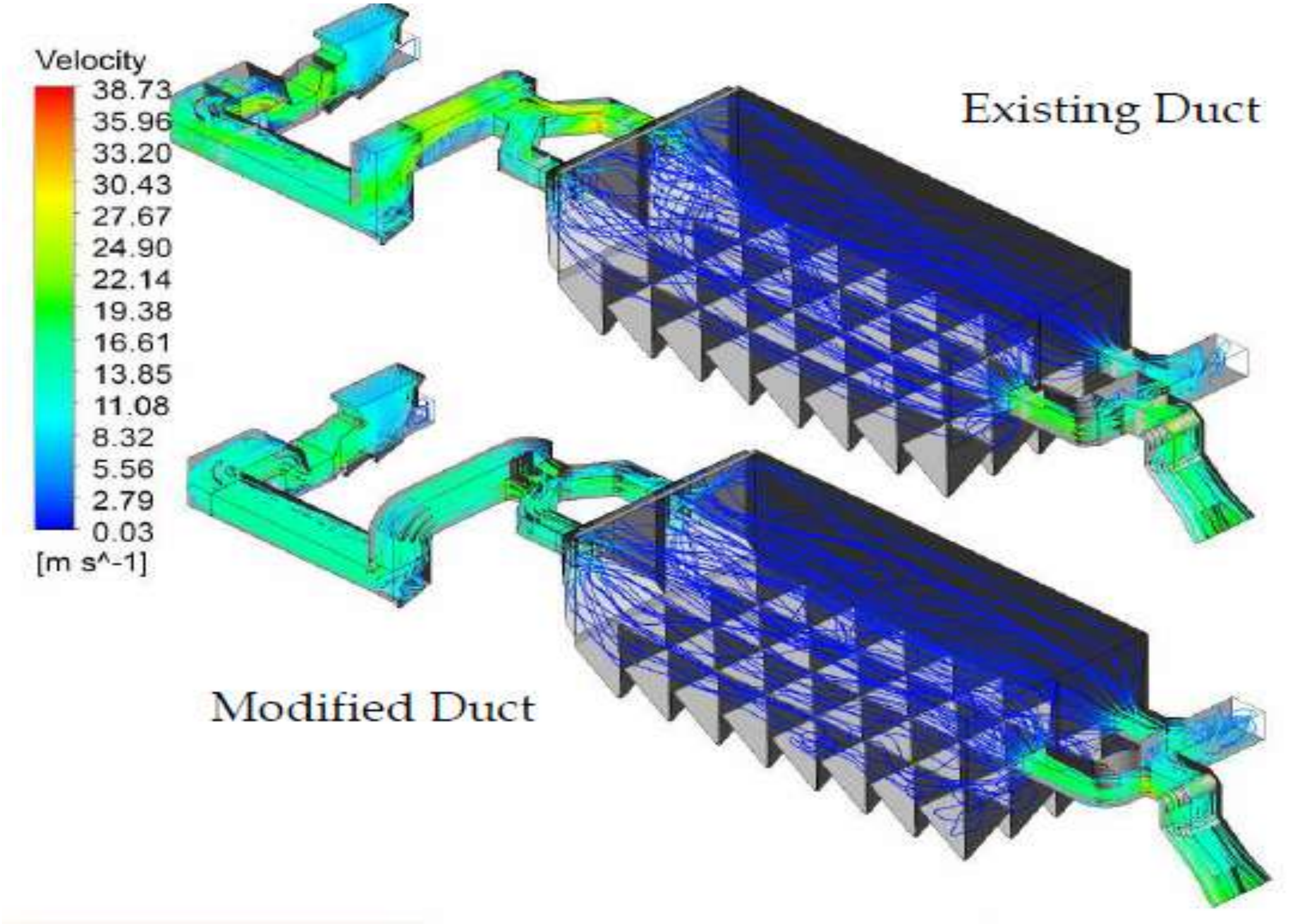


# Computational Fluid Dynamics : Steps

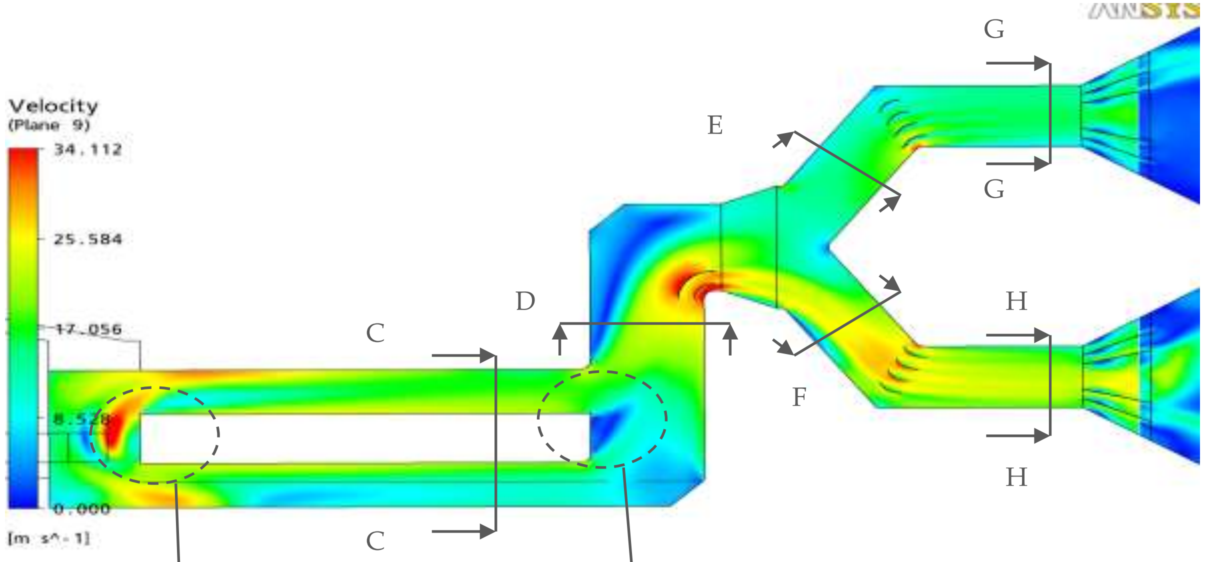
- ✓ Identification of problem
- ✓ Data Collection
- ✓ Modeling
- ✓ Mesh Generation
- ✓ Boundary Condition
- ✓ Evaluation
- ✓ Modification/ Analysis
- ✓ Simulation
- ✓ Verification



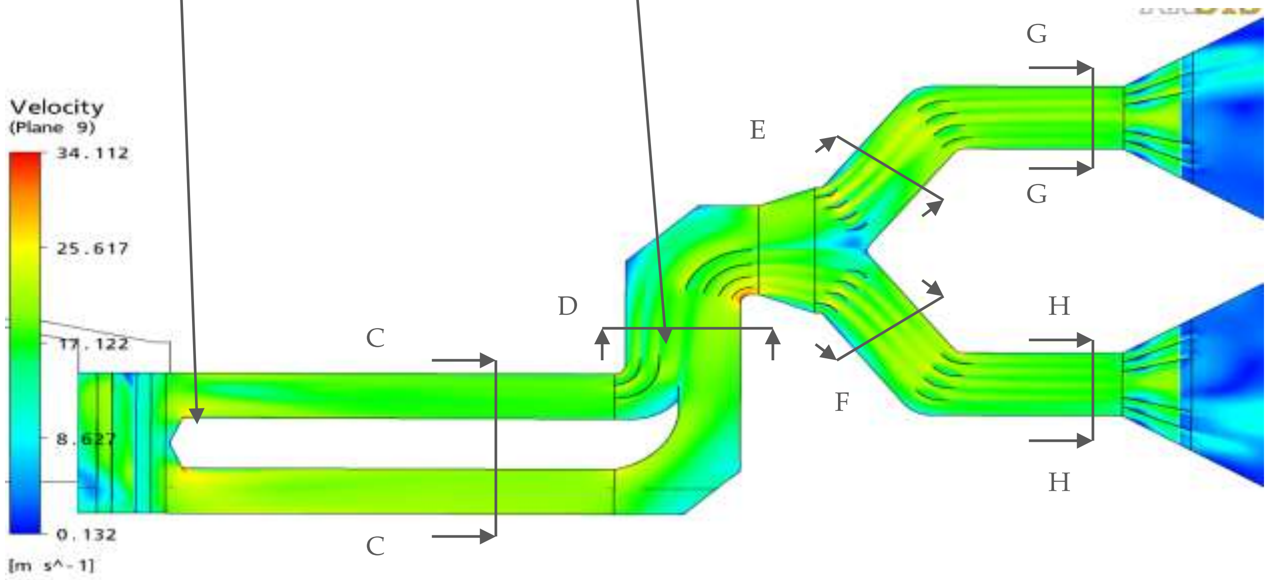
# CFD RESULTS: AH Outlet TO ESP Outlet



# Comparative -CFD Results For AH A Pass




Velocity Plot: Existing_ESP Inlet Duct




Velocity Plot: Modified_ESP Inlet Duct

After Modification of Designing the of duct plates & Guide plates at marked location, the immediate flow obstruction & flow Concentration has been avoided & flow is nearly uniform in the duct.

# CFD Results

**Table shows pressure drop across different sections for existing and modified case**

| Location | Existing Model pressure drop mm WC | Modified Model Pressure drop mm WC |
|---|---|---|
| APH outlet to ESP Inlet | 43.4 | 20.5 |
| ESP outlet to ID fan Inlet | 17.3 | 10.0 |

* ***Net Pressure Drop across the system : 30mmWC in Modified Model.***

# Duct Testing pre Installation

## Measurement of Velocity of Flue Gas In Duct

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| Client | Jindal Power Ltd U-3 | Date of Test: 08.11.2013 | | Duct Location - APH Outlet A Pass | | | |
| Unit Load : 248 MW | | ID A | 191 AMP. | Damper Position ID A | 80% | FD A | 29 AMP. |
| | | ID B | 196 AMP. | Damper Position ID B | 80% | FD B | 30 AMP. |
| | | | | Scoop position | 70% | | |

| | | | | | |
|---|---|---|---|---|---|
| Static Pressure (Ps)-mmWc | -176.14 | Pitot Tube Constant | 0.9 | Duct Dimensions - 3530 X 8840 | |
| Barometric Pre. (Ba)-mmHg | **760** | | | | |
| Abs.Pressure (Pu)-mmWc | 747.05 | Temperature(tu)° C | 150 | Density (Do) = 1.3 kg/Nm3 | |
| Pu=Ba+Ps/13.6 | 747.05 | Temperature(Tu)° K | 423 | Du = Do*(Pu/760)*(273/Tu) | 0.8247 |

| | Dynamic Pressure = Pd mmWC | | | | | | Velocity (Vu) = 4.4*K*Sq.rt.(Pd/Du) | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | Transit point no./distance from end | | | | | | | | | | | | |
| Port No. → / Reading ↓ | 1 | 2 | 3 | 4 | | | 1 | 2 | 3 | 4 | | | Avg. Vel. m/sec |
| 1 | 8.4 | 5.1 | 2.5 | 6.4 | | | 12.6 | 9.8 | 6.9 | 11.0 | | | 10.10 |
| 2 | 8.1 | 6.4 | 2.8 | 7.6 | | | 12.4 | 11.0 | 7.3 | 12.0 | | | 10.69 |
| 3 | 5.8 | 5.3 | 4.6 | 4.6 | | | 10.5 | 10.1 | 9.3 | 9.3 | | | 9.81 |
| 4 | 3.6 | 3.0 | 1.3 | 2.5 | | | 8.2 | 7.6 | 4.9 | 6.9 | | | 6.92 |
| 5 | 14.5 | 7.4 | 4.6 | 5.3 | | | 16.6 | 11.8 | 9.3 | 10.1 | | | 11.96 |

| | | | |
|---|---|---|---|
| Av. Pd = | 5.49 | Av. Vel. (m/sec) = | 10.21 |

# Duct Testing post Installation

MECHWELL

## Measurement of Velocity of Flue Gas In Duct

| | | | | | |
|---|---|---|---|---|---|
| Client | Jindal Power Ltd U-3 | Date of Test: 24.02.2014 | | Duct Location - APH Outlet A Pass | |
| Unit Load : 250 MW | | | | | |

| | | | | | |
|---|---|---|---|---|---|
| ID A | 141 AMP. | Damper Position ID A | | FD A | 32.5 AMP |
| ID B | 143 AMP. | Damper Position ID B | | FD B | 32.6 AMP |
| | | Scoop position | | | |

| | | | | | |
|---|---|---|---|---|---|
| Static Pressure (Ps)-mmWc | -197.40 | Pitot Tube Constant | 0.9 | Duct Dimensions - 3530 X 8840 | |
| Barometric Pre. (Ba)-mmHg | 760 | | | | |
| Abs.Pressure (Pu)-mmWc | 745.49 | Temperature(tu)$^0$ C | 132 | Density (Do) = 1.3 kg/Nm3 | |
| Pu=Ba+Ps/13.6 | 745.49 | Temperature(Tu)$^0$ K | 405 | Du = Do*(Pu/760)*(273/Tu) | 0.8596 |

| | Dynamic Pressure = Pd mmWC | | | | | | Velocity (Vu) = 4.4*K*Sq.rt.(Pd/Du) | | | | | | |
|---|---|---|---|---|---|---|---|---|---|---|---|---|---|
| | Transit point no./distance from end | | | | | | | | | | | | |
| Port No. → / Reading ↓ | 1 | 2 | 3 | 4 | 5 | | 1 | 2 | 3 | 4 | 5 | | Avg. Vel. m/sec |
| 1 | 13.2 | 3.6 | 5.6 | 6.6 | 0.8 | | 15.5 | 8.1 | 10.1 | 11.0 | 3.7 | | 9.68 |
| 2 | 13.5 | 5.8 | 3.0 | 6.9 | 5.6 | | 15.7 | 10.3 | 7.5 | 11.2 | 10.1 | | 10.95 |
| 3 | 9.9 | 1.3 | 2.8 | 4.6 | 8.9 | | 13.4 | 4.8 | 7.1 | 9.1 | 12.7 | | 9.45 |
| 4 | 11.7 | 2.0 | 2.3 | 0.8 | 8.9 | | 14.6 | 6.1 | 6.5 | 3.7 | 12.7 | | 8.72 |
| 5 | 5.8 | 3.3 | 3.0 | 2.0 | 5.1 | | 10.3 | 7.8 | 7.5 | 6.1 | 9.6 | | 8.25 |

Av. Pd = 5.48

Av. Vel. (m/sec) = 10.00

# Performance Certificate

JINDAL POWER

MECHWELL

To,

M/s. Mech-Well Industries Ltd
Malay, Plot No.607/201,
Anand C.H.S,Motawni Road,
Nasik Road,Nasik-422101,
Ph:0253-2453556,
Email Id : nasik@mechwell.org
Kind attn : Mr. Mehul Shah, Head CAE-Business Development.

Dear Sir,

**Reference is made to our Work order nos.**
**1. Order No : 4400005556 CFD Analysis (4x250 MW) and duct test Dt.17.08.2013**
**2. Order No : 4100007559 Supply of Diverter plates for Duct Dt.07.11.2013**
**3. Order No : 4400006000 Work of Erection of Diverter Plates Dt. 07.11.2013**

**Subject: CFD ANALYSIS FROM APH OUTLET TO ID FAN INLET.**

In reference to aforesaid subject we wish to appreciate and acknowledge the successful completion of project in Unit-3 of 4 x 250 MW Jindal Power Ltd, Tamnar for the above mentioned work orders.

| Parameters | Before CFD Analysis | After Implementation of CFD Analysis | Result after 15 month continuous running |
|---|---|---|---|
| Load (MW) | 248 MW | 250MW | 250MW |
| ID FAN-A | 191Amp | 141 Amp | 148 Amp |
| ID FAN-B | 196Amp | 143 Amp | 149 Amp |

Your technical approach was very effective and the whole job was carried out in a systematic manner in stipulated time frame of 15 days of annual shutdown of unit-3
Our best wishes for their future endeavours.

Thanking you,

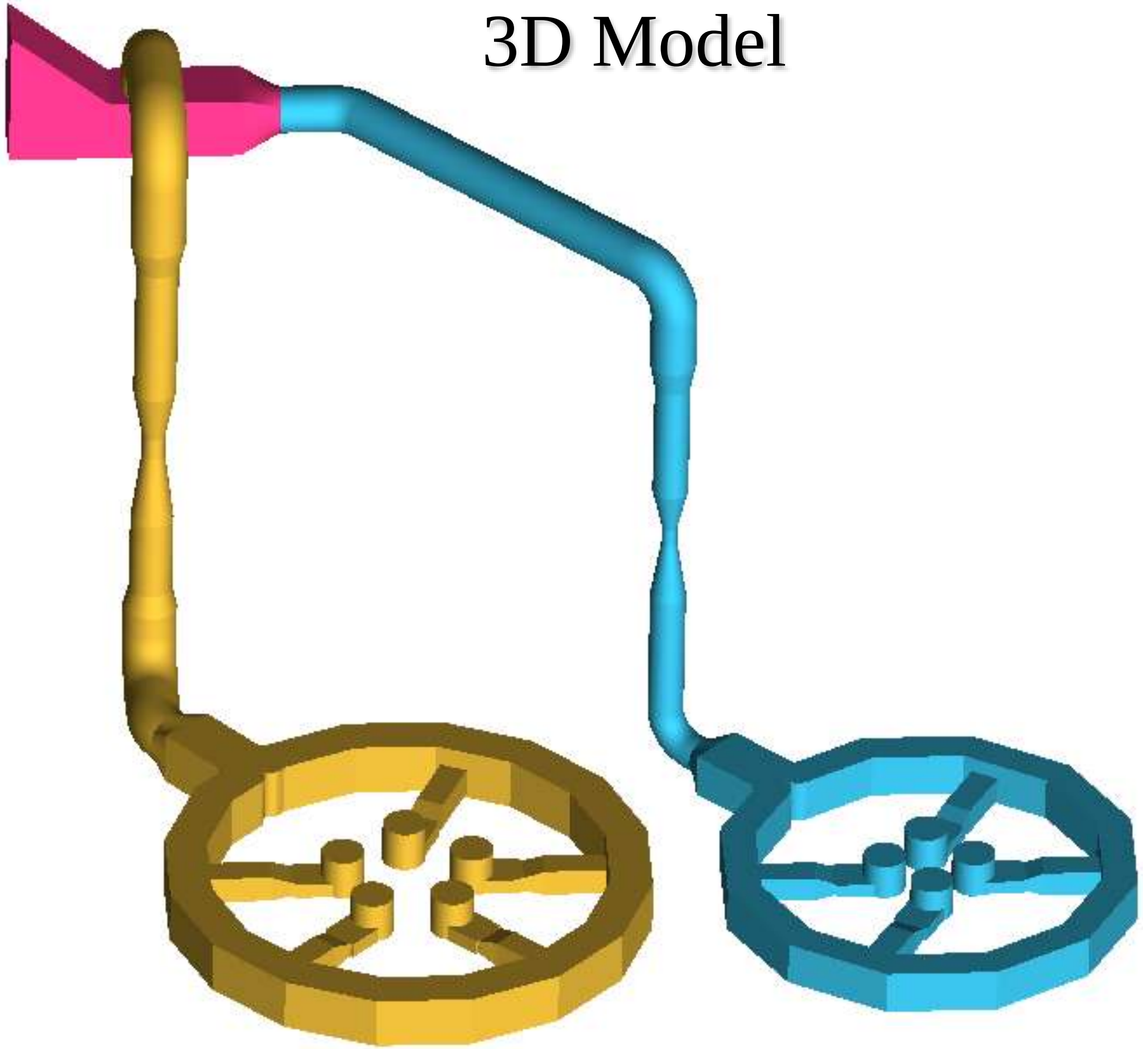
Yours faithfully,

14.7.15
(Rafique Ahmed)
AVP

Jindal Power Ltd
Registered Office Tamnar 496 107, District Raigarh, Chhattisgarh
T +91 7767 281701 F +91 7767 281995 W www.jindalpower.com

# *CFD Analysis of Combustion Air Ducting of NHT Heaters for flow distribution in burners*

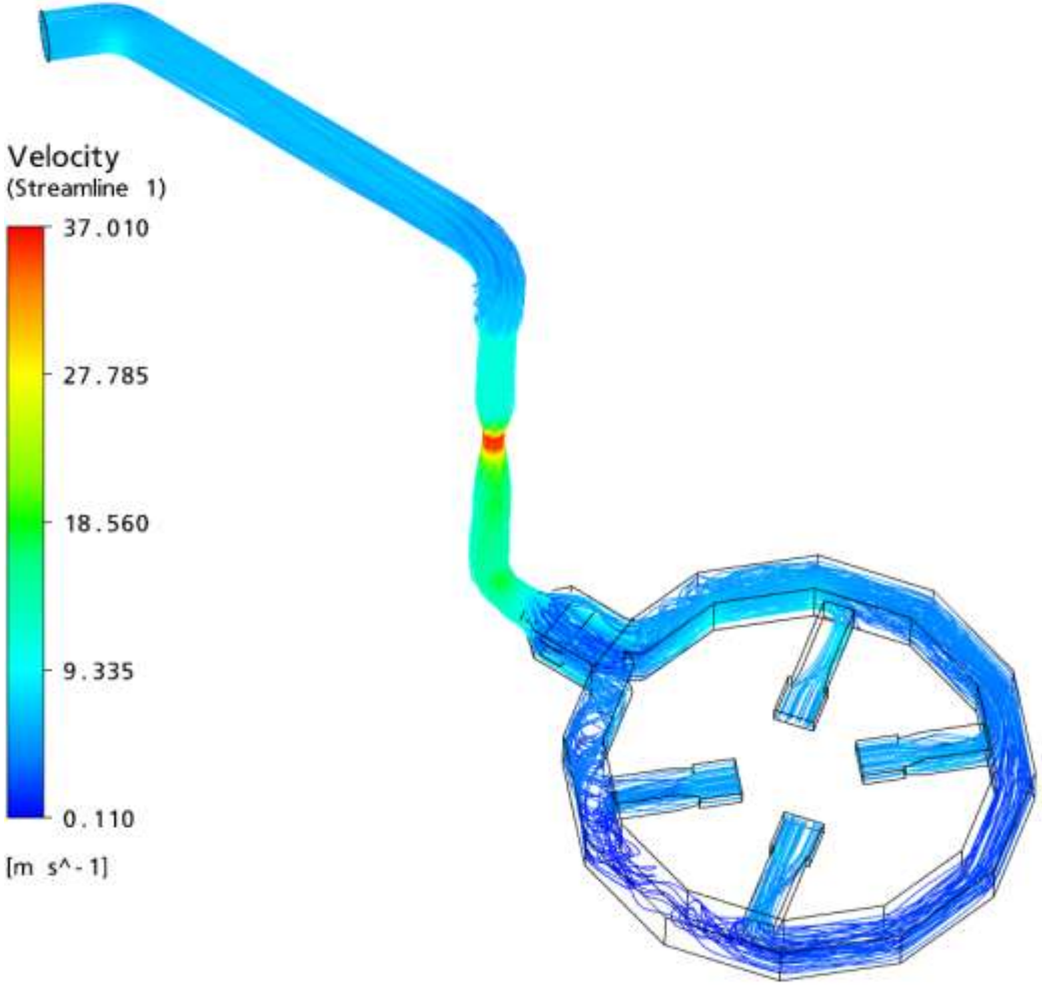
3D Model

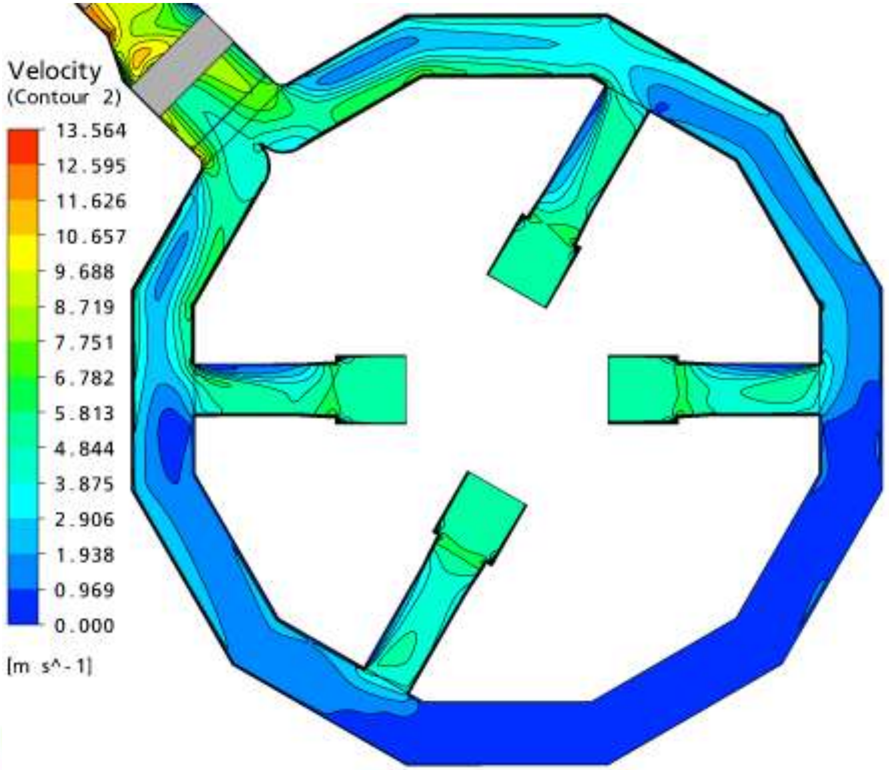


# CFD Results for140-F001 Duct_Case-I

## CASE I

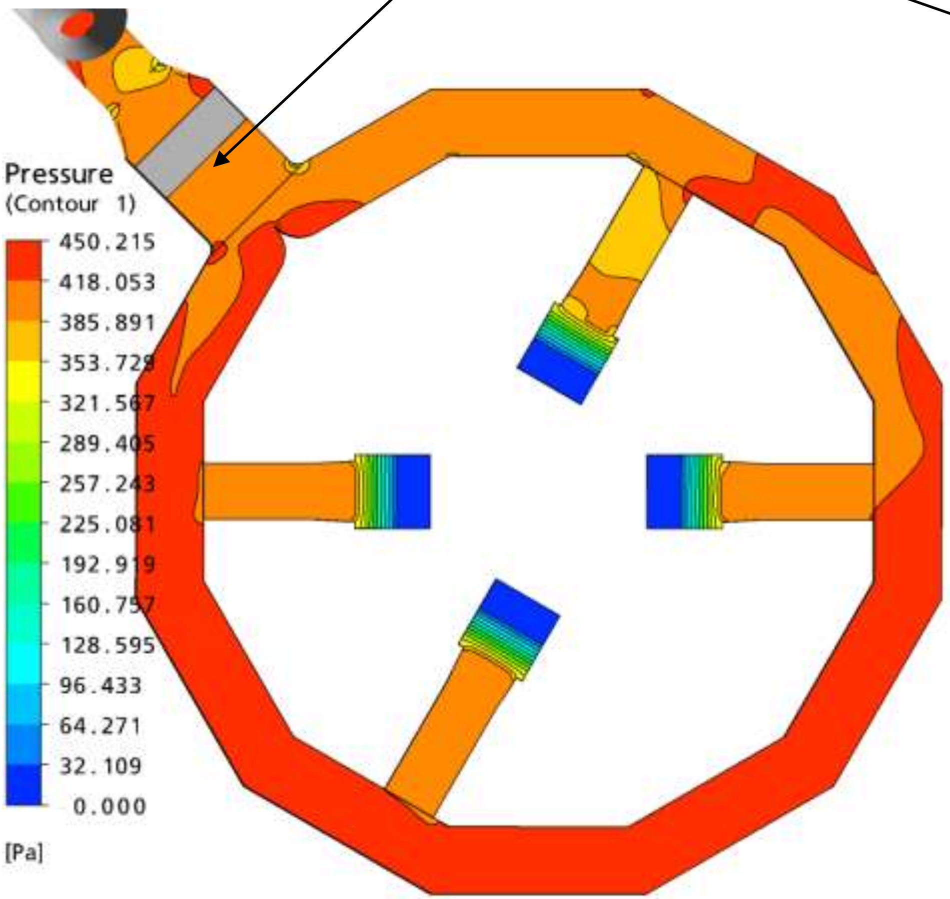
### Flow Rate:15350 Kg/hr at Temp. 9 C   & Pressure at Each burner inlet: 39 mmWC



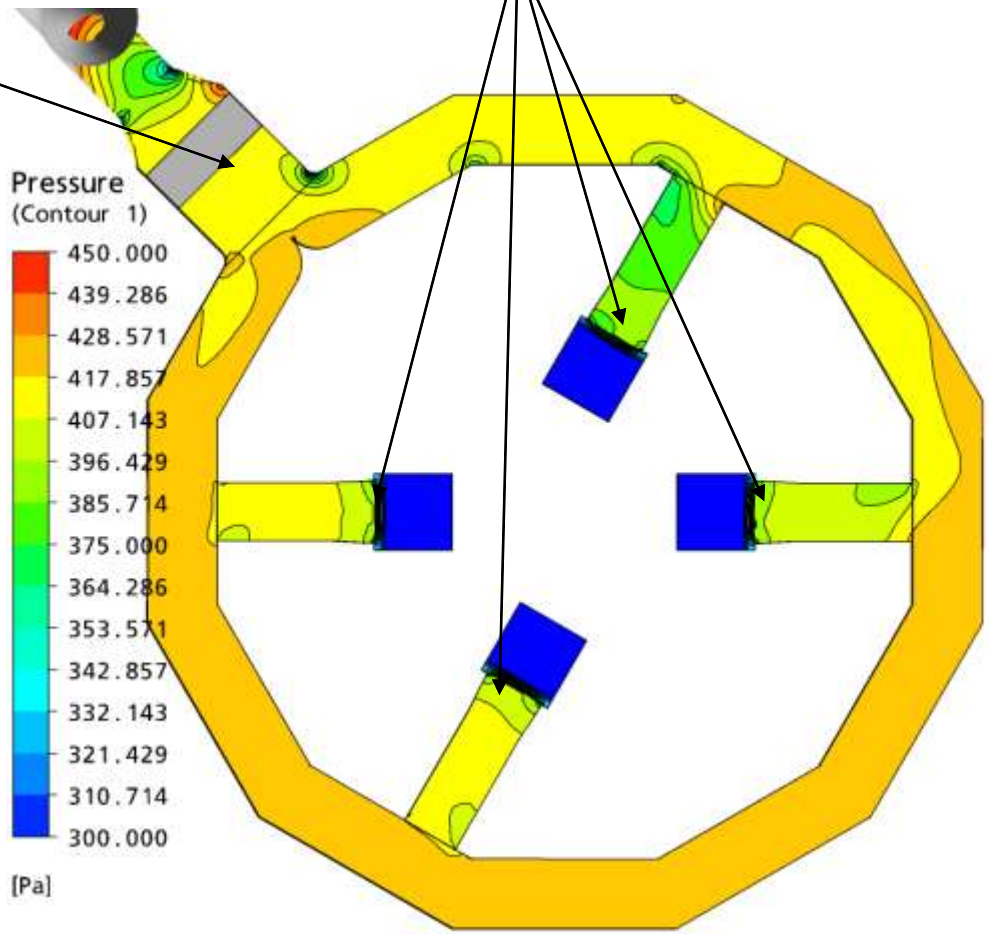



Static Pressure: 41.2 mm WC

Static Pressure : 39 mm WC

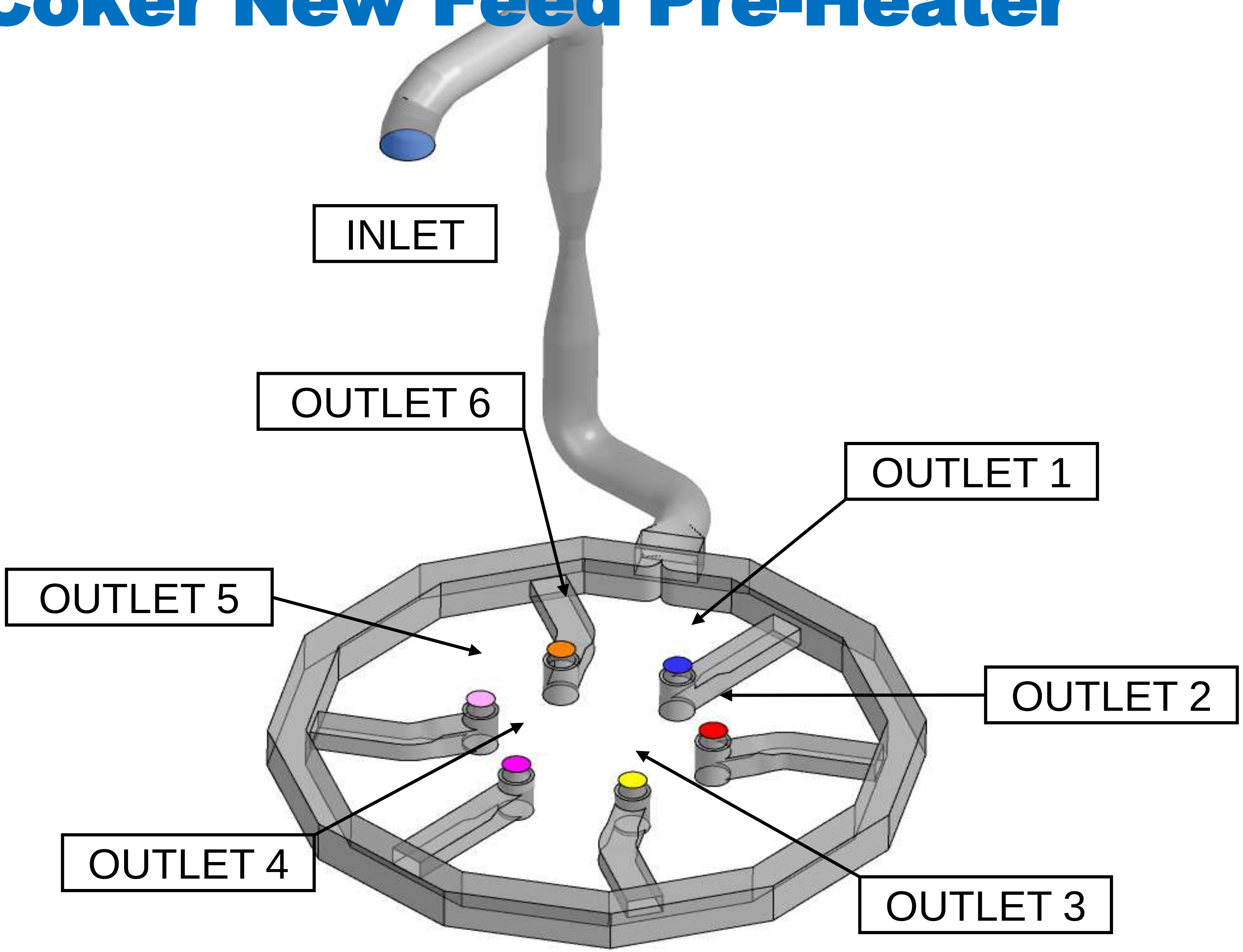



Pressure Contour Plot in Plenum duct(At the center plane of Plenum Duct)

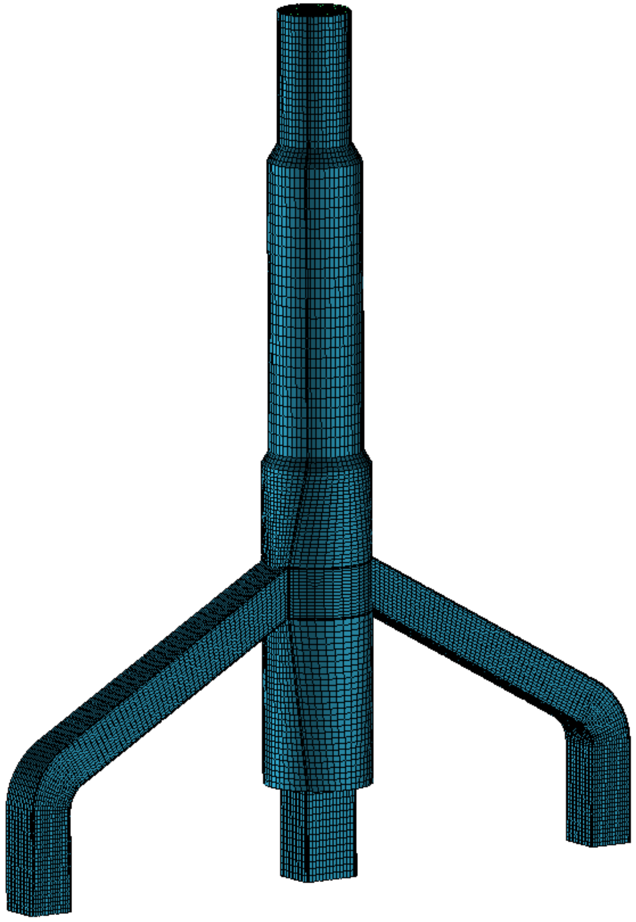



Pressure Contour Plot in Plenum duct(At the center of plane of Plenum Duct) with user specified values

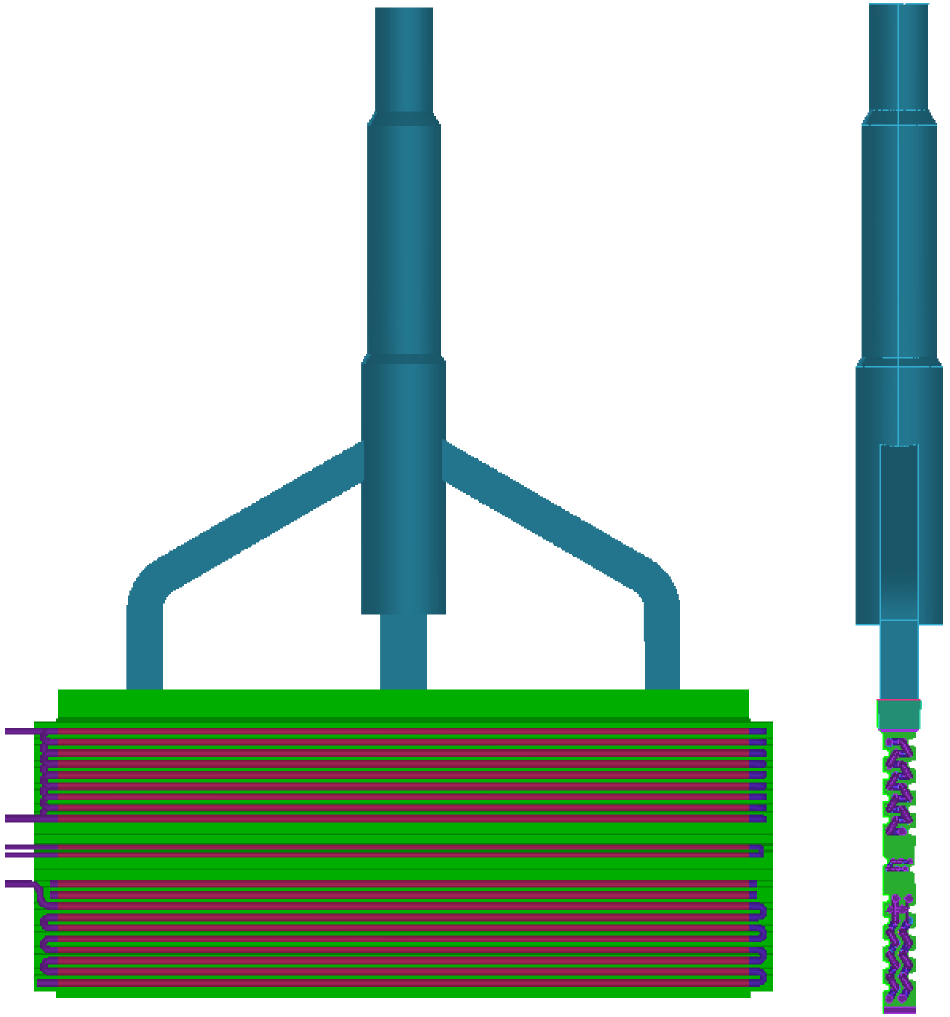
# CFD Analysis of Flow in CA Ducting of Coker New Feed Pre-Heater



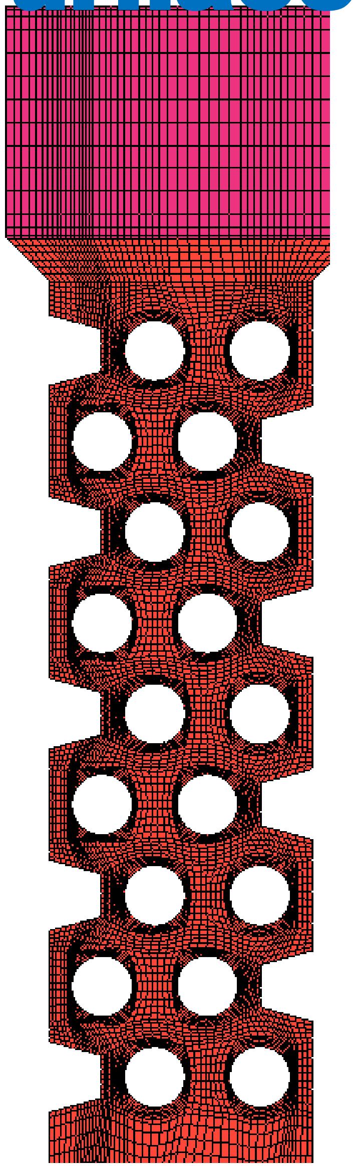
# CFD analysis for EDC Cracker Furnace duct



Elevation

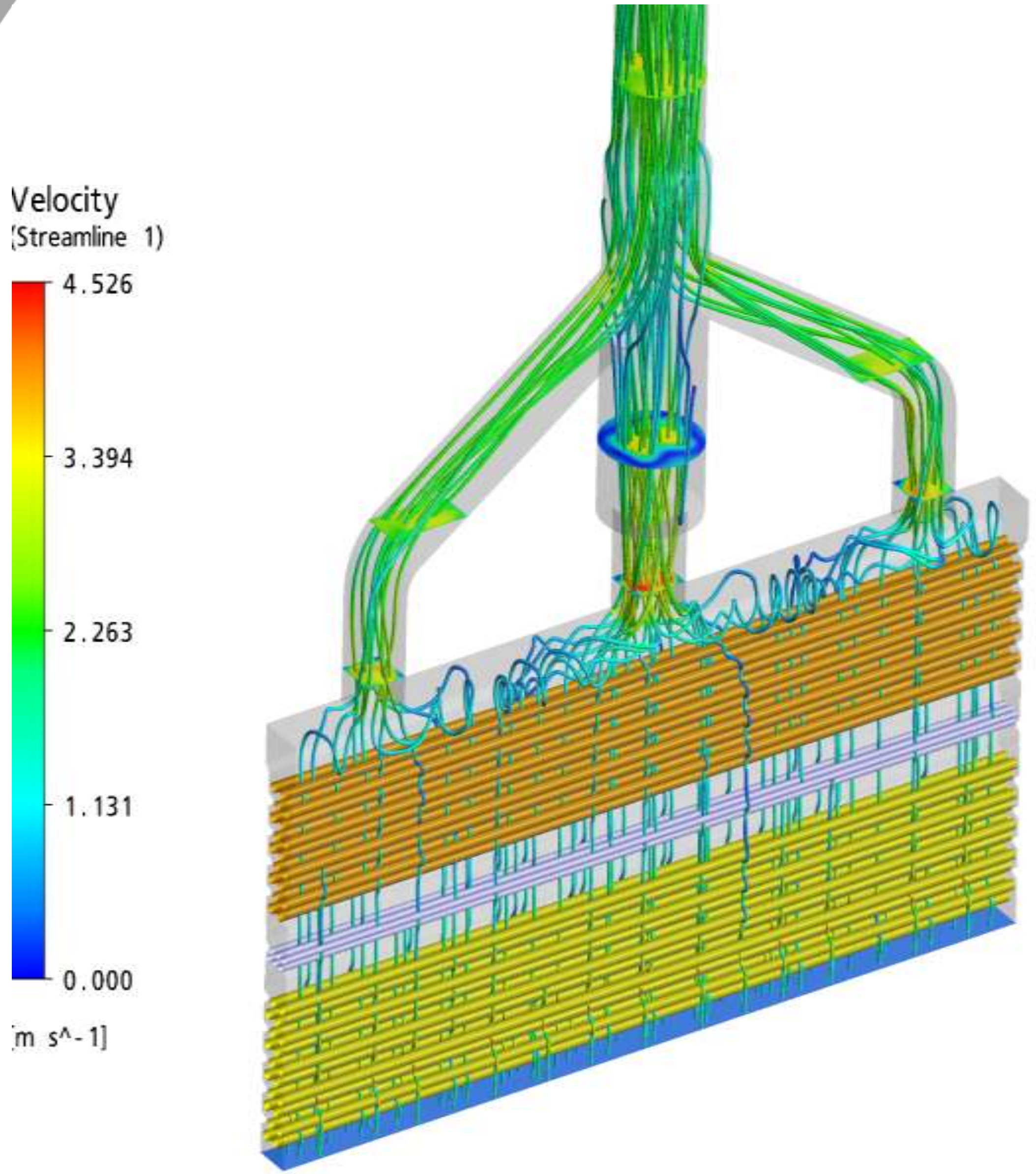


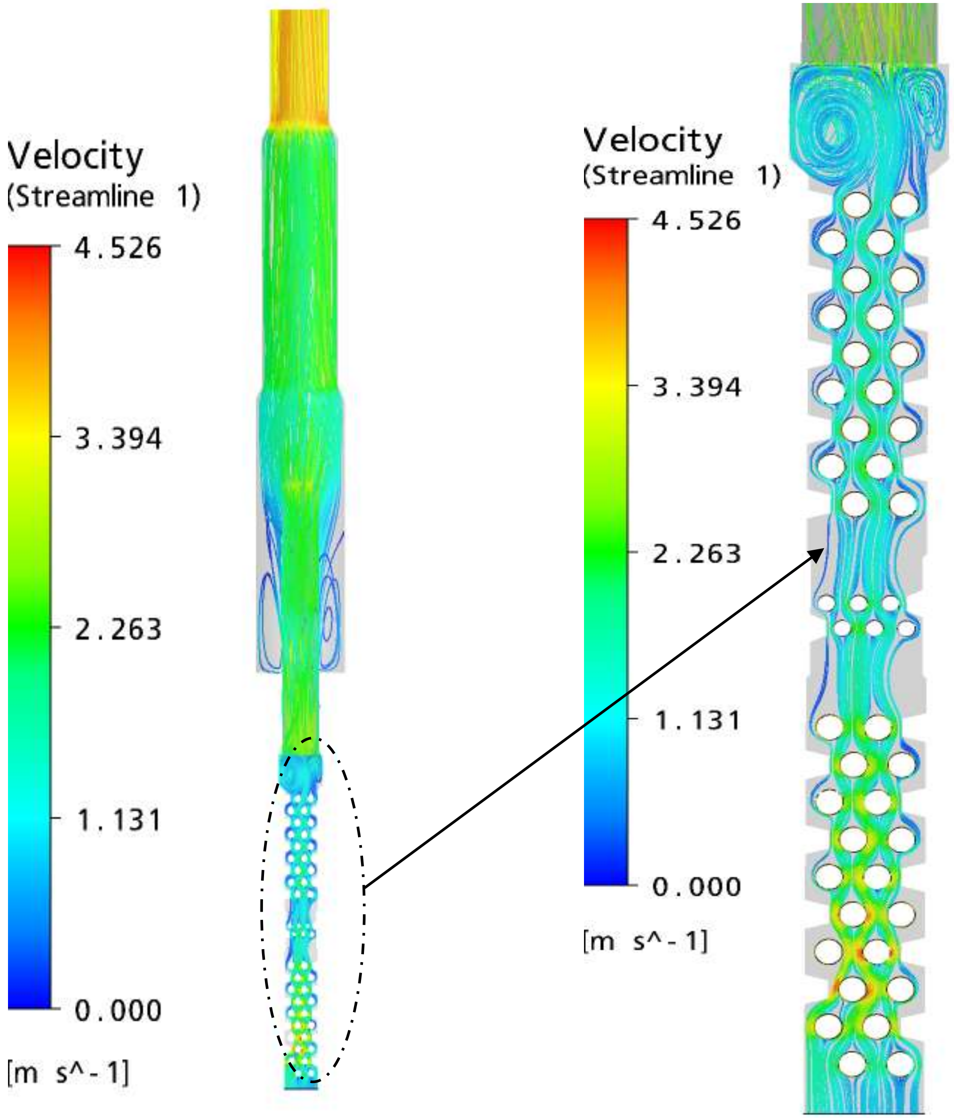
Side View



Mesh

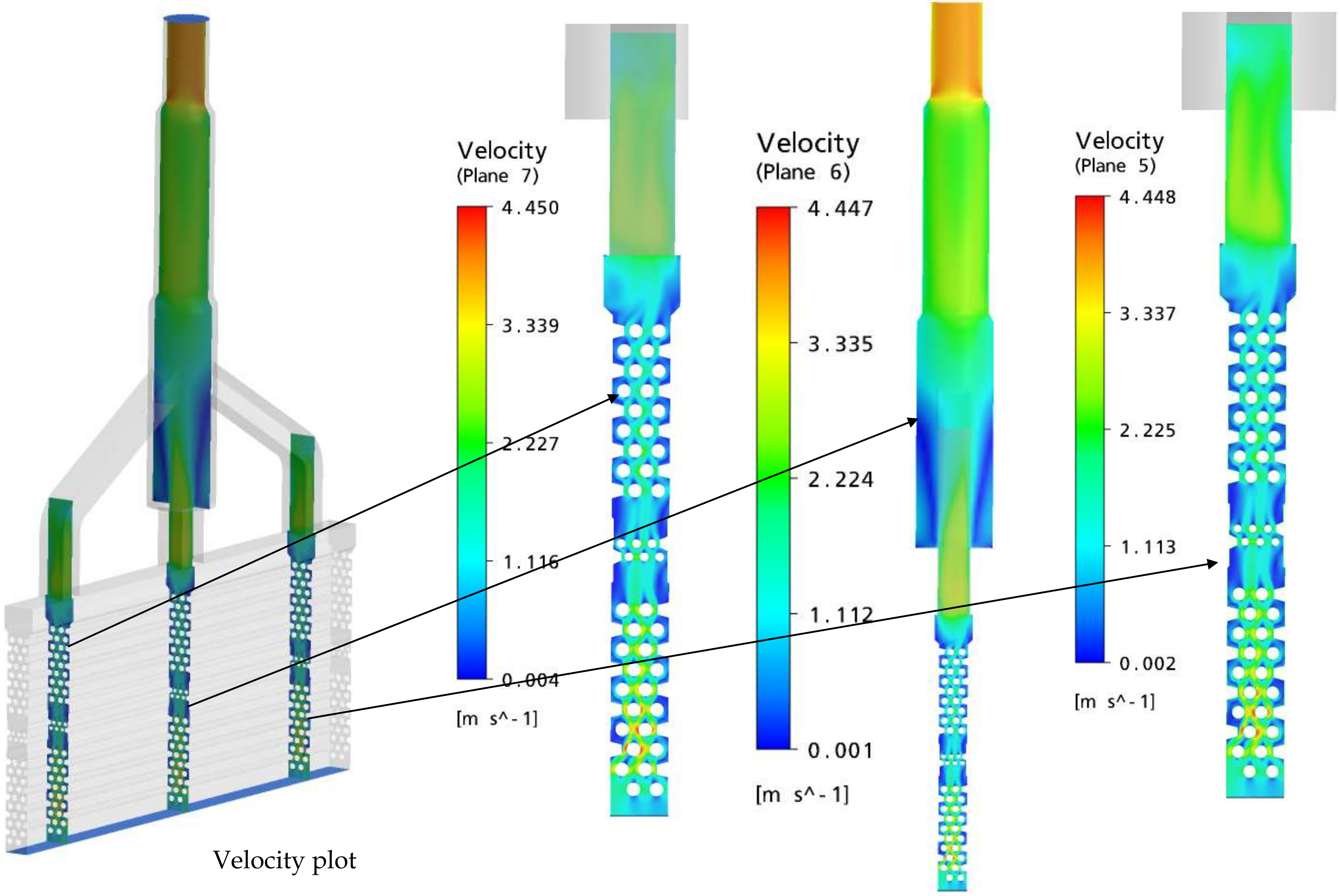
# CFD results - Streamline




Streamline plot

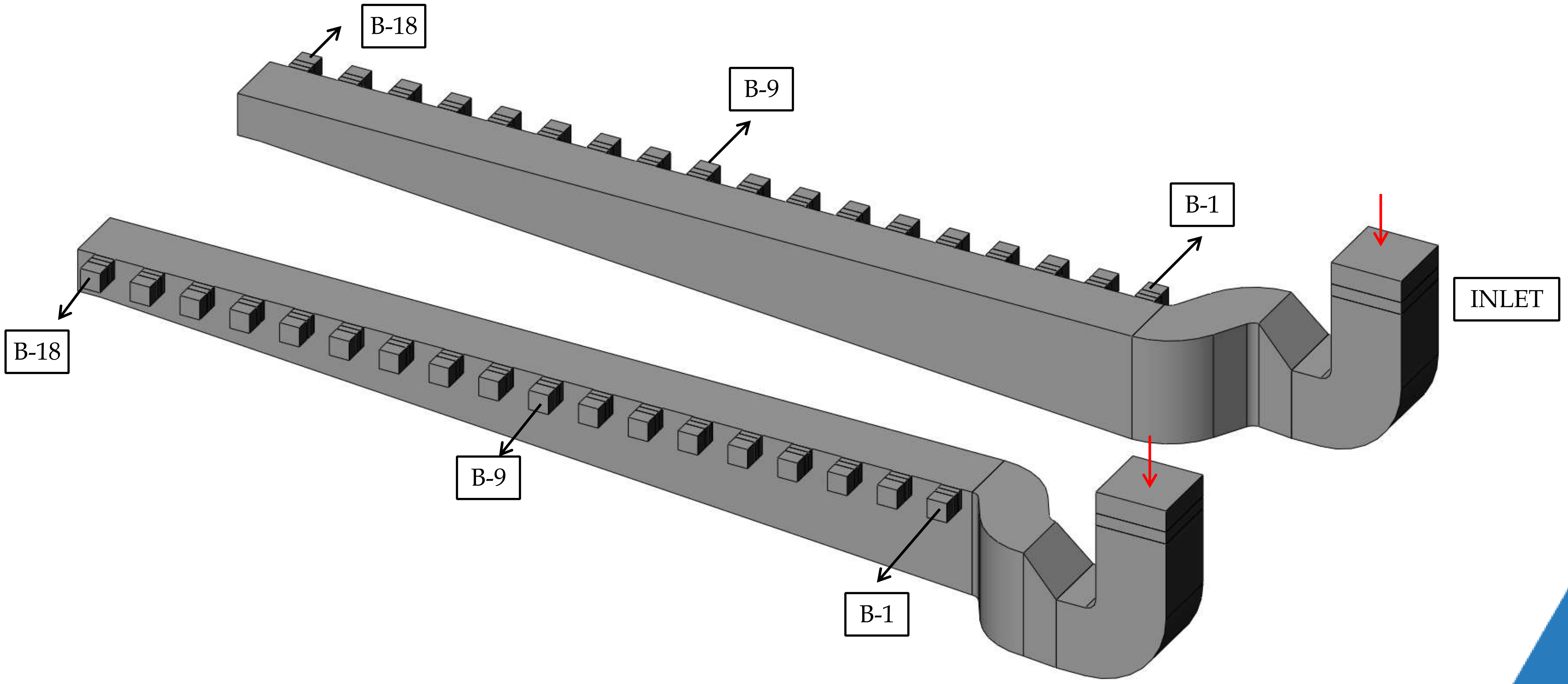
Streamline plot



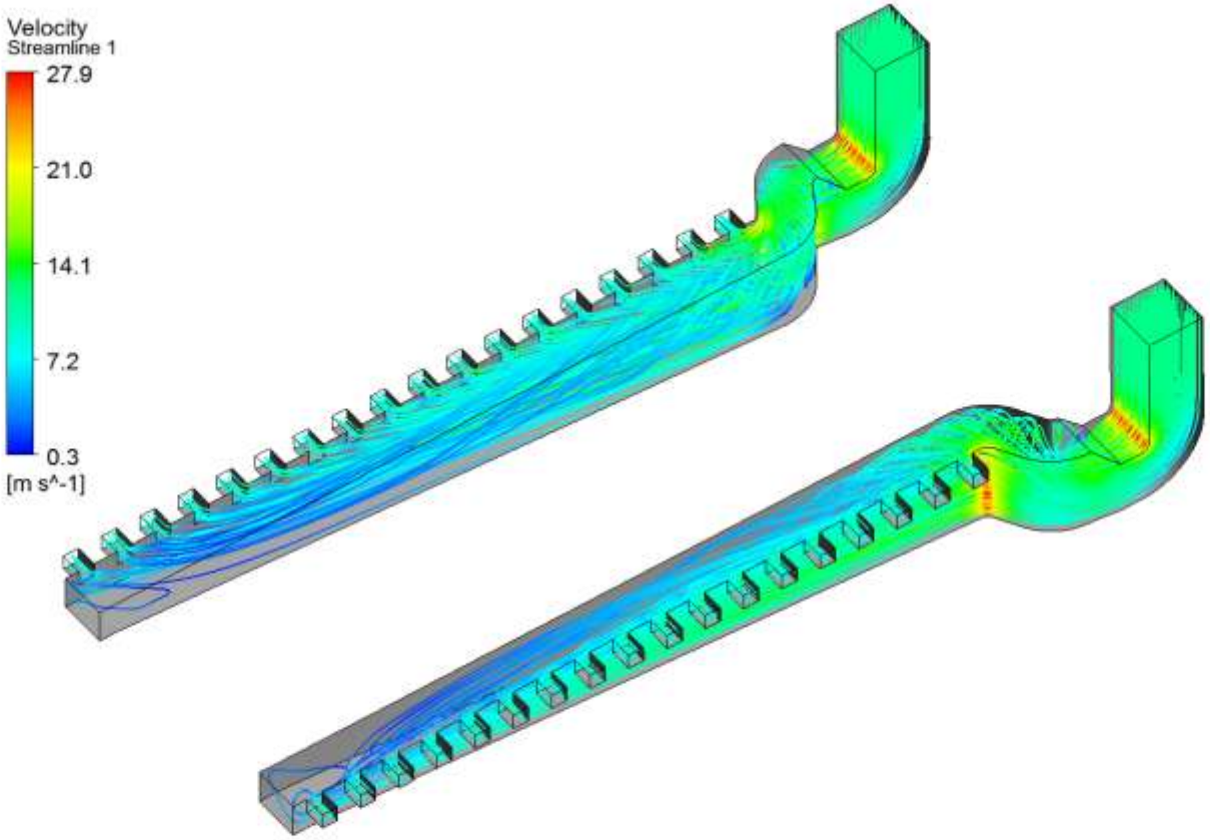

Velocity plot

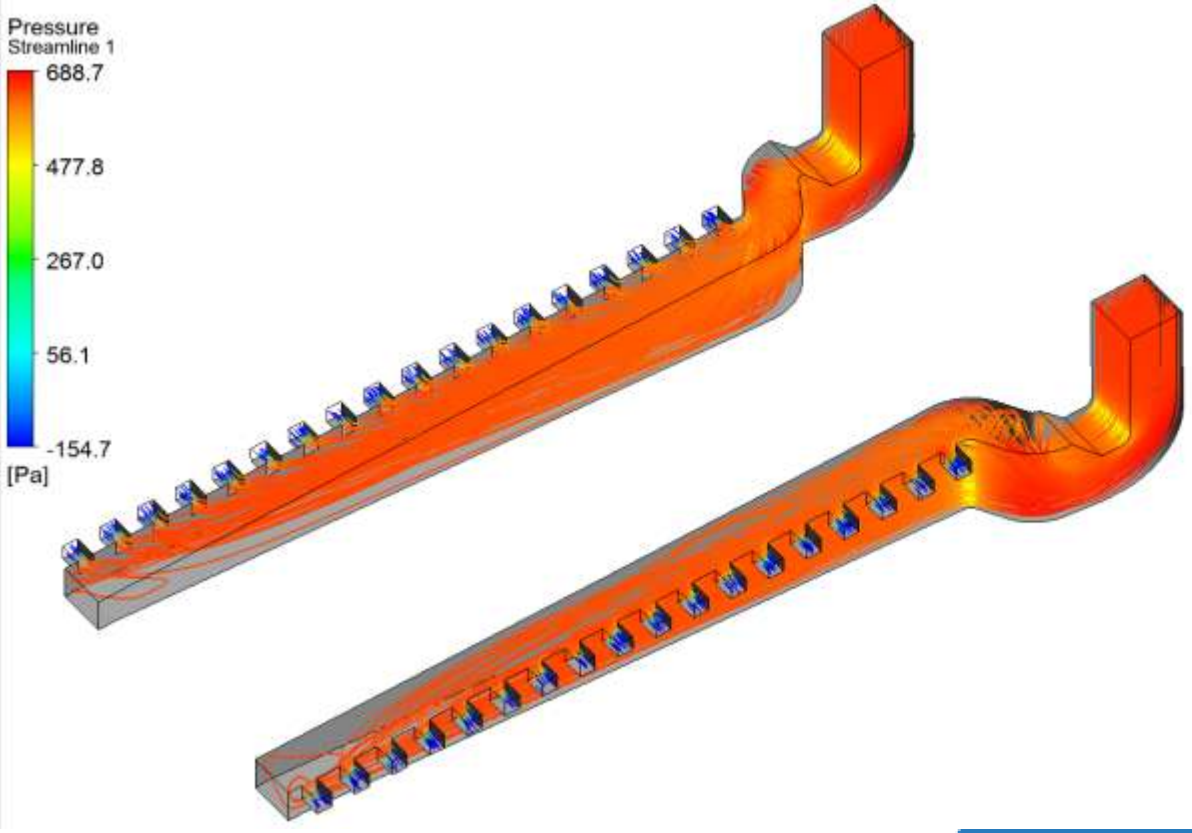
Detailed Velocity plot at various location

# 3D Cad Model for Crude Heaters

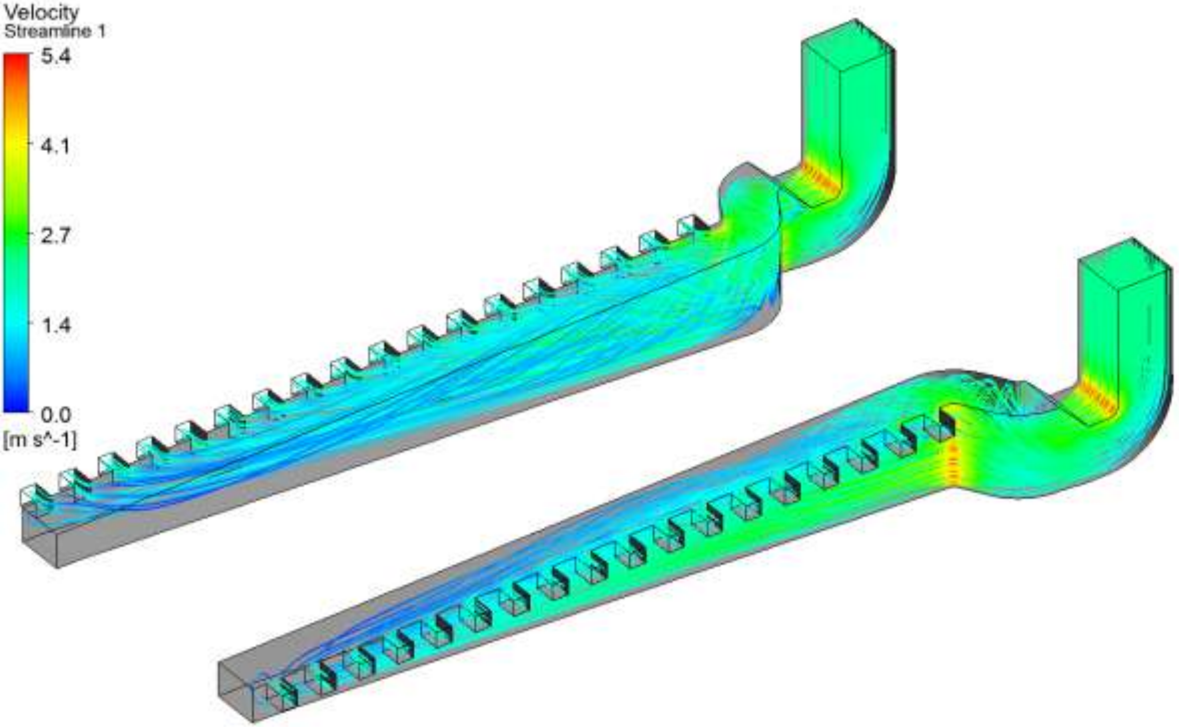


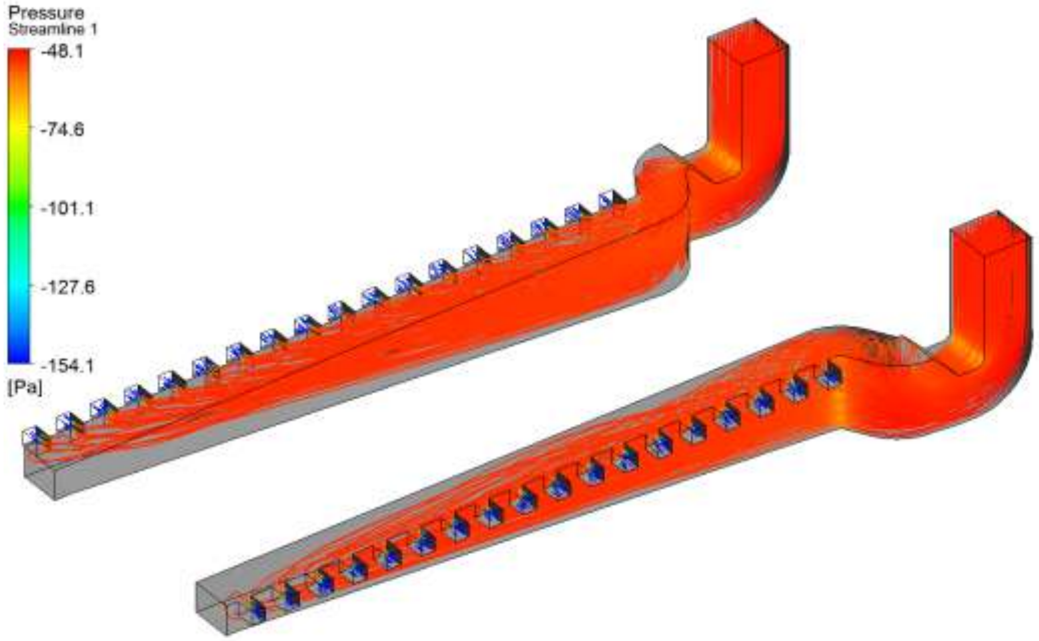
# VELOCITY STREMLINES INSIDE COMBUSTION AIR DUCT – DESIGN CASE



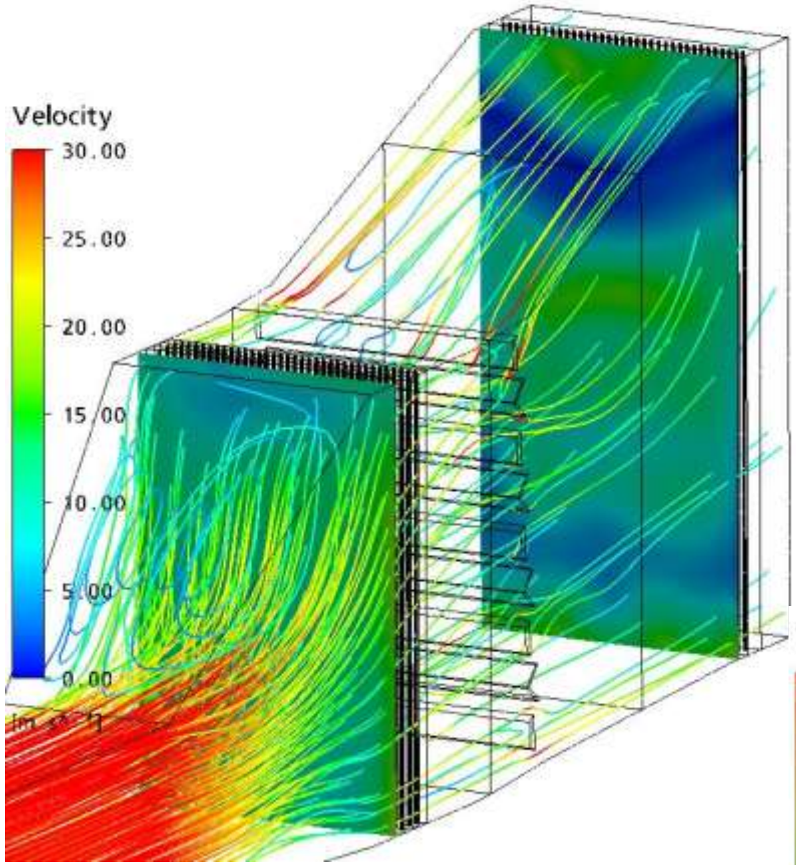



# CFD REULTS FOR Minimum Case

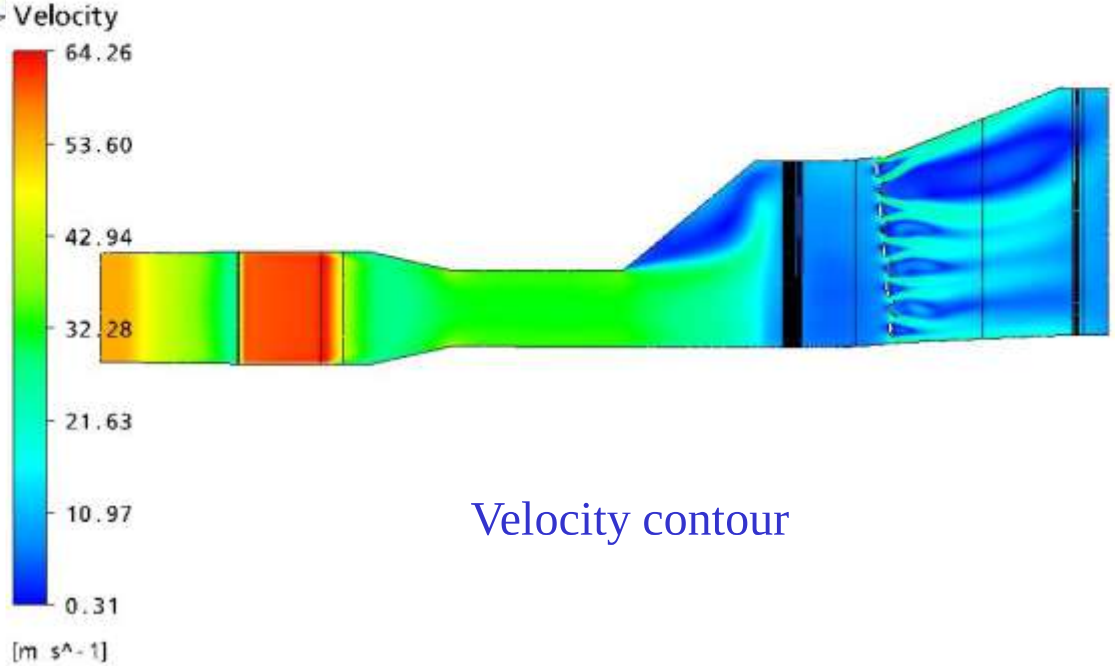


# CFD Analysis of HRSG System

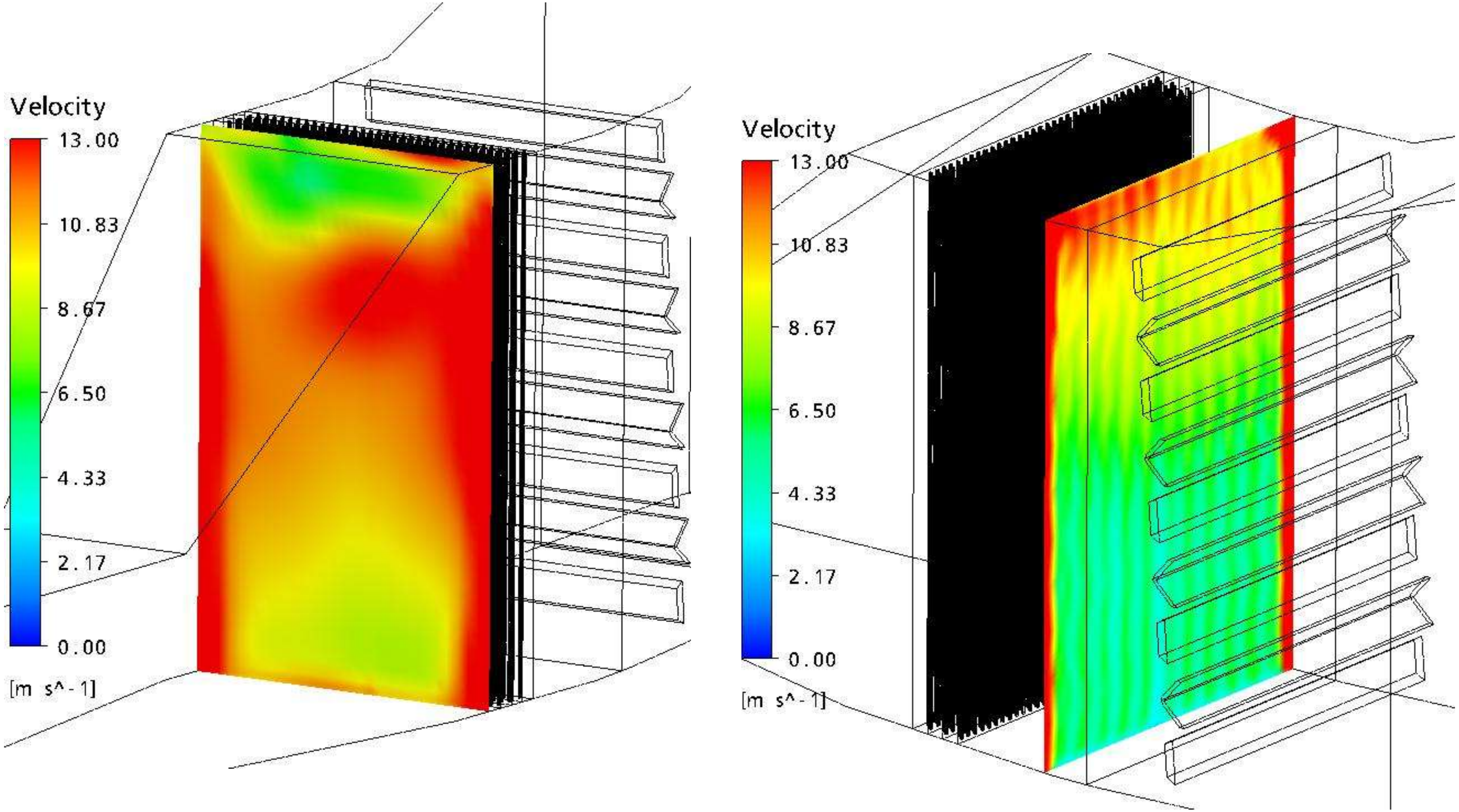



Streamline and Velocity Contour



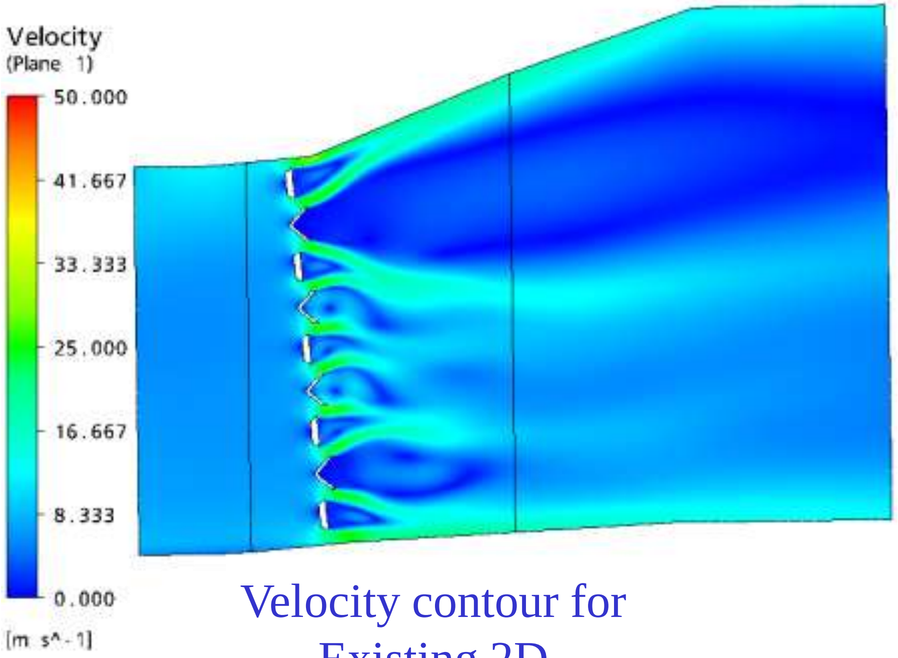

Velocity contour

# CFD Results

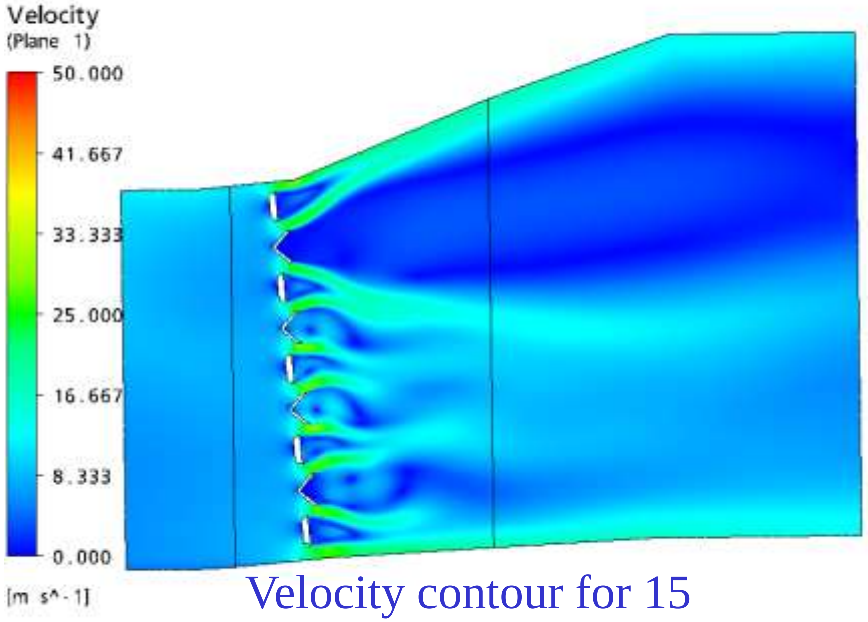



Velocity contour

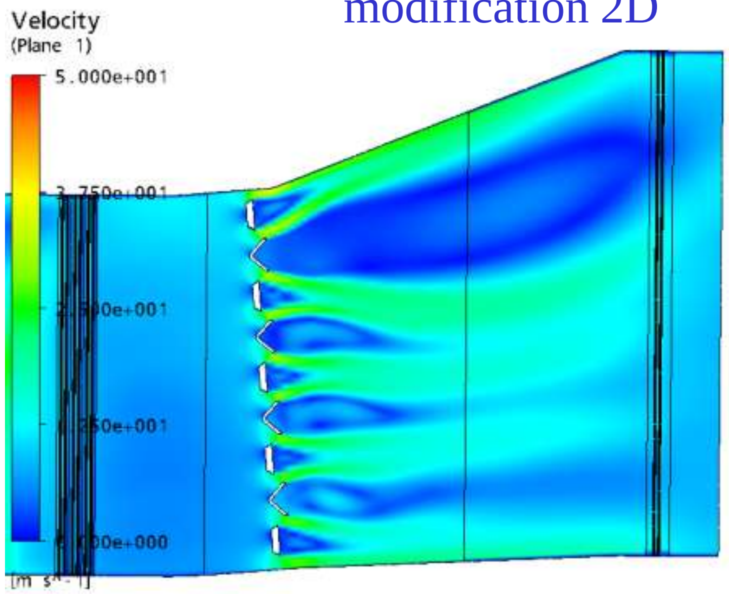
# Velocity Pattern




Velocity contour for Existing 2D

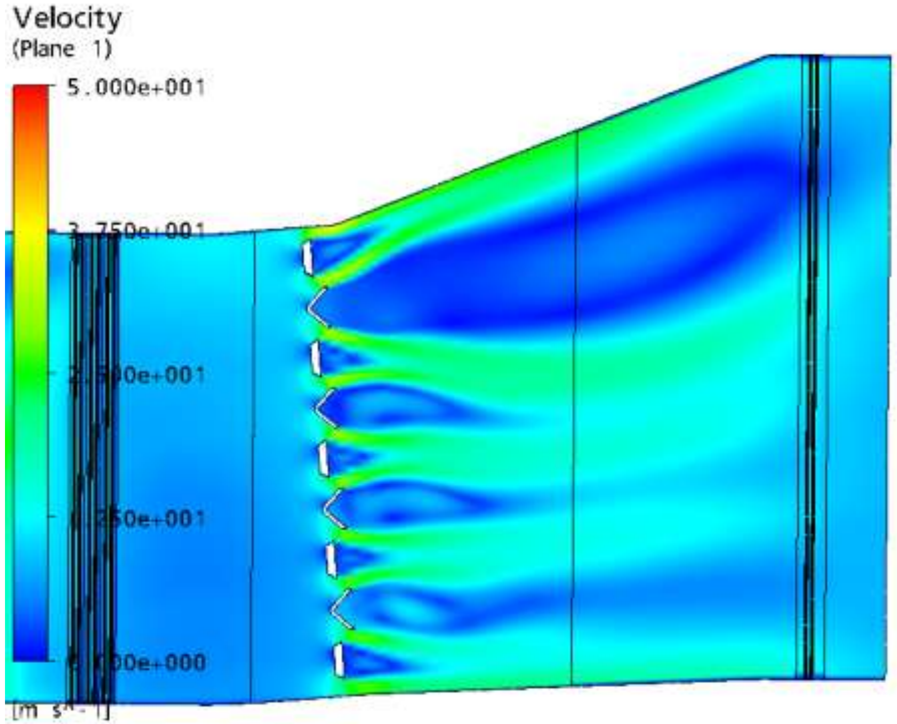


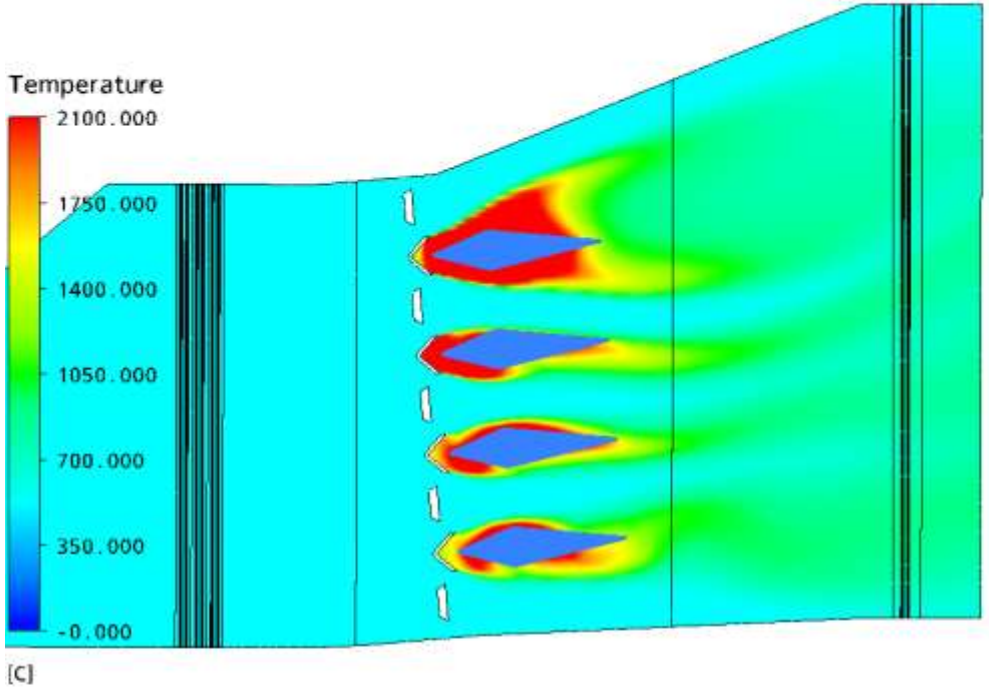

Velocity contour for 15 modification 2D




Velocity contour for 15 modification 3D

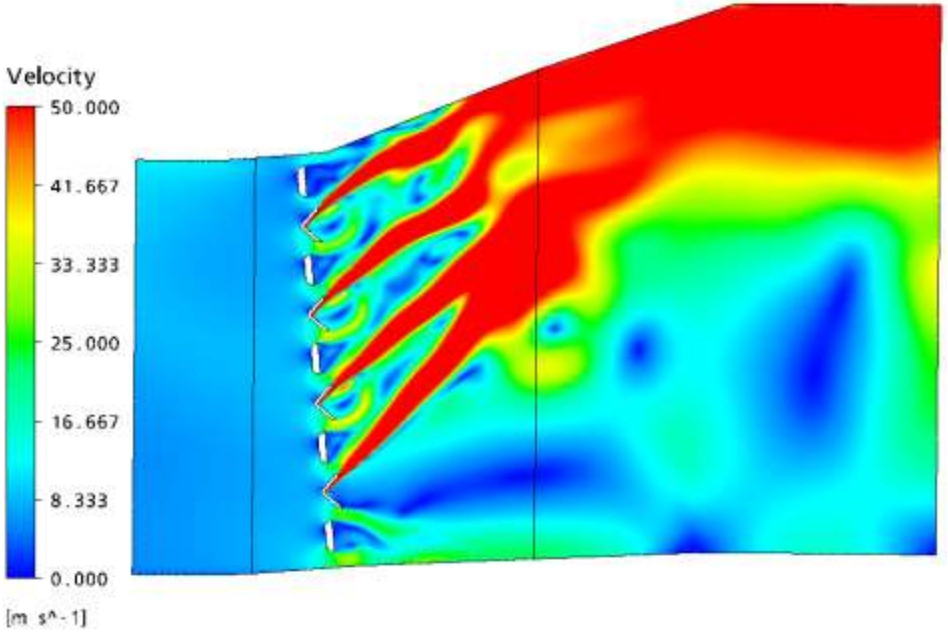
# CFD Results after adding heat source




Temperature contour

# Velocity and temperature Contour



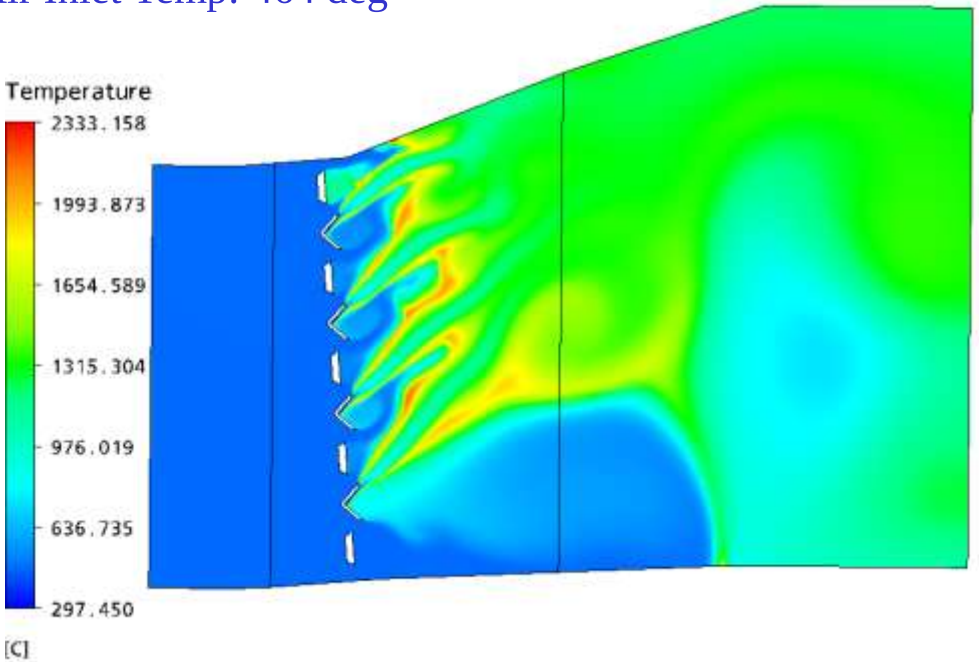

Velocity contour

Velocity profile is extracted from the 3D simulation and used in case of 2D analysis for 15 modification.

Fuel Flow : 0.088839 Kg/s

Air Flow : 0.227579 Kg/s

Air Inlet Temp: 464 deg

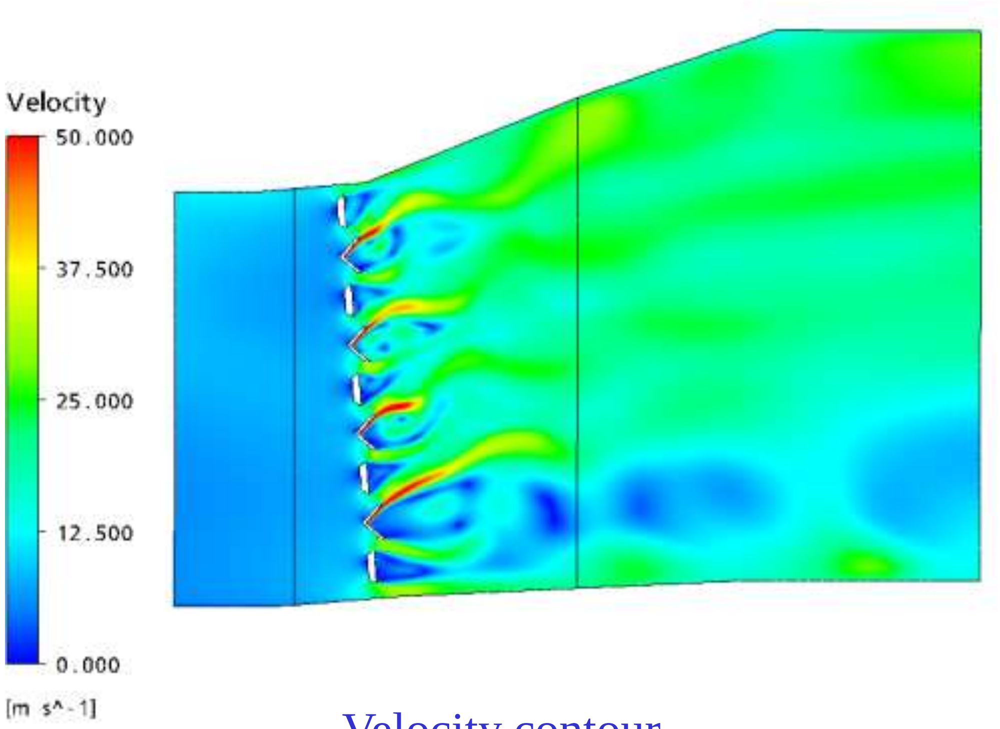



Temperature contour

Fuel Flow : 0.029613 Kg/s

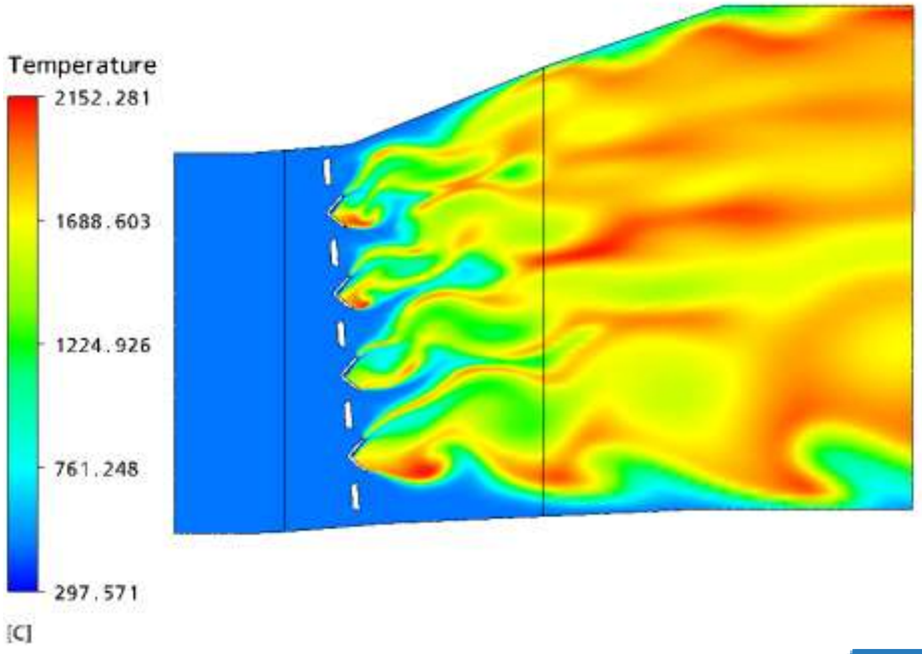
Air Flow : 0.227579 Kg/s

Air Inlet Temp: 464 deg




Velocity contour

Velocity profile is extracted from the 3D simulation and used in case of 2D analysis for 15 modification.



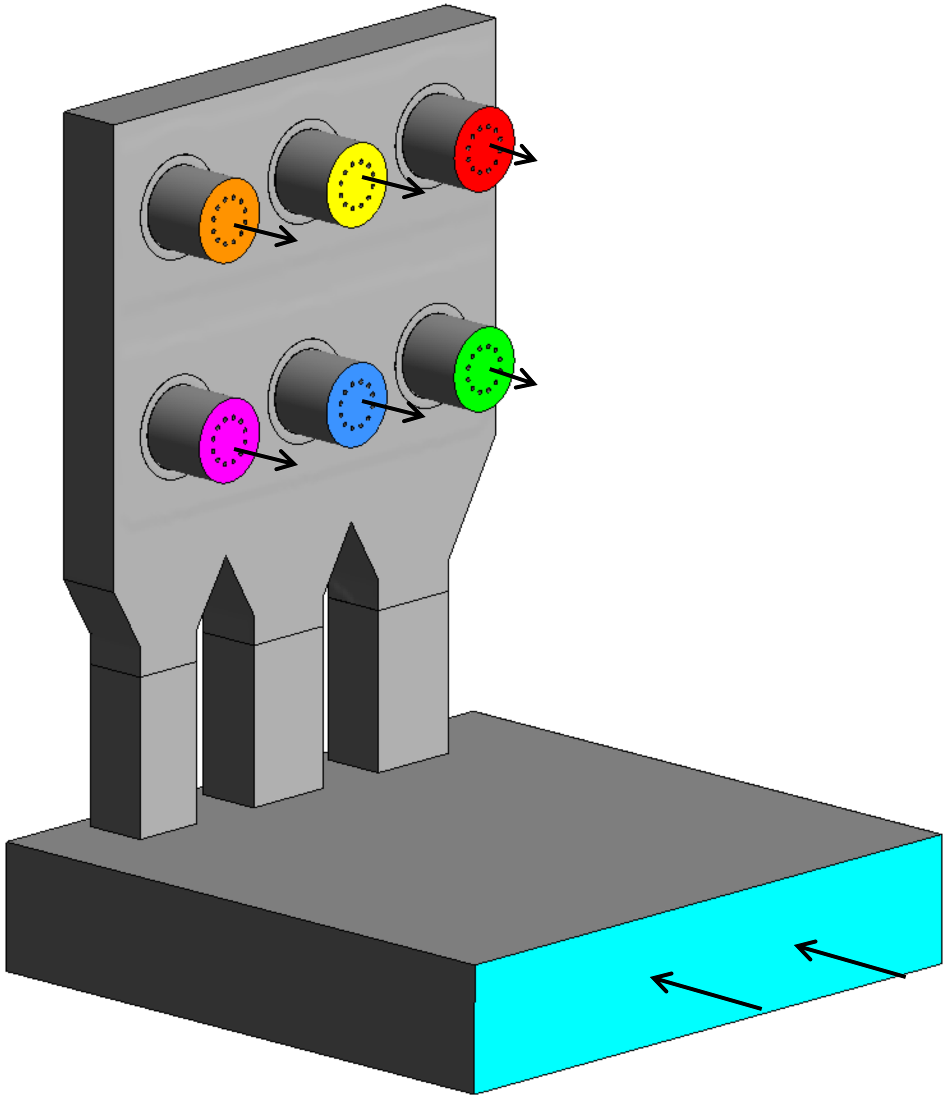

Temperature contour

# OBJECTIVE

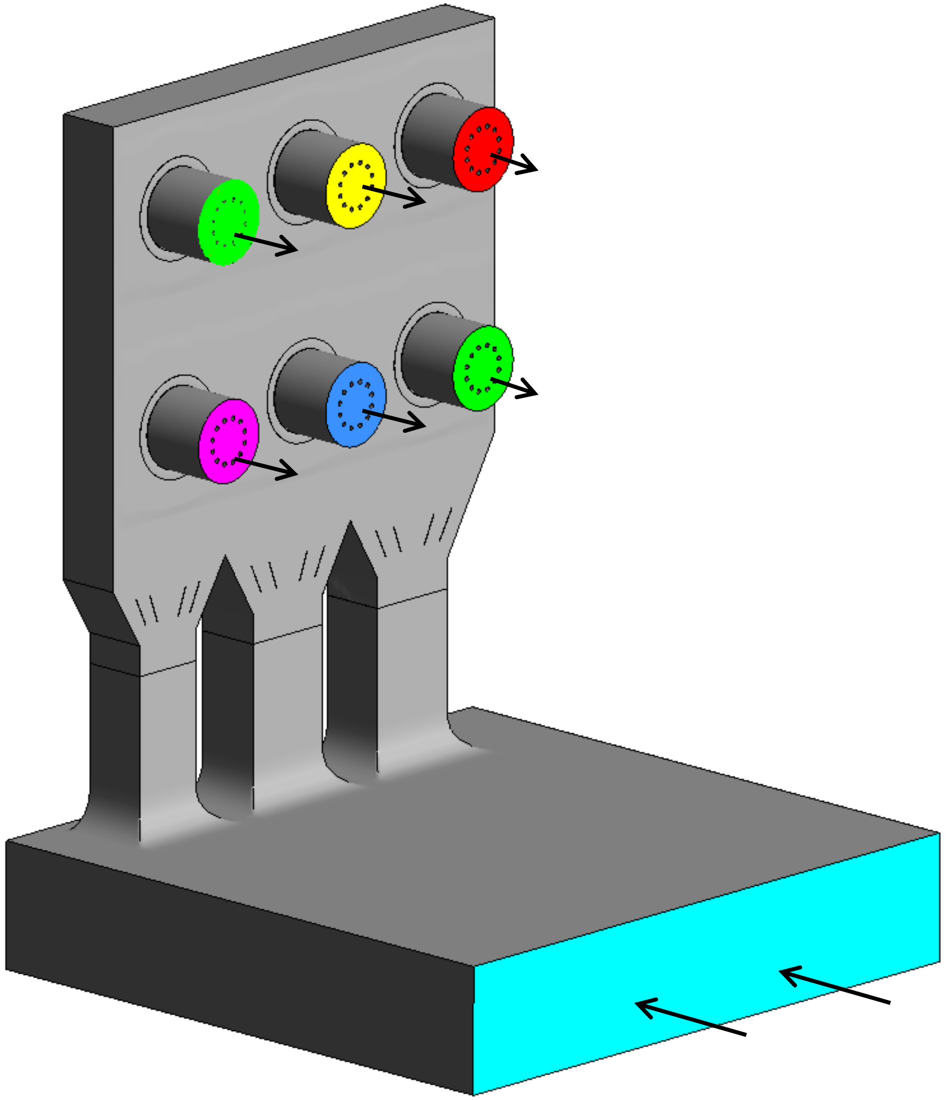
Objective of the project are as follows:

- Turbulence which will been minimized by avoiding flow separation and recirculation using CFD technique
- Supply of correct quantity of secondary air to each burner to be ensured and Reduce Pressure drop across APH to wind box (ducting's) as compare to existing system which will result in reducing /maintaining wind box pressure to improve power saving of FD fan.
- The simulation domain to focus on equating high velocity zones to achieve uniform stratification and reduction in pressure drop and turbulence caused by uneven air flow distribution Detailed flow modeling for air/fuel and mixtures

# 3D MODEL

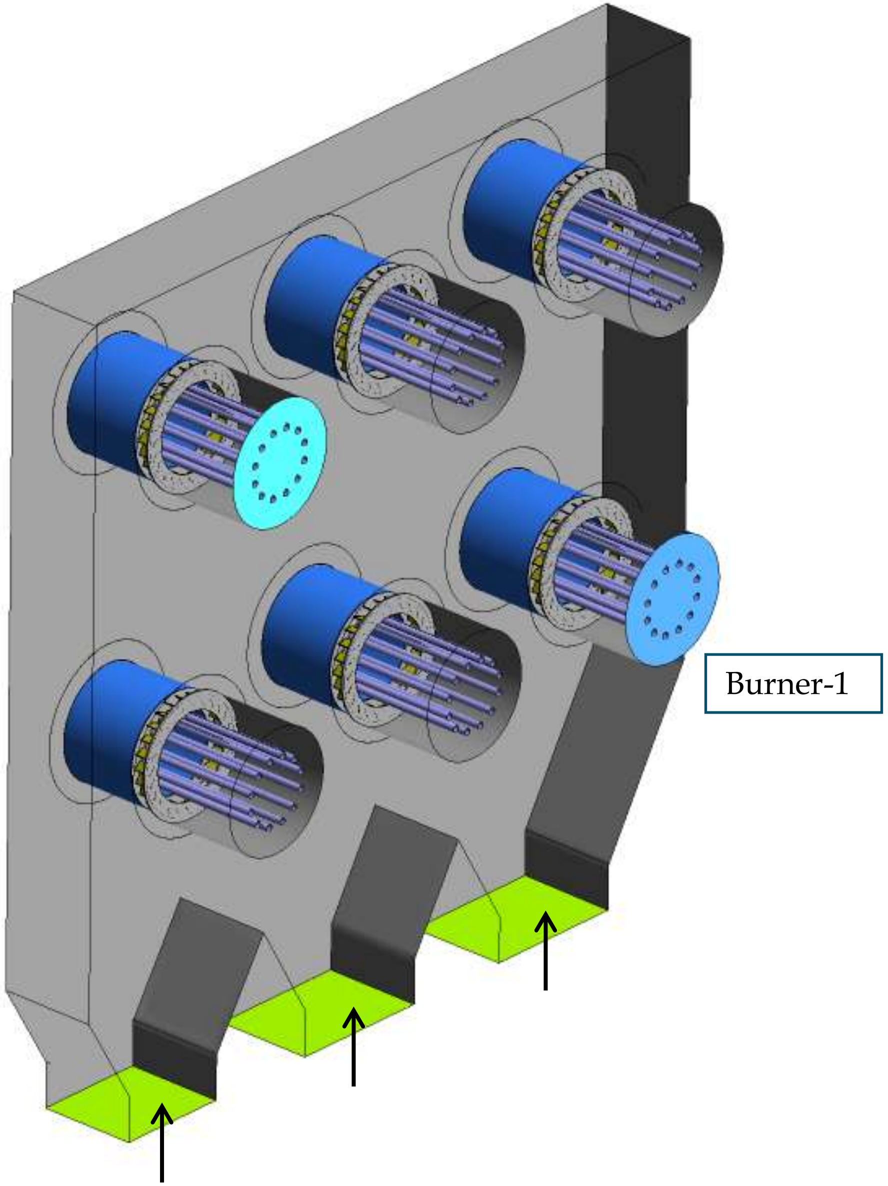


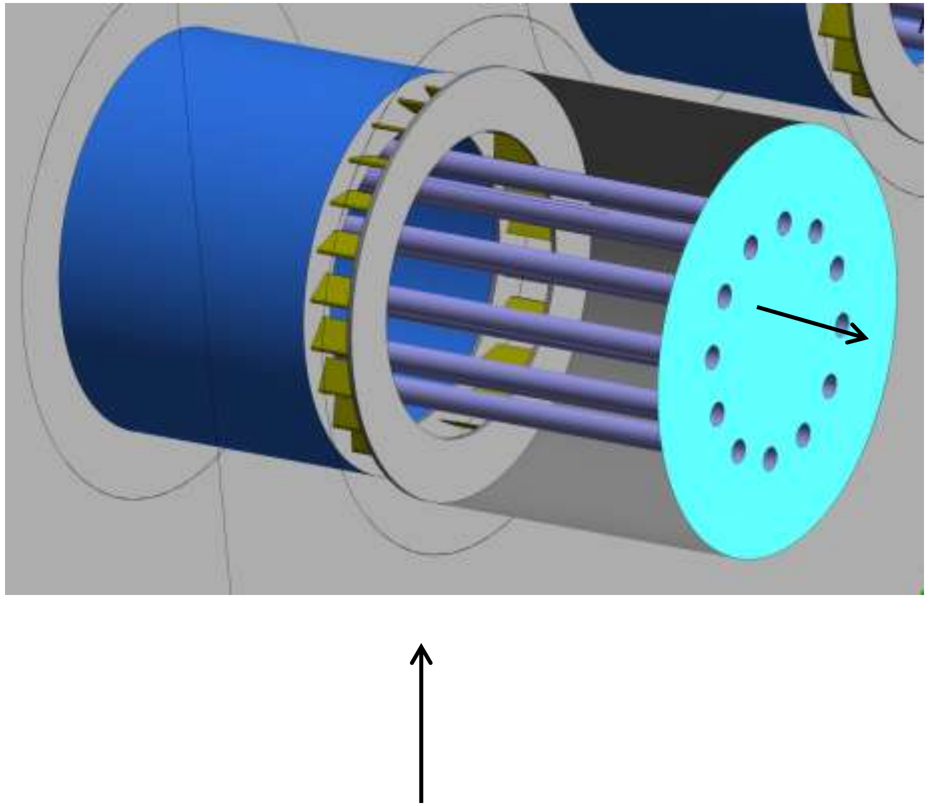
EXISTING



MODIFIED

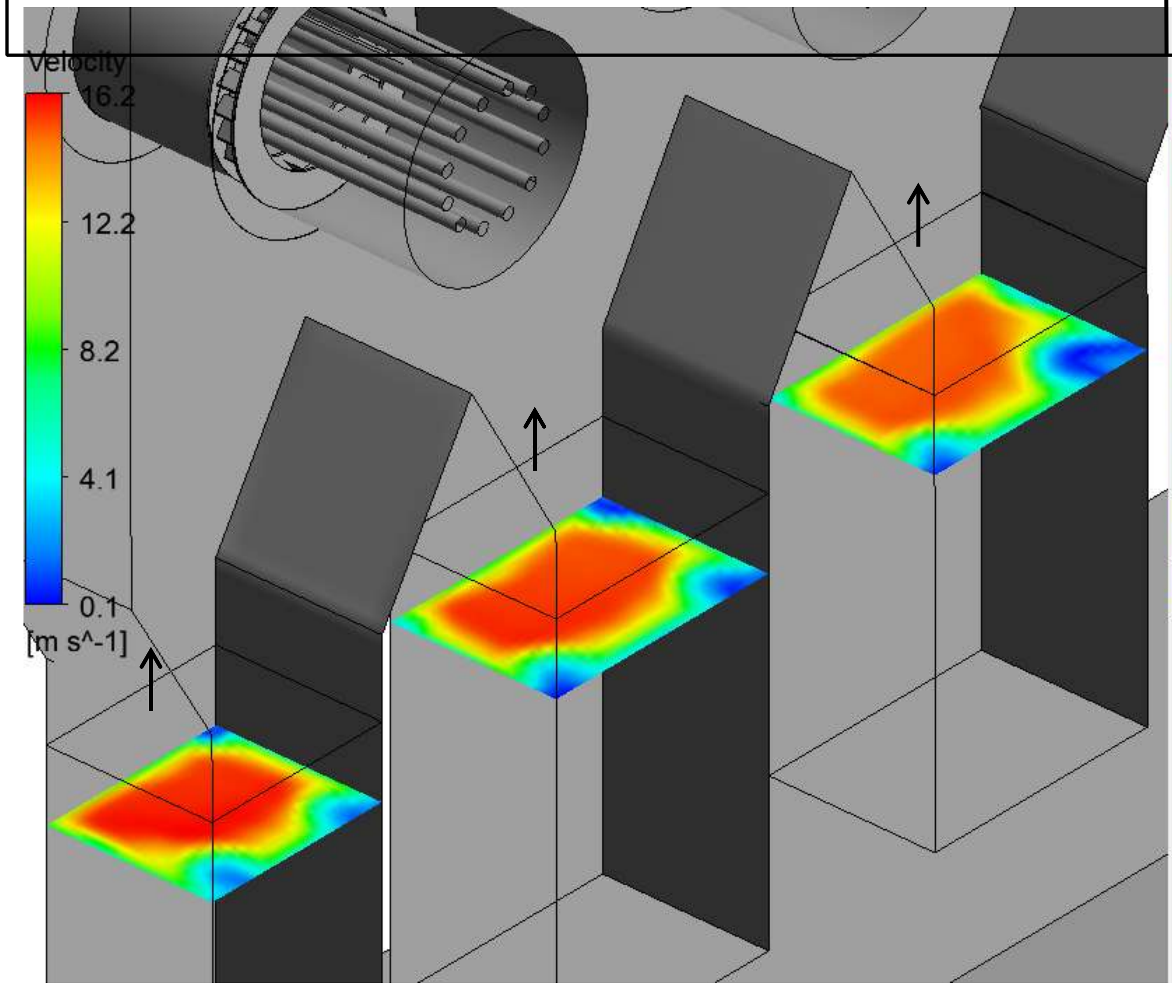
# EXISTING 3D MODEL



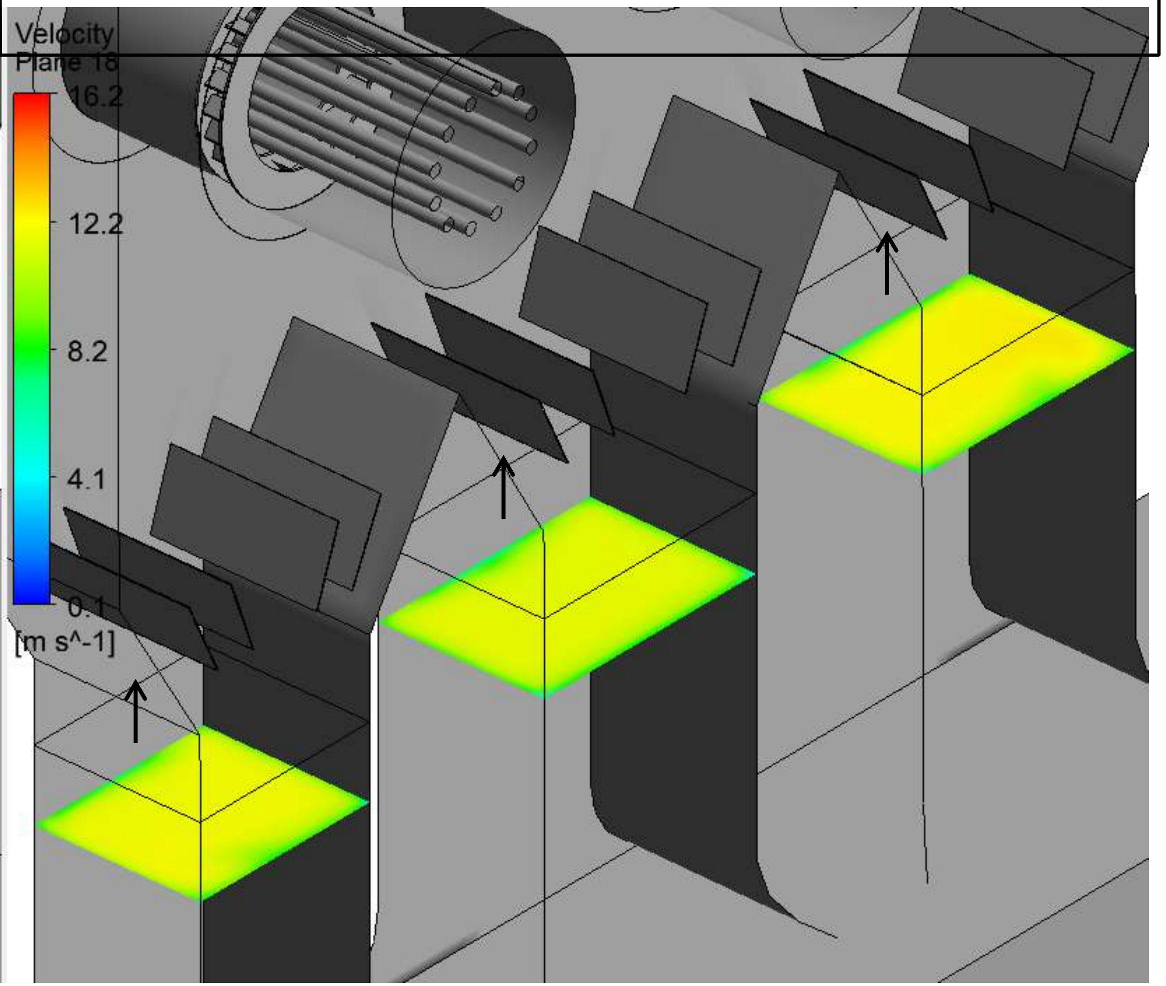
Burner Details

# CFD RESULTS




EXISTING

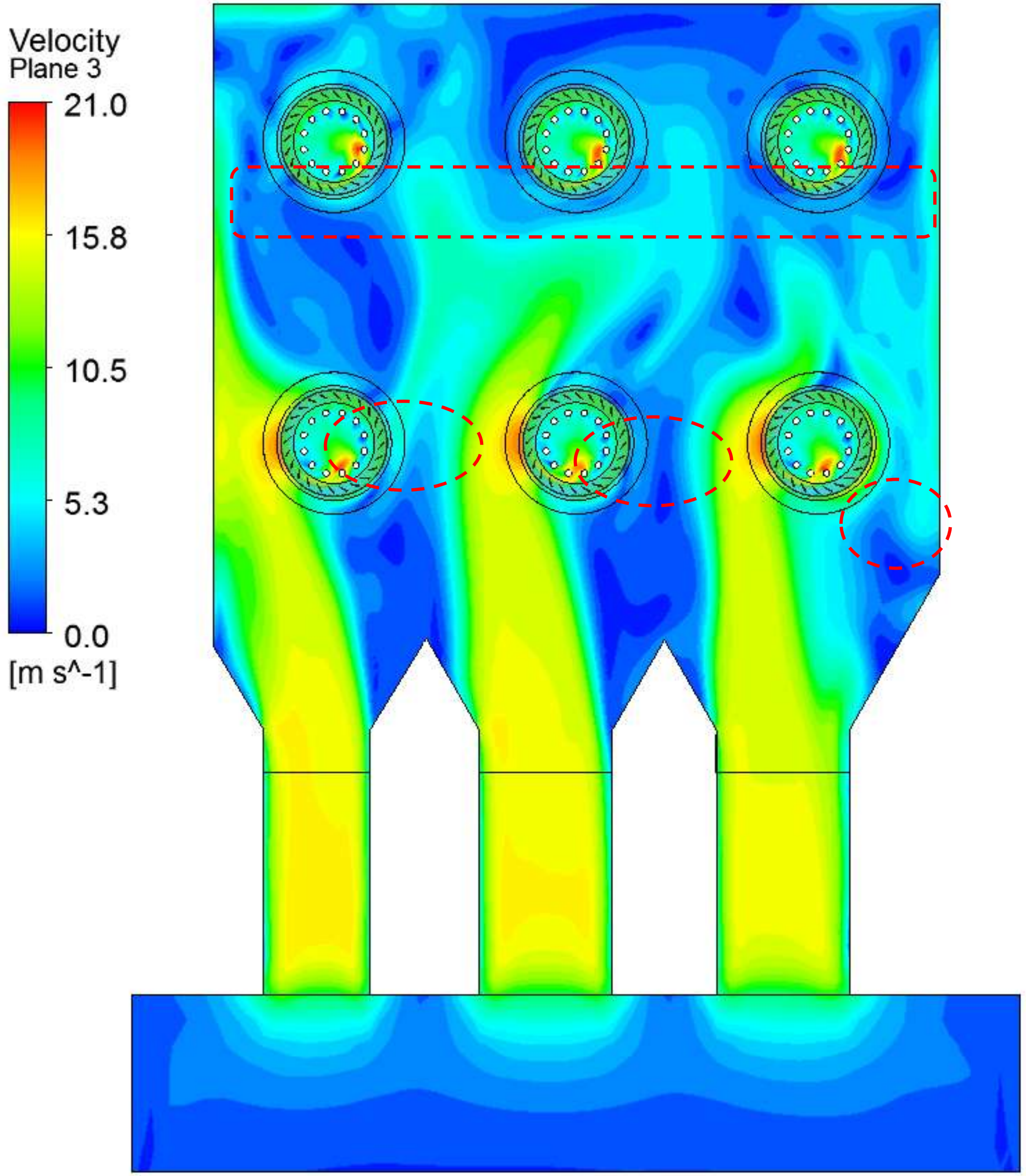
From Velocity plot, it can be observed velocity is concentrated at the centre of the plane because of sharp corners to the duct



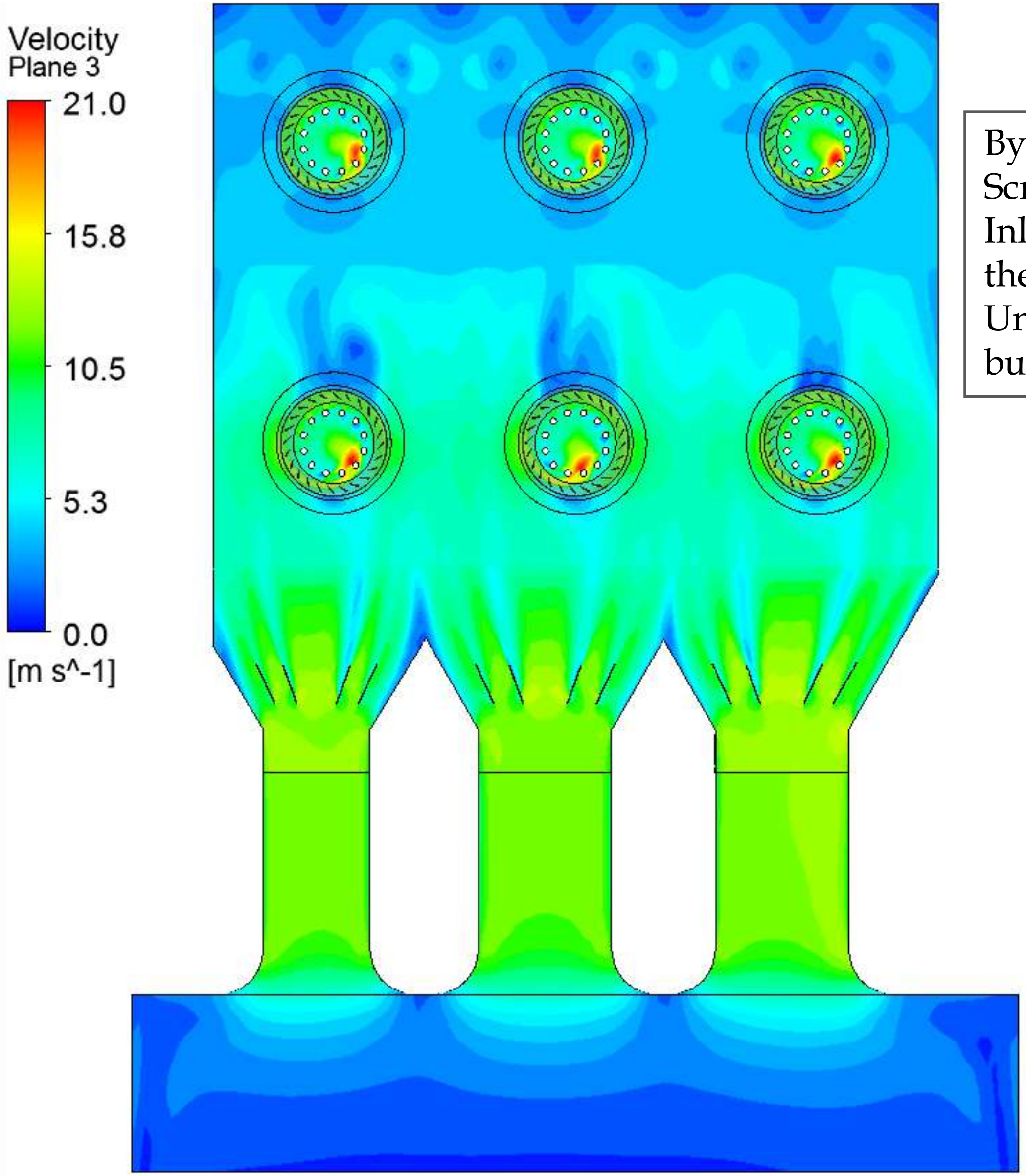

MODIFIED

From Velocity plot, it can be observed flow is more uniform & after design of fillets

# CFD RESULTS



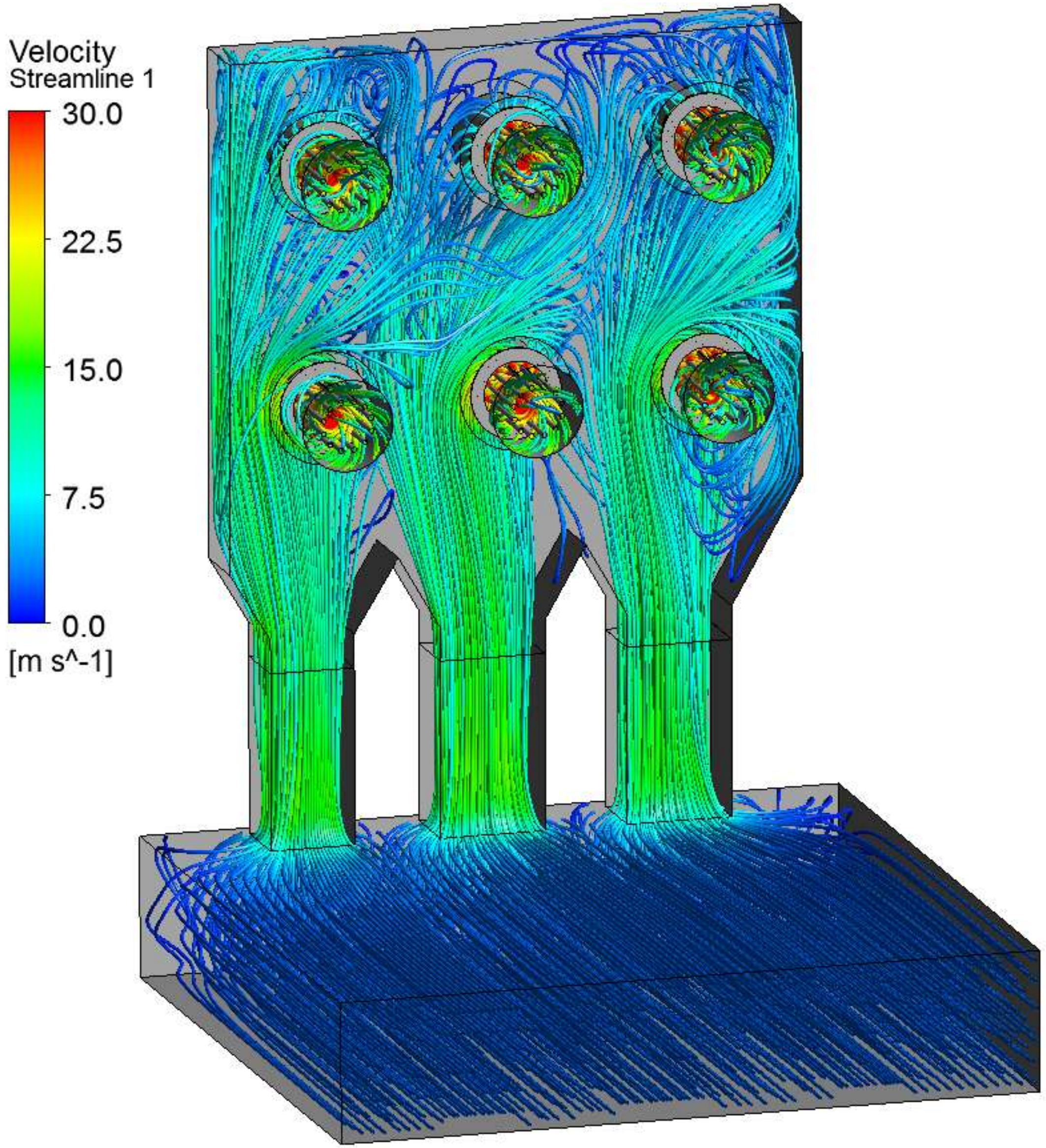

EXISTING



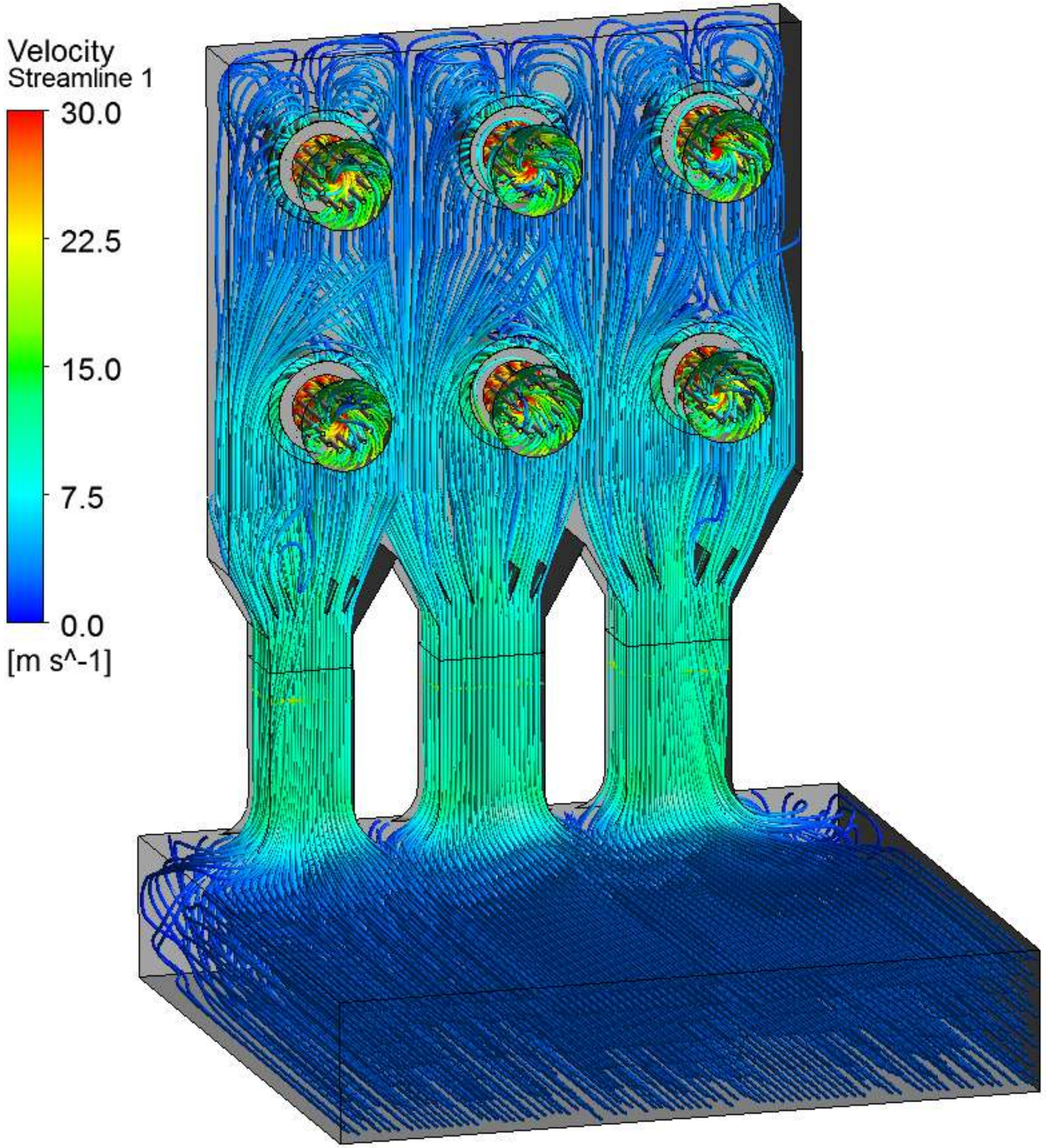

MODIFIED

By Design the GD Screens , Modified Inlet Duct , Baffles, the Flow become Uniform near to burners and Inlet.

# CFD RESULTS



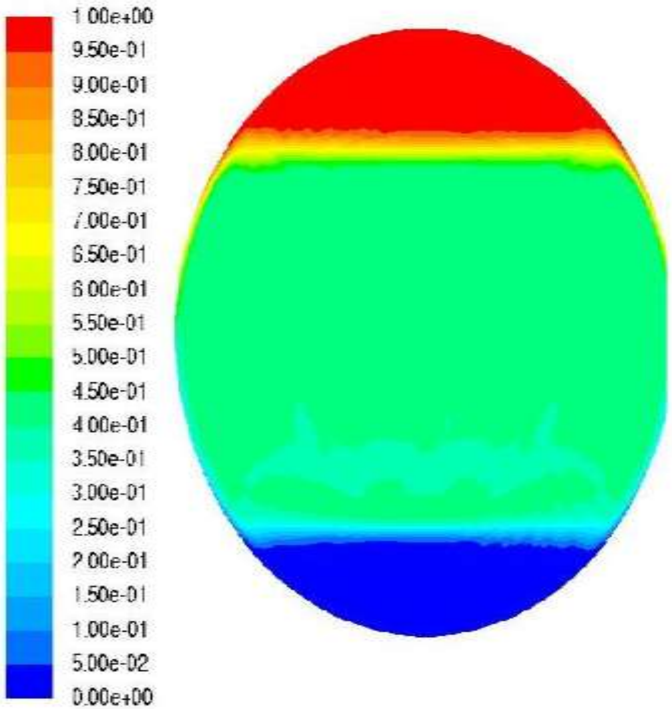

EXISTING



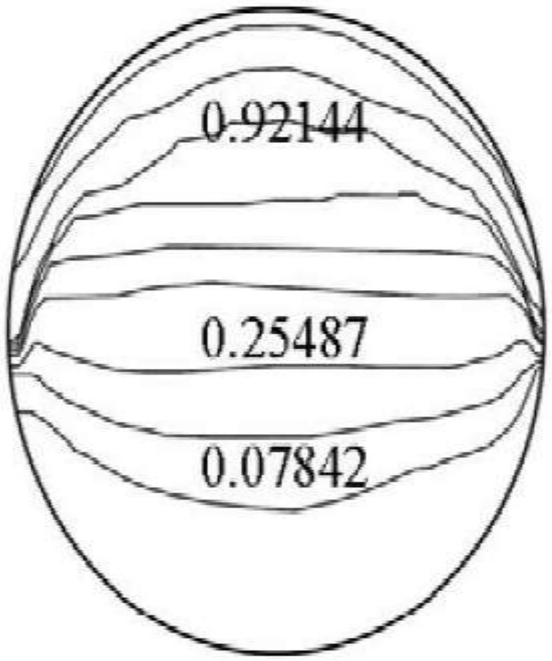

MODIFIED

# CFD for Oil & Gas Separation

Three–Phase Separation ( Oil-Gas-Water)

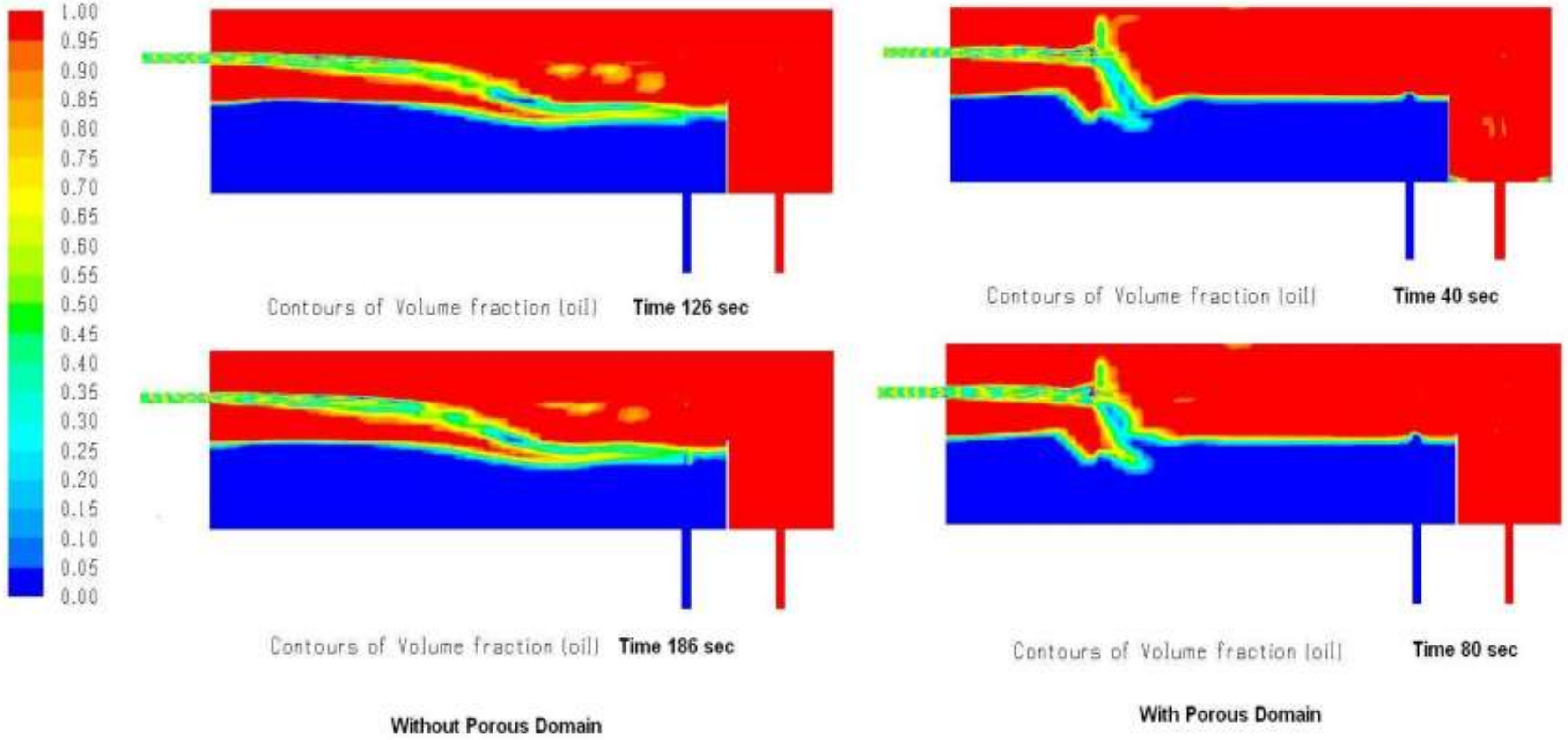



CFD




Experimental

# `CFD Analysis for Oil & Gas Separation

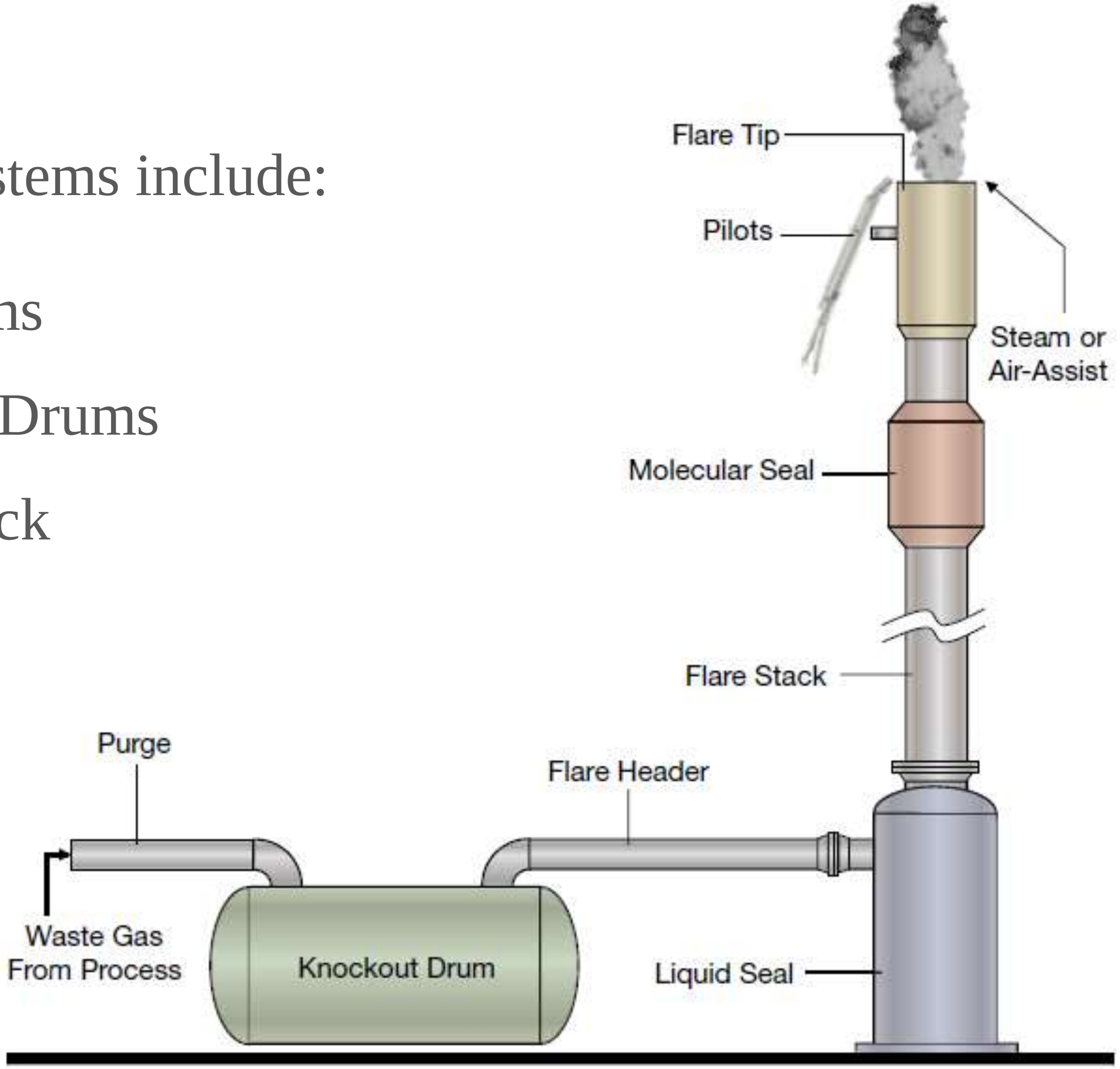



Two Phase Separation with and without porous domains at various Time Steps ( Transient Analysis)

CFD Analysis can also increase the life span of a flare system by reducing the flare.

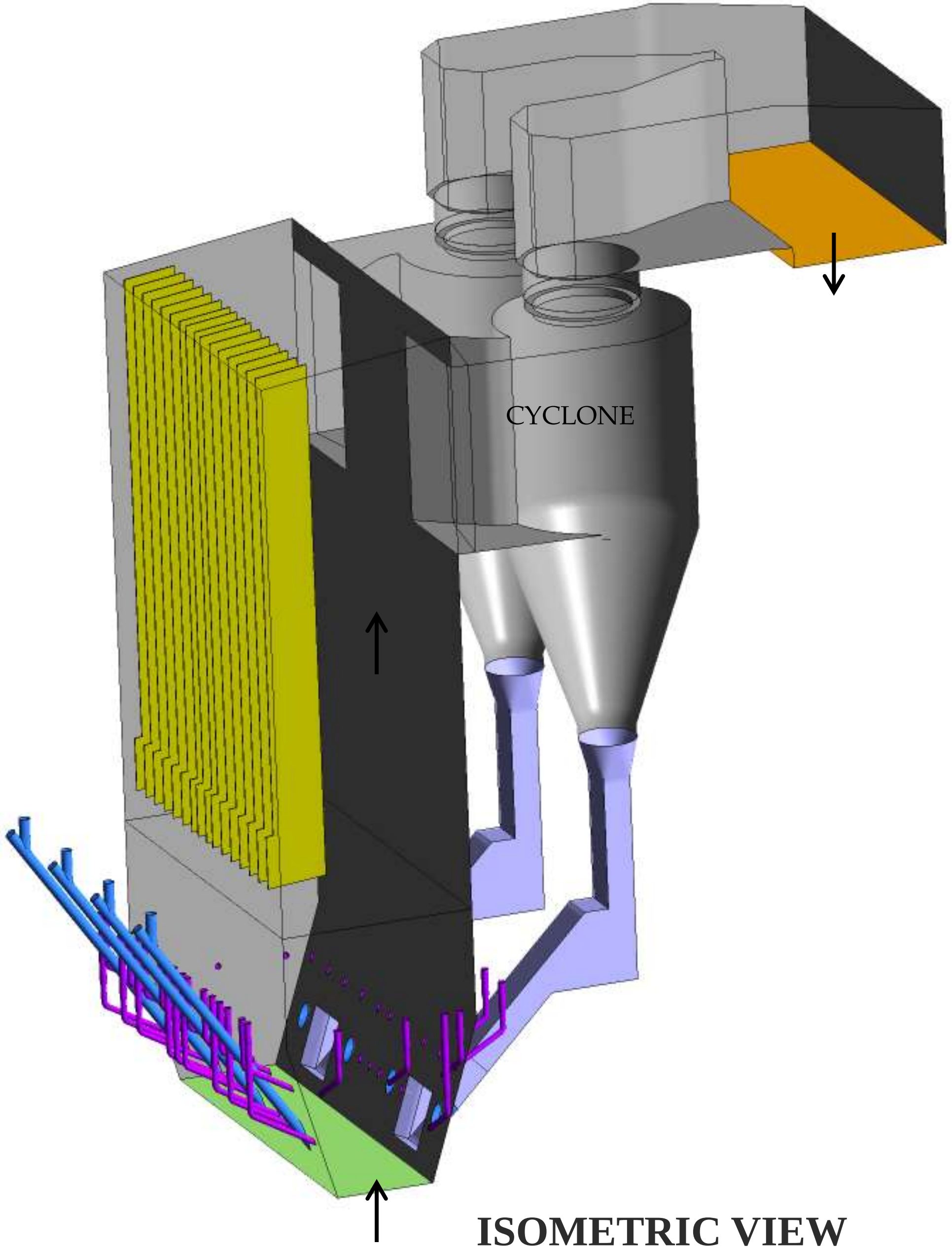
CFD Analysis for Flare Systems include:

- Knockout Drums
- Liquid Seal Drums
- Flare Stack

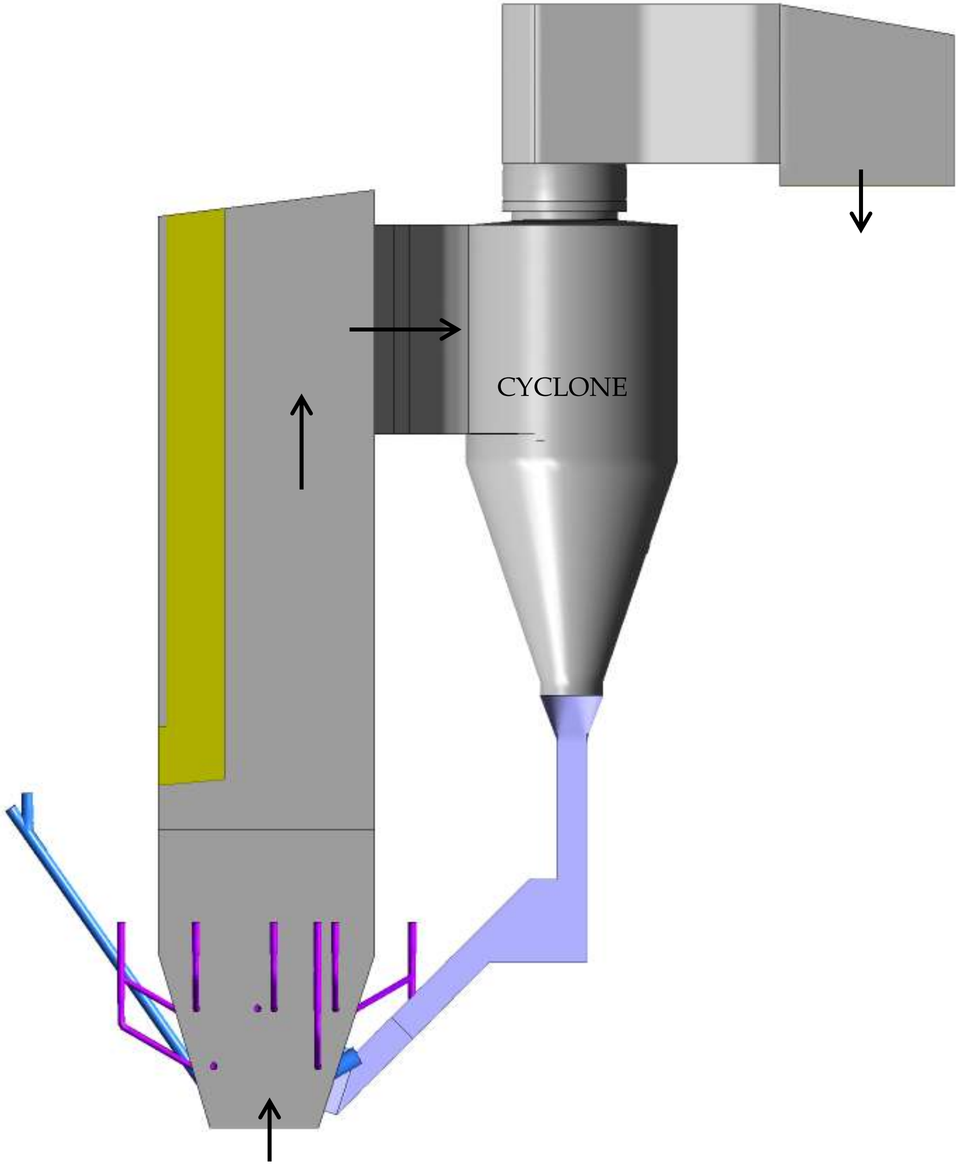


# CFD IN CFBC BOILERS

# CFBC BOILER- CFD MODEL



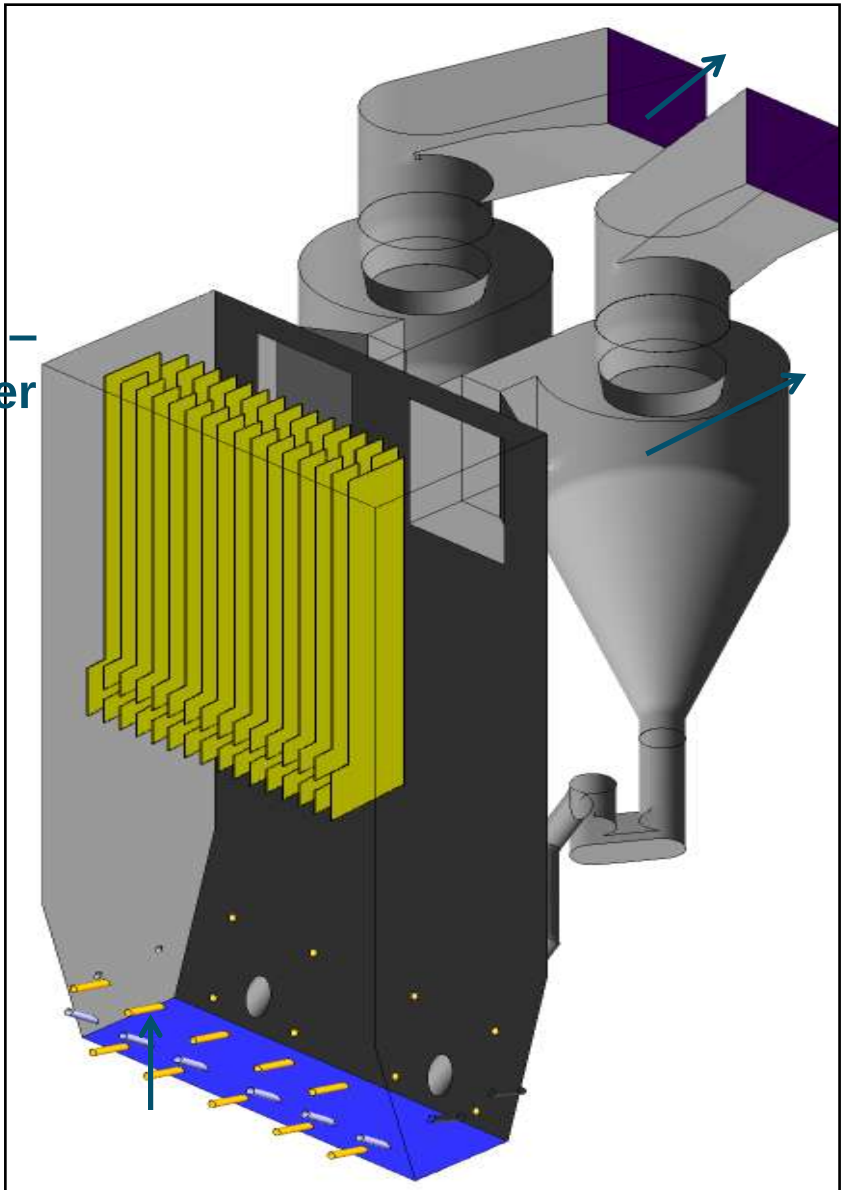

**ISOMETRIC VIEW**



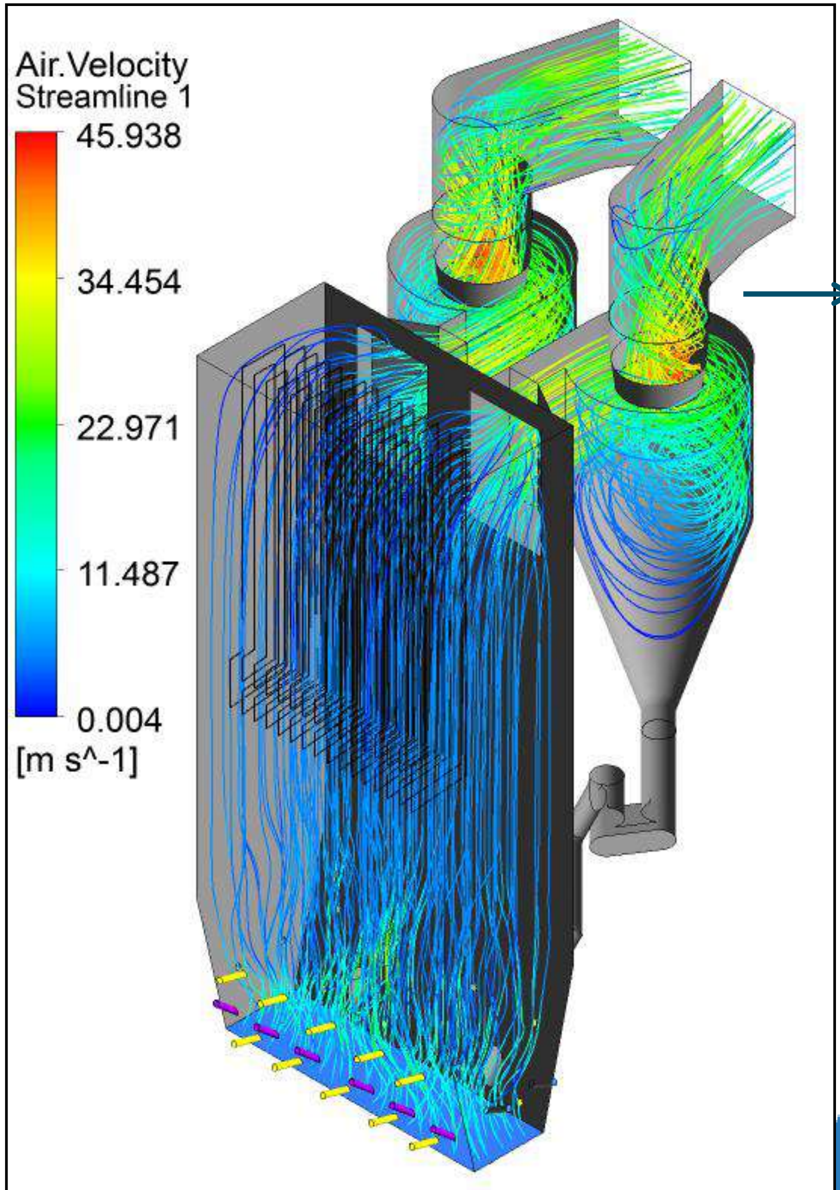

**FRONT VIEW**

# Cfbc Boiler

Cad Model – CFBC Boiler

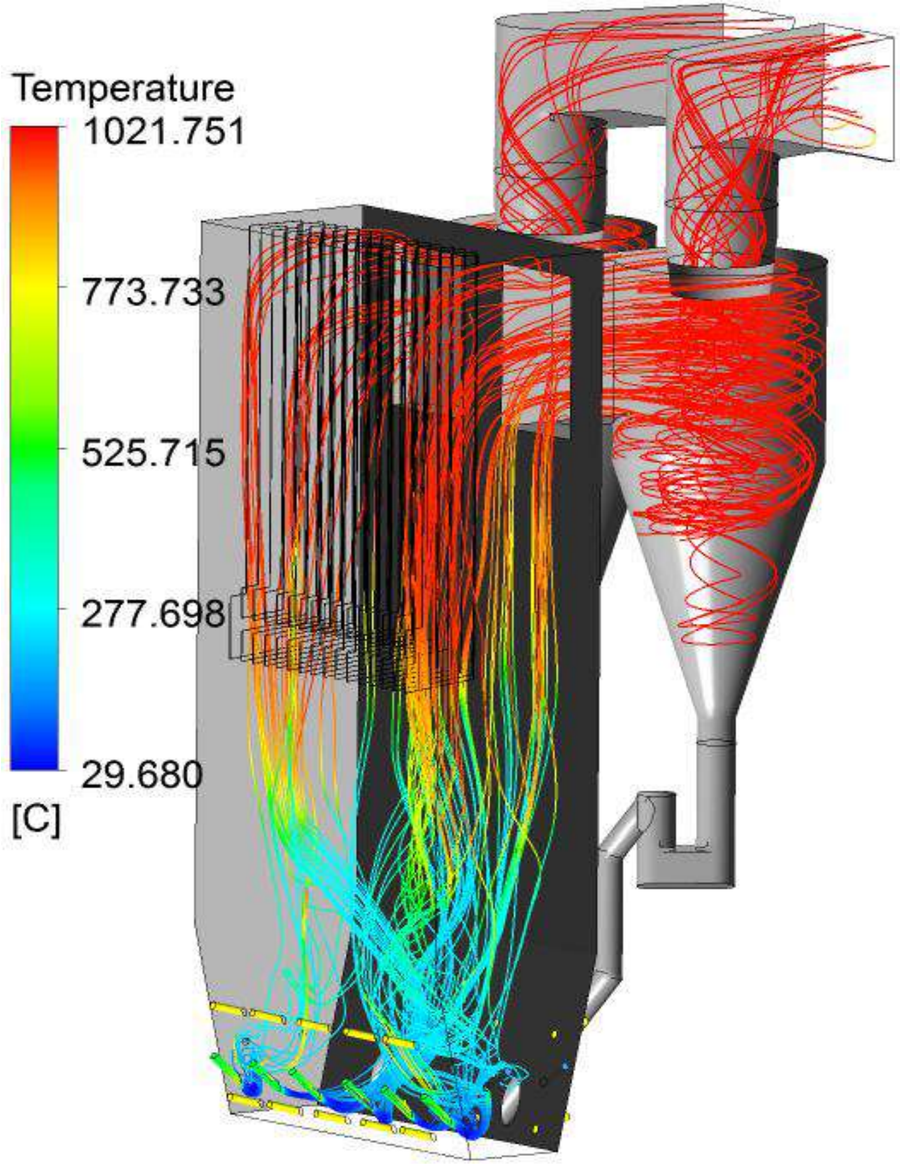


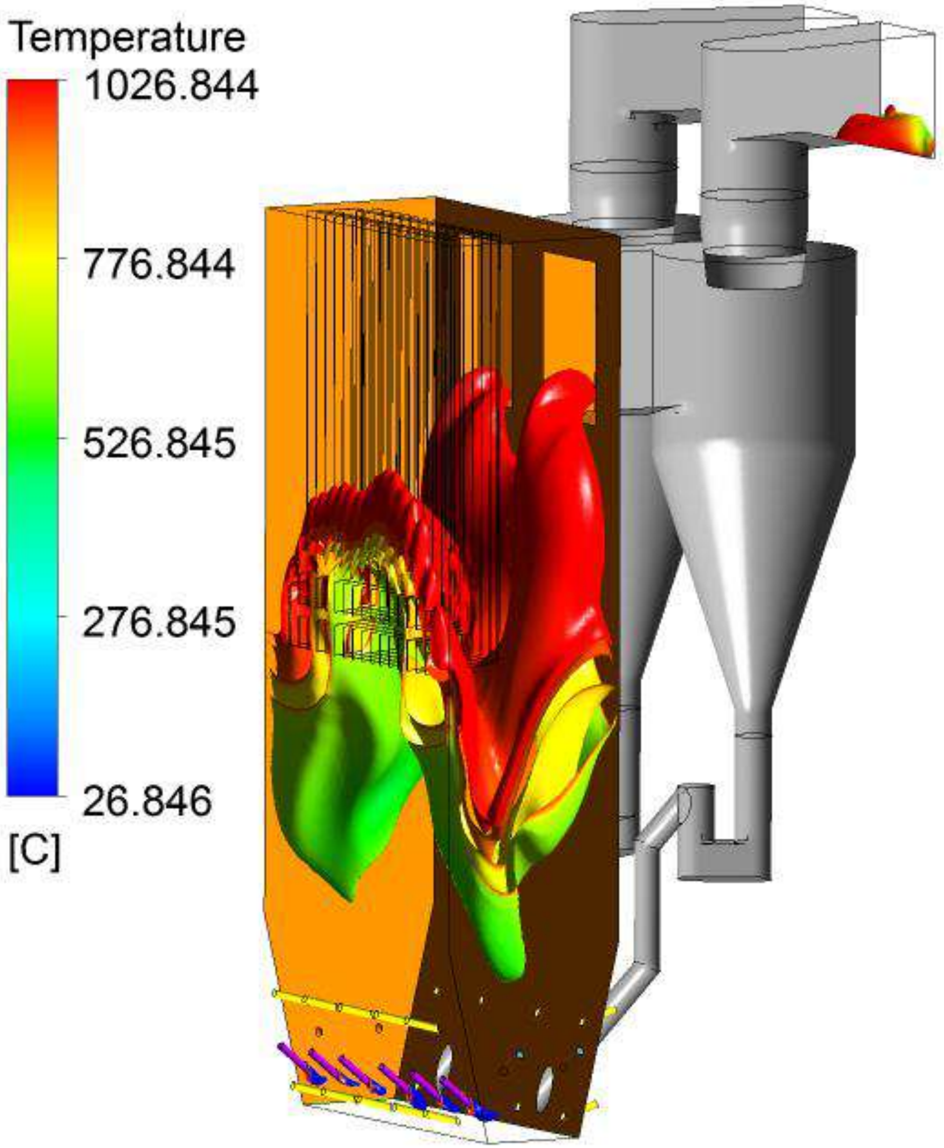
Cyclone



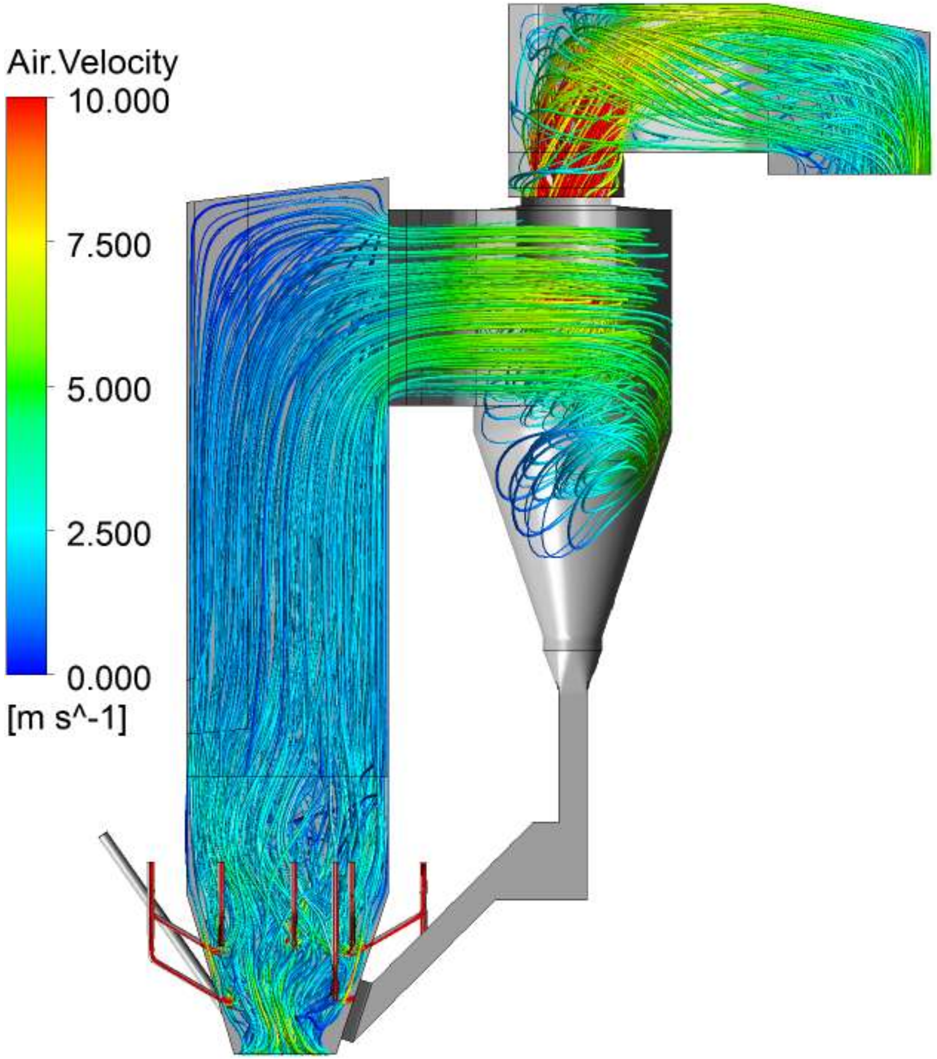

Cold Flow Simulation

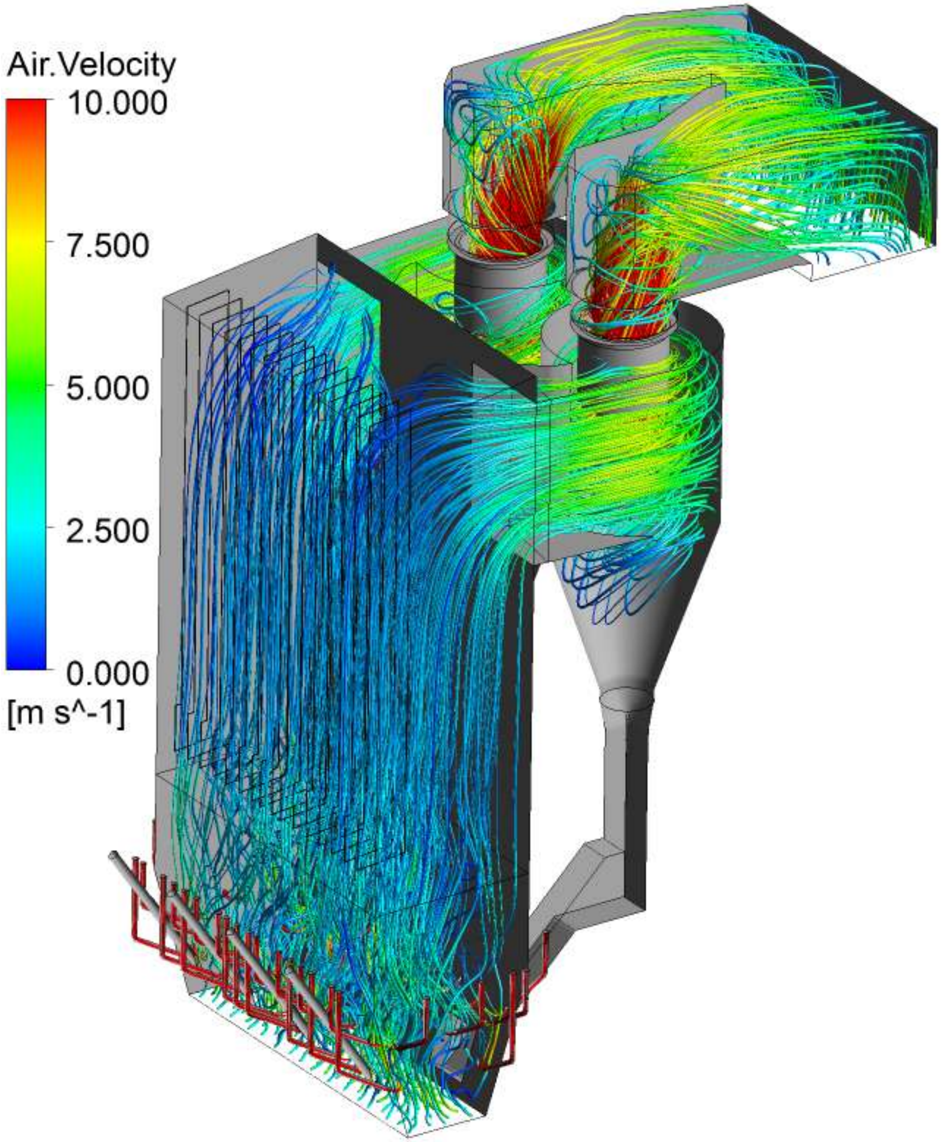
# CFBC Boiler-simulation & Combustion Analysis



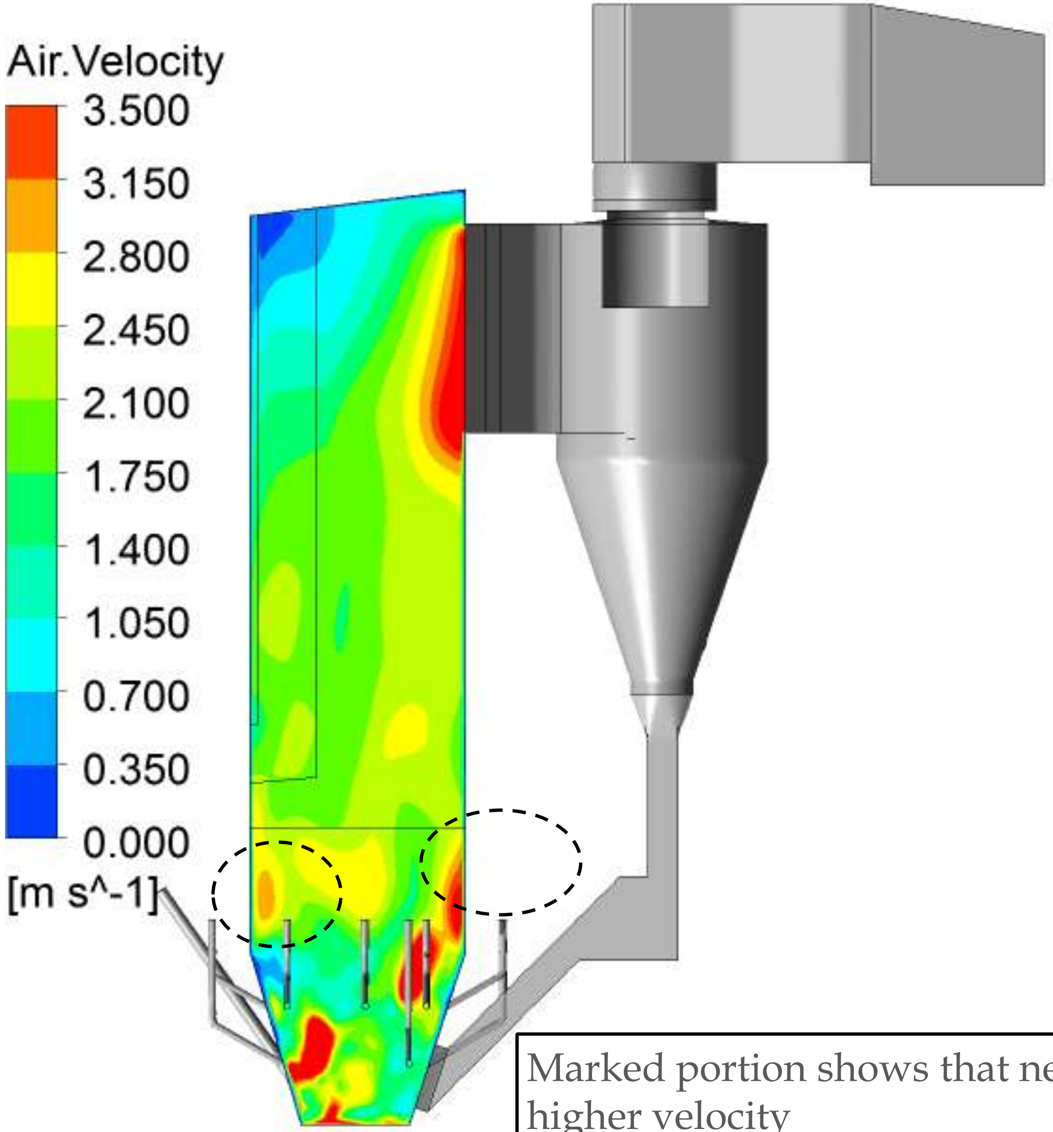
# EXISTING SIMULATION RESULTS



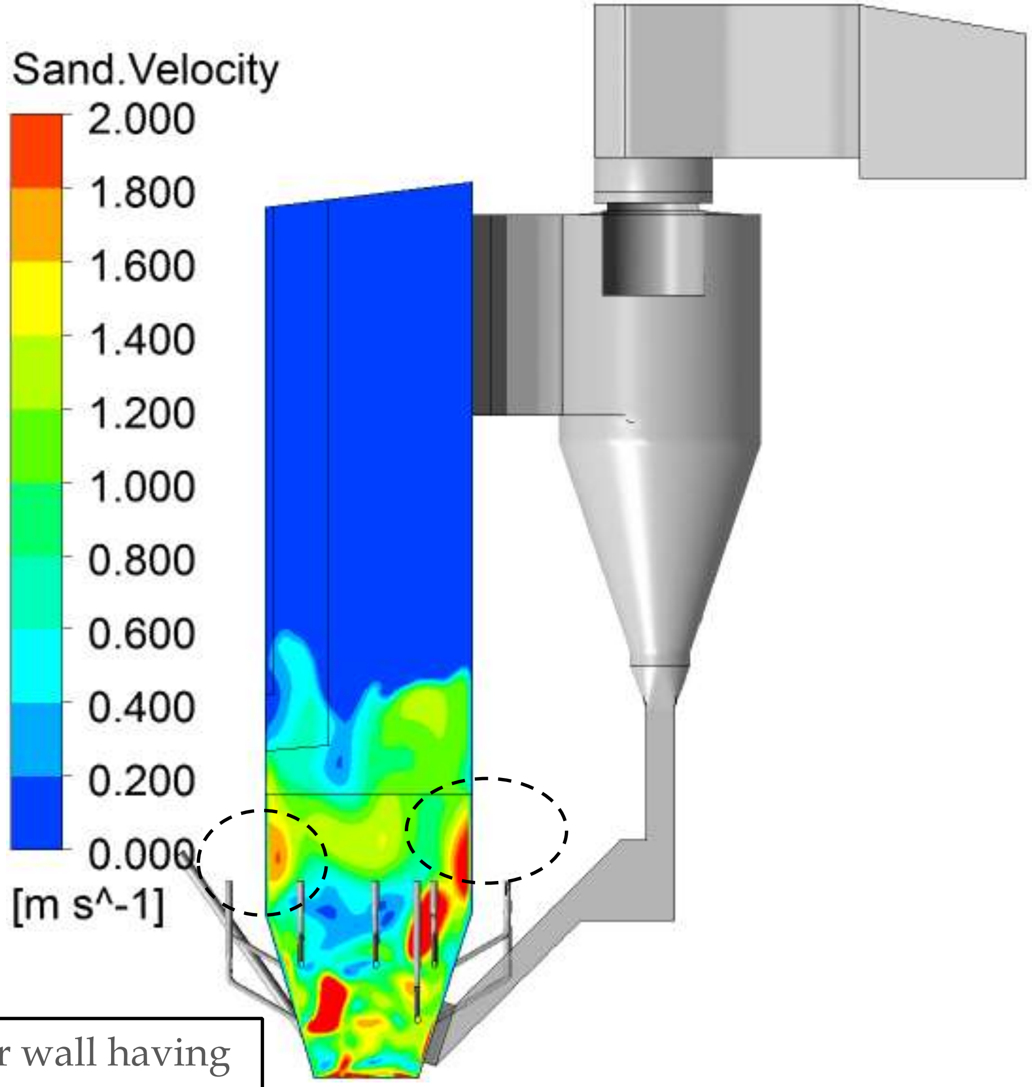




PA AIR VELOCITY FROM NOZZLES

# EXISTING SIMULATION RESULTS



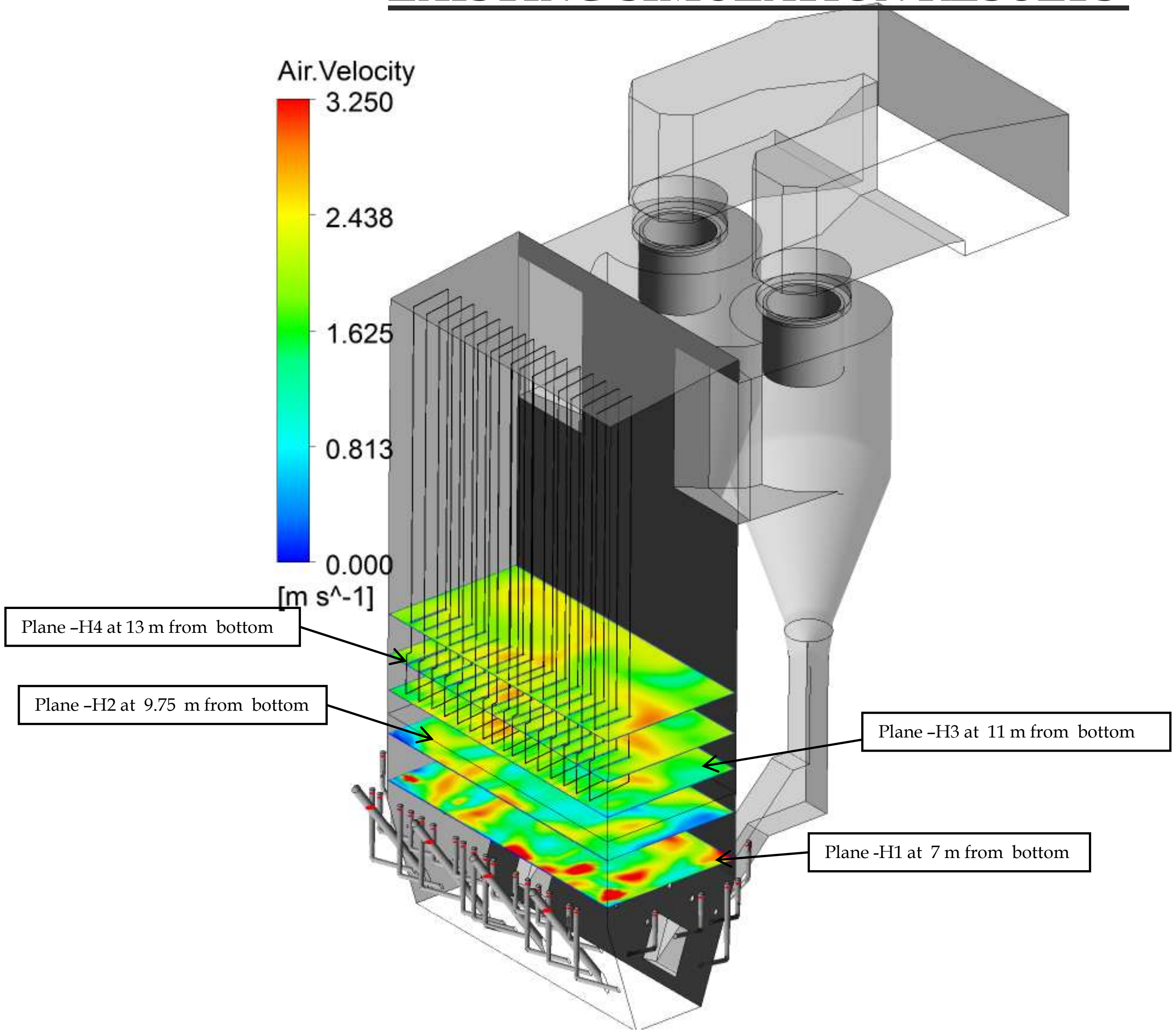

Air velocity on plane-1




Sand velocity on plane-1

Marked portion shows that near rear wall having higher velocity

# EXISTING SIMULATION RESULTS




FLUIDIZATION VELOCITY AT THE PLANES AT DIFFERENT ELEVATION

# OBSERVATION FROM EXISTING BOILER ANALYSIS

- CFD Analysis of Fluidization has been Carried out for the Existing Case.
- It can be observed that near rear wall at the center of furnace at 10 to 11.5 m height ,Gas velocity as well as material velocity is higher which is cause of the erosion
- The un- equal flow through the SA Ports affecting on the fluidization velocity in the furnace & causing the erosion of the front or rear wall.
- The flow from SA ports to be distributed such that good fluidization velocity should be maintained through out the furnace. The required flow from SA ports can be derived from the CFD Analysis.
- Flow separation has been observed at the cyclone Outlet Duct & very less flow near the bottom wall of the duct which is causing the material deposition

# These Customers Trust Us

MECHWELL

## OEM's

BHEL

THERMAX

L&T MHI Boilers Pvt. Ltd.

ISGEC Heavy Engineering Ltd.

DOOSAN

## Thermal Power Stations

NTPC

MAHAGENCO – Maharashtra State Power Generation Co. Ltd.

adani

JINDAL POWER

TATA

HPGCL

GEB

GMR – Creating tomorrow today

LANCO

PSPCL – Punjab State Power Corporation Ltd.

CESC Limited

balco

vedanta

ESSAR POWER

WBSEB

HINDALCO – Aditya Birla Group

# Thank-you !!!

Akshay Shah
CAE – R&D Center for Innovation , Nasik
Contact Details:-8275016434 / 9881080433
Mail id:- akshay@mechwell.org ,
nasik@mechwell.com