CFD Modeling References

Furnace Improvements

www.heatflux.com

Computational Fluid Dynamics

- Computational Fluid Dynamics (CFD) is a branch of fluid mechanics, used to solve and analyze problems of fluid flow through numerical analysis of conservation equations for mass, momentum, energy, etc.
- In a heater, CFD can be used to understand operational issues such as flame impingement, high tube metal temperatures, flue gas recirculation, etc.
- CFD simulations are being used as a part of design process to evaluate the performance of a complex design prior to actual construction

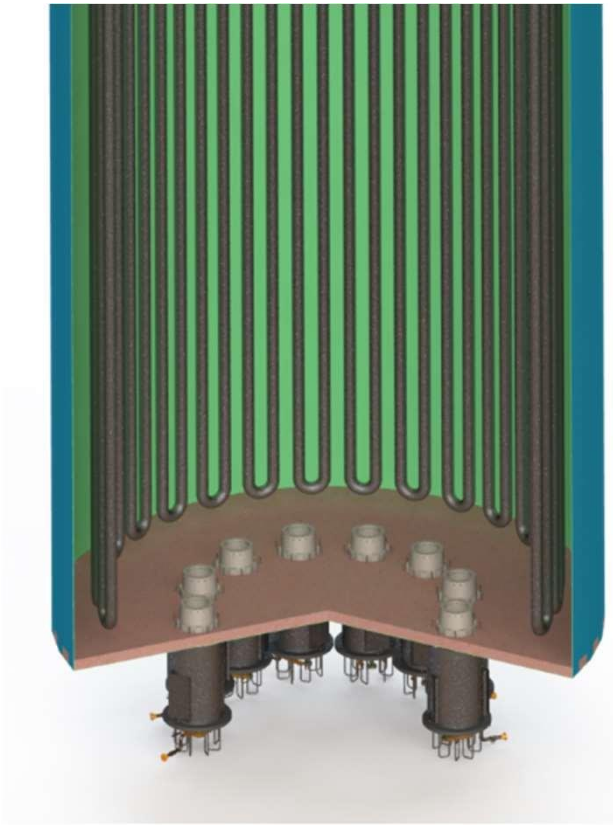
Computational Fluid Dynamics

- At Furnace Improvements, CFD simulations are being utilized to:
  - Study design and operating conditions of heaters,
  - Identify problems,
  - Evaluate various possible design modifications and
  - Provide proposed solutions

CFD Applications

- FIS has developed best practices for CFD modeling of fired heaters
  - Combustion modeling of burners and radiant section
  - Achieving uniform combustion air flow distribution across all the burners
  - Reducing pressure losses for ID fan suction and discharge ducts
  - Achieving uniform air/flue gas flow at inlet of APH
  - Design of Ammonia Injection Grid and mixing device to achieve uniform mixing of flue gas and ammonia-air mixture
- Below are case study based on executed projects for these application

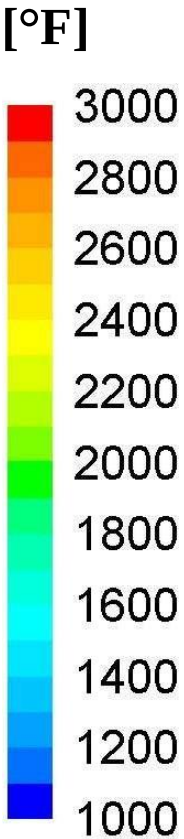
Project #: FIS-428
Scope of Work: Combustion Modeling of heater

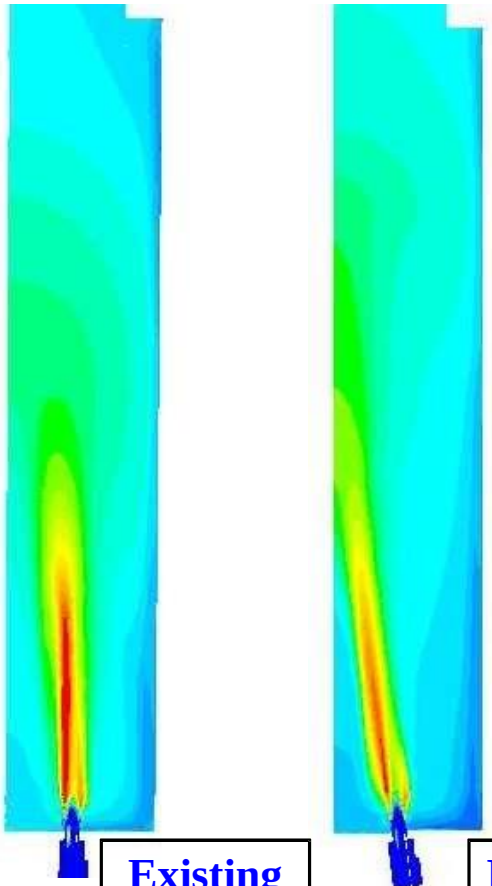


Introduction

- Existing burner configuration had issues of flames leaning towards the radiant tubes and impingement of hot flue gases on radiant tubes
- CFD analysis was done for the existing and proposed configurations
  - It was proposed to install burners with higher air side pressure drop and inclining towards the centre of the heater
  - Proposed simulation results showed reduction in tube metal temperatures and improved flue gas flow pattern

Flue Gas Temperature Contours



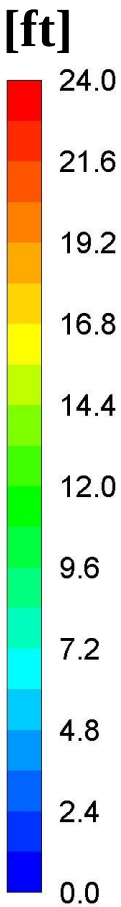


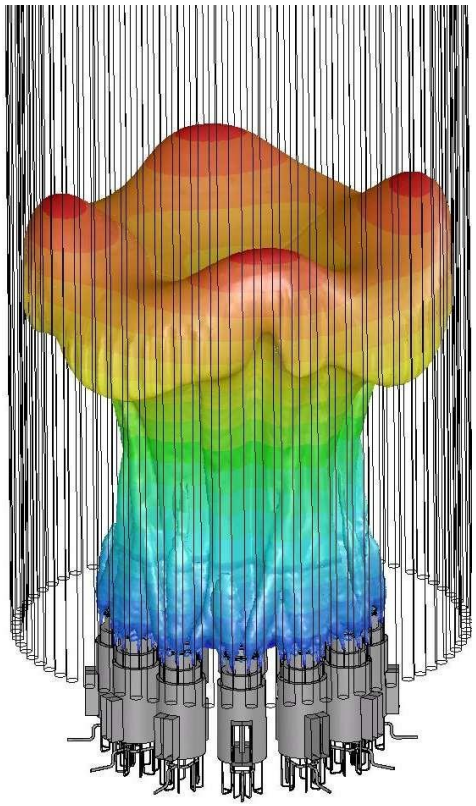
Existing

Proposed Inclined Firing

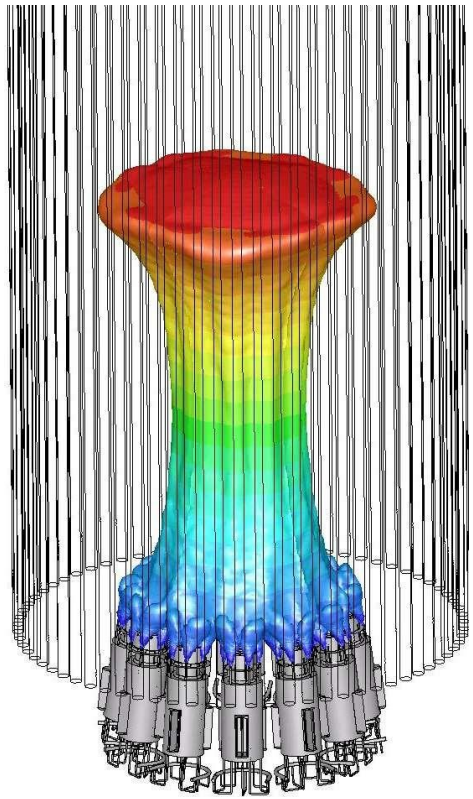
Smaller size of burners were used which increased air side pressure drop. Burners were also inclined towards the centre of heater. The proposed configuration improved flue gas flow pattern and reduced radiant tube metal temperatures significantly.

Comparison of Flame Profiles





Existing

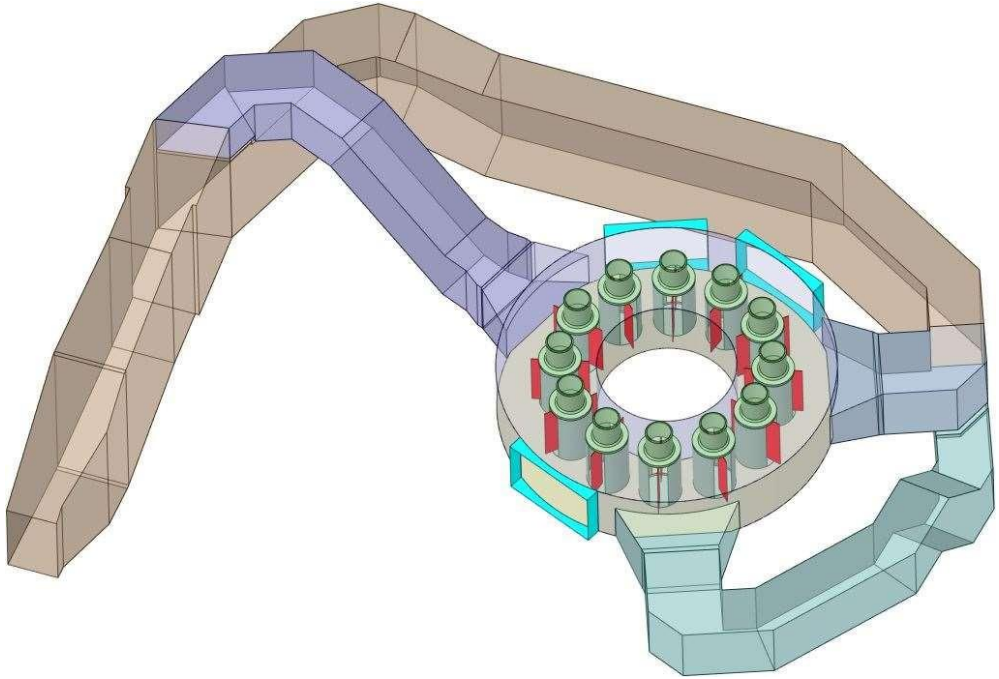


Proposed Inclined Firing

- Inclined firing improves the flame characteristics
- Flames are more concentrated in the center of the heater
- No flame impingement on radiant tubes

Summary

- The proposed solution of using smaller size burners and inclined configuration was implemented
- Heater is running fine post implementation
- Client is happy with heater's improved performance



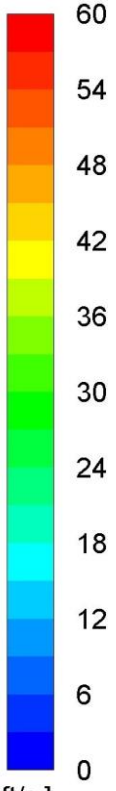
Project #: FIS-428

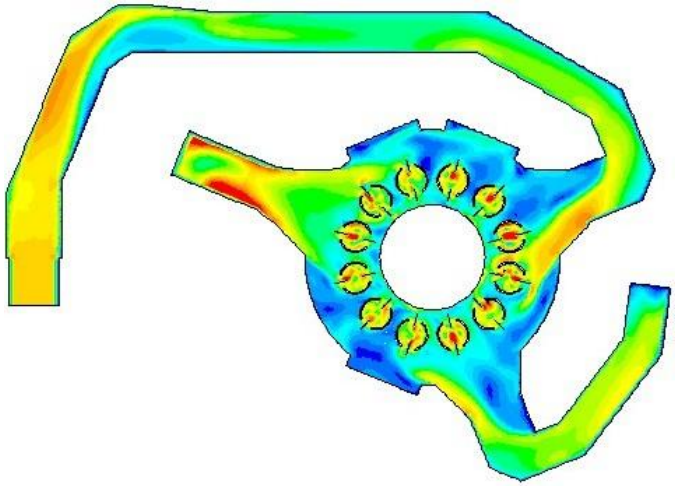
Scope of Work: Achieving Uniform Air Flow Distribution across all the burners

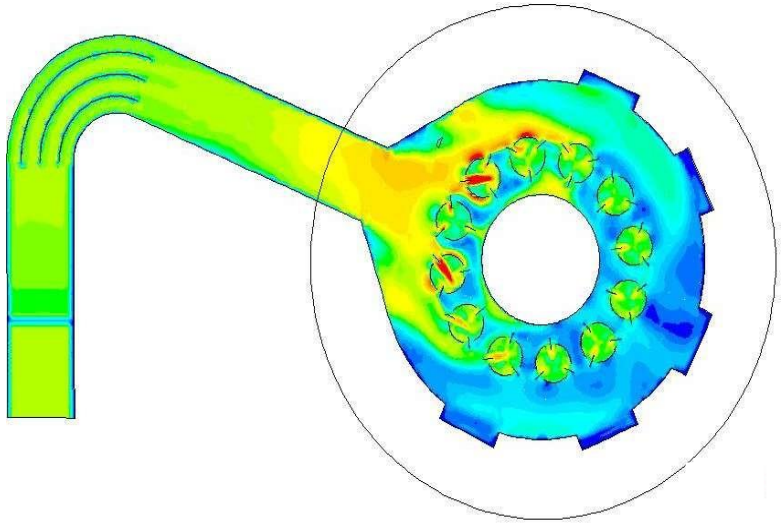
Introduction

- Existing combustion air distribution duct had three branches connecting to burner plenum
  - There was considerable mal distribution in air flow across all the burners
- CFD analysis was done for the existing configuration of combustion air duct and burner plenum
  - It was found that few burners were receiving very high and very low air flow
- Burner plenum and duct configuration was modified to achieve equal air flow distribution across all the burners

Comparison of Velocity Contours





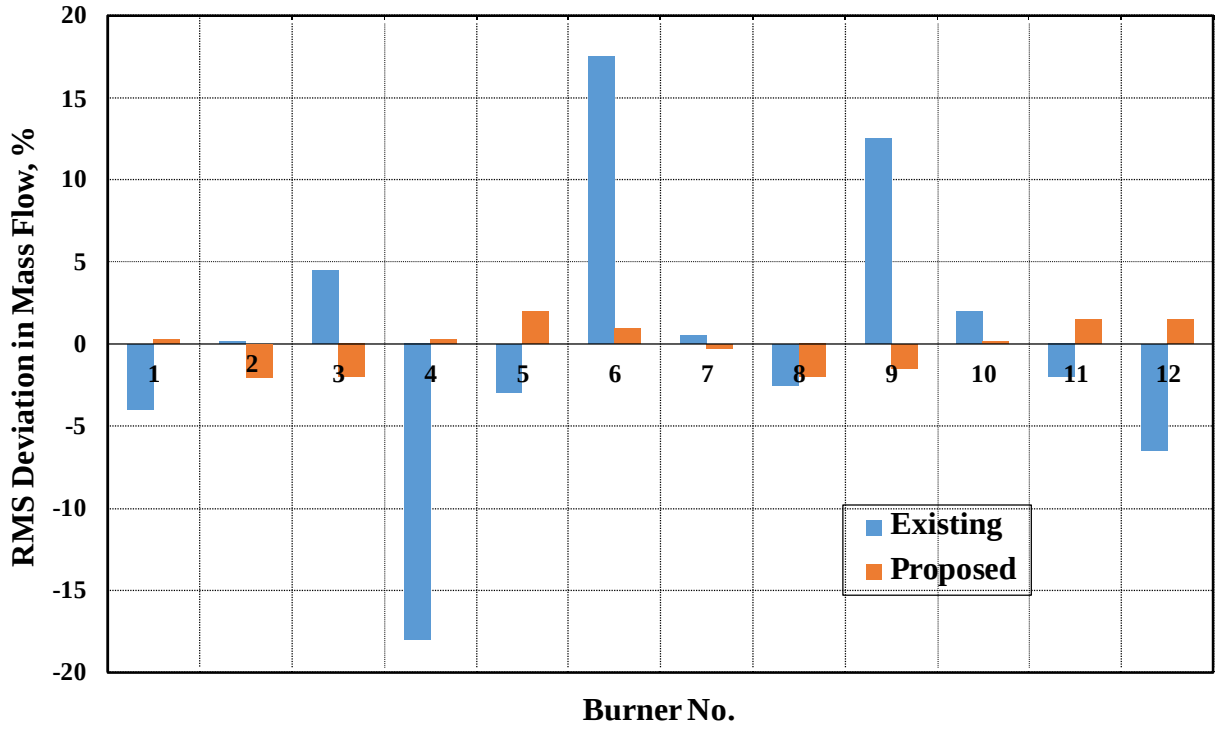


Existing

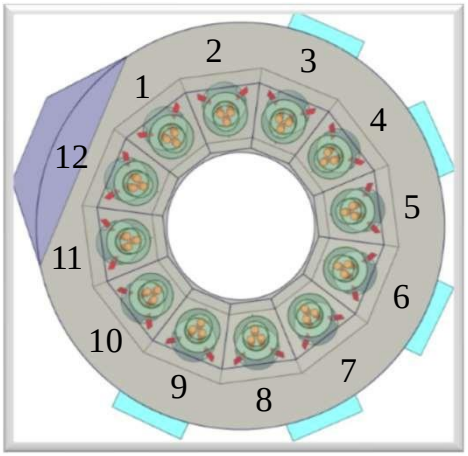
Proposed

The proposed burner plenum and duct configuration, had improved air flow distribution across all the burners

Comparison of Air Flow Distribution

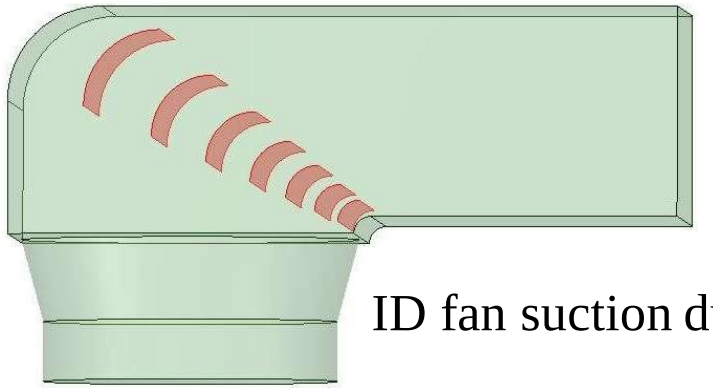


Mass flow deviation for the proposed case is within ± 3%

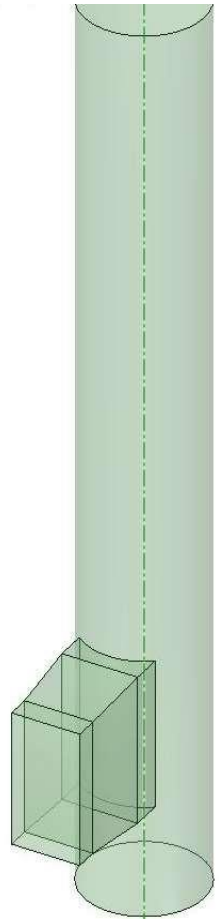


Summary

- The proposed duct layout with modified burner plenum was implemented
- Improved air flow distribution across all burners gave better combustion and flame characteristics
- Client is happy with heater's improved performance



ID fan suction duct



ID fan discharge duct

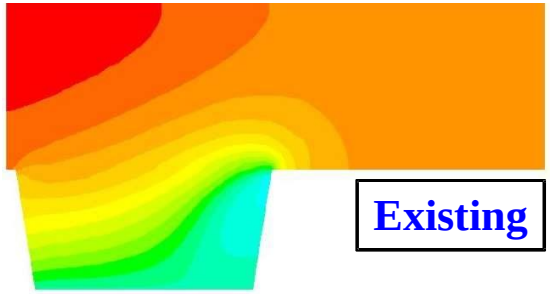
Project #: FIS-428

Scope of Work: Reducing Pressure Losses for ID Fan Suction and Discharge ducts

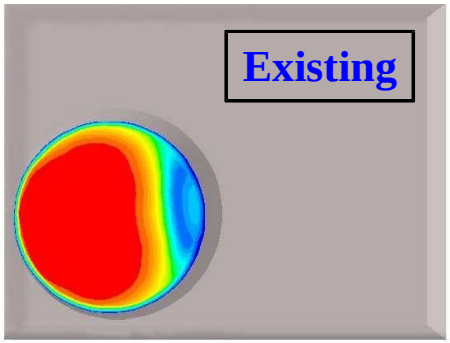
Introduction

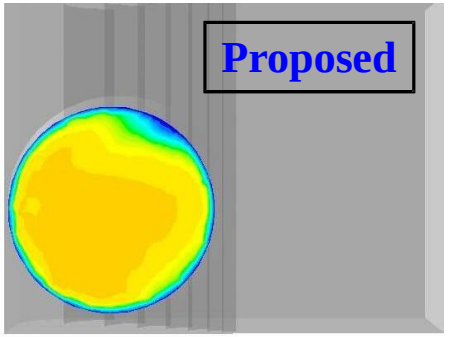
- Existing ID fan location was such that it had a sharp turning duct on the suction side and short discharge duct going into the stack
- CFD analysis was done for the existing configuration of ID fan suction and discharge ducts
  - It was found that due to the sharp bend, there was uneven flow distribution at the fan inlet and large pressure drop in the duct
  - On the discharge side, all the flue gas was flowing along the stack wall, leading to large pressure drop and non-uniform flow in the stack

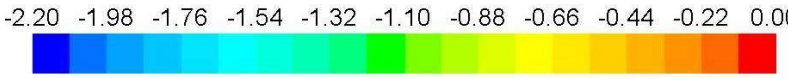
ID Fan Suction Duct Results



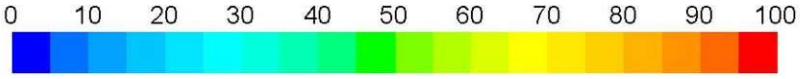
Minor modifications to the duct were made to have smooth bend, and a gradual transition to fan inlet. Small turning vanes were used in the duct to achieve uniform flow profile at fan inlet. Around 0.5 inches w.c. pressure drop was reduced on the suction side.





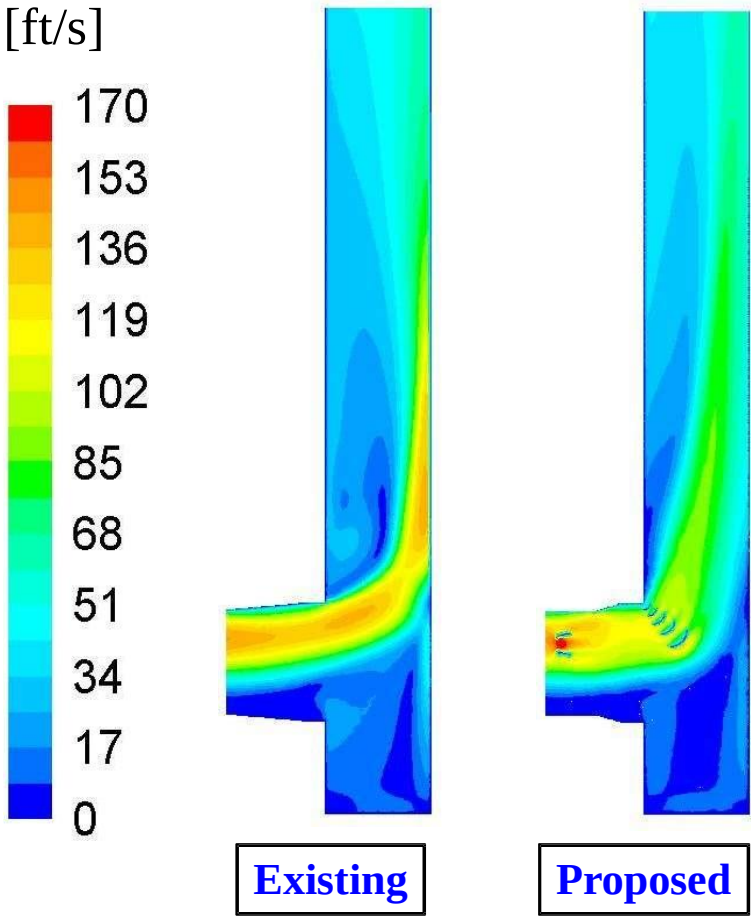


Pressure (inches w.c.)



Velocity (ft/s)

ID Fan Discharge Duct Velocity Contours



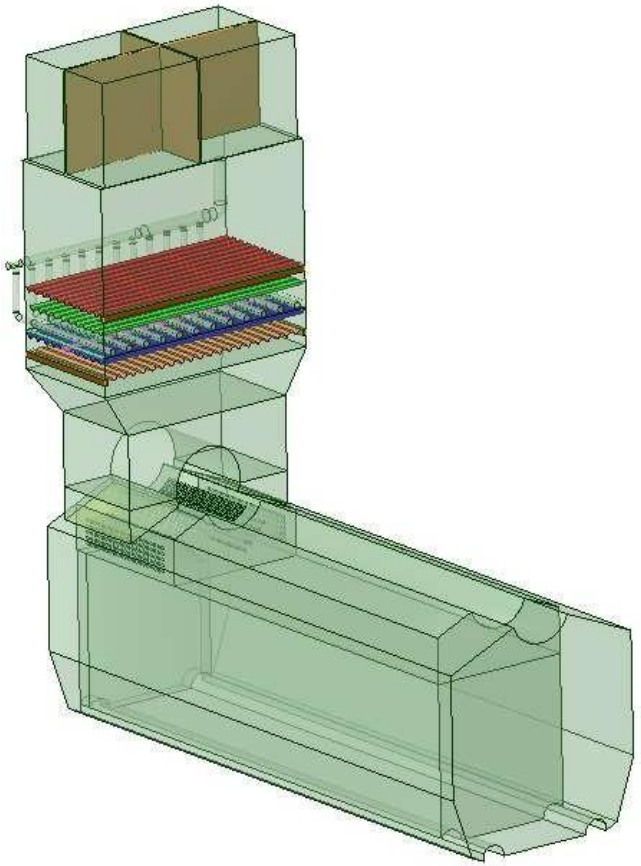
Duct from ID fan outlet to stack was modified to have larger opening. Few entry baffles and turning vanes were installed to direct the flow into the stack, and achieve uniform flow profile in the stack cross section. A pressure drop of 0.25 inches w.c. was reduced with the proposed modifications.

Summary

- Proposed duct modification for both ID fan suction and discharge ducts were implemented
- Post implementation client observed better performance of the fan
- Client reported reduction in fan amperage reading
  - This is an indication of reduced pressure losses in the proposed duct configuration

Project #: FIS-435

Scope of Work: Design of Ammonia Injection Grid (AIG) and mixing baffles



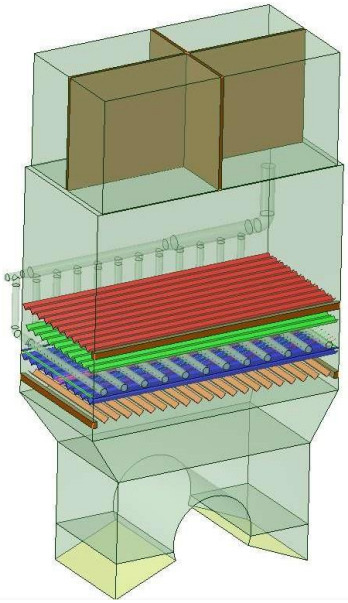
Introduction

- Flue gases out of a H-type boiler were passed through SCR unit for NOx reduction. Ammonia-air mixture is injected into the flue gas upstream of SCR unit. It is desired to achieve uniform mixing of flue gas and ammonia-air streams at SCR inlet to ensure proper reduction of NOx
- Challenges: Residence time in the duct around 0.5 s, duct height of 9’ from AIG to SCR inlet

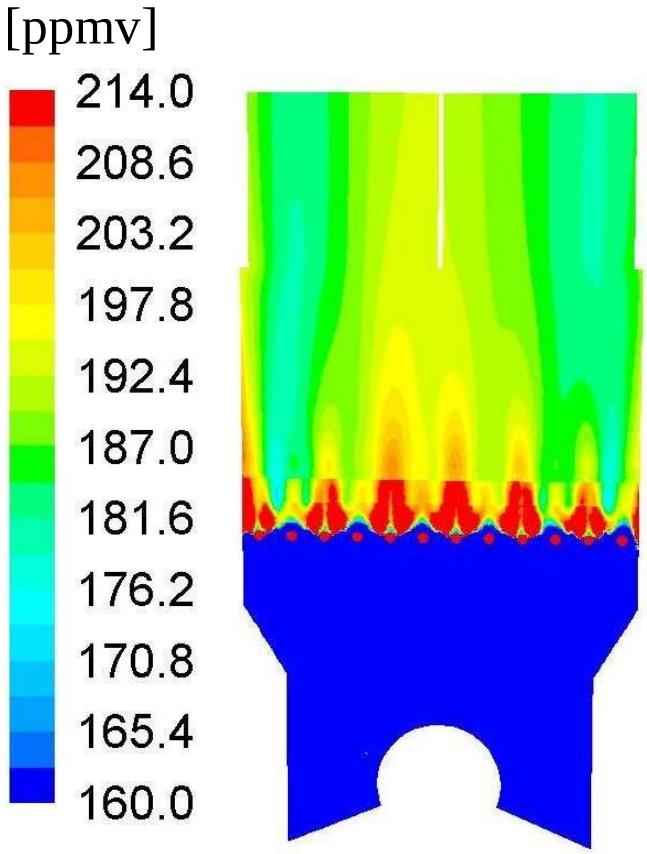
Modeling Details

- To ensure adequate performance of SCR catalyst following design criteria needs to be satisfied
  - RMS deviation of flue gas velocity at AIG and SCR inlet within ±15%
  - RMS deviation of NH_3/NOx molar ratio within ±5%
- Initially simulations were carried out to achieve uniform flue gas velocity, later multiple design of AIG was evaluated to achieve uniform mixing
- Due to the constraint of duct height and low residence time, two rows of angled baffles above AIG were used to facilitate mixing of gases

AIG & Baffles Design



Contours of NH_3 distribution in the duct

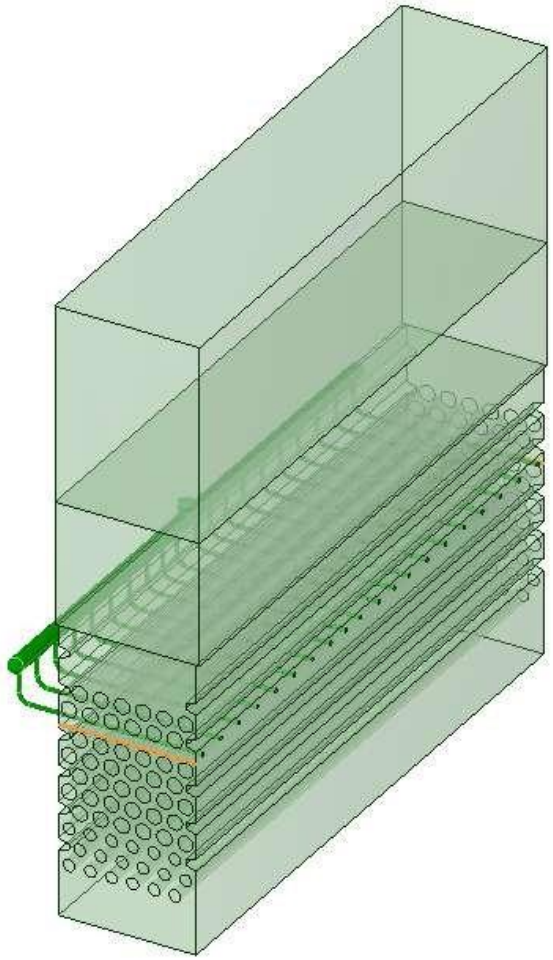


Contours of NH_3 in the duct show mixing of gases. RMS deviation of NH_3 distribution at SCR inlet was within ± 3%. RMS deviation of velocity distribution at SCR inlet was within ± 5%.

Summary

- To achieve uniform velocity and mixing in a short duct height was challenging
- Multiple rows of angle baffles and design of AIG helped in achieving the design criteria required by SCR catalyst
- The proposed design of AIG and mixing baffles was implemented
- Client reported reduction in NOx in flue gas as per the design

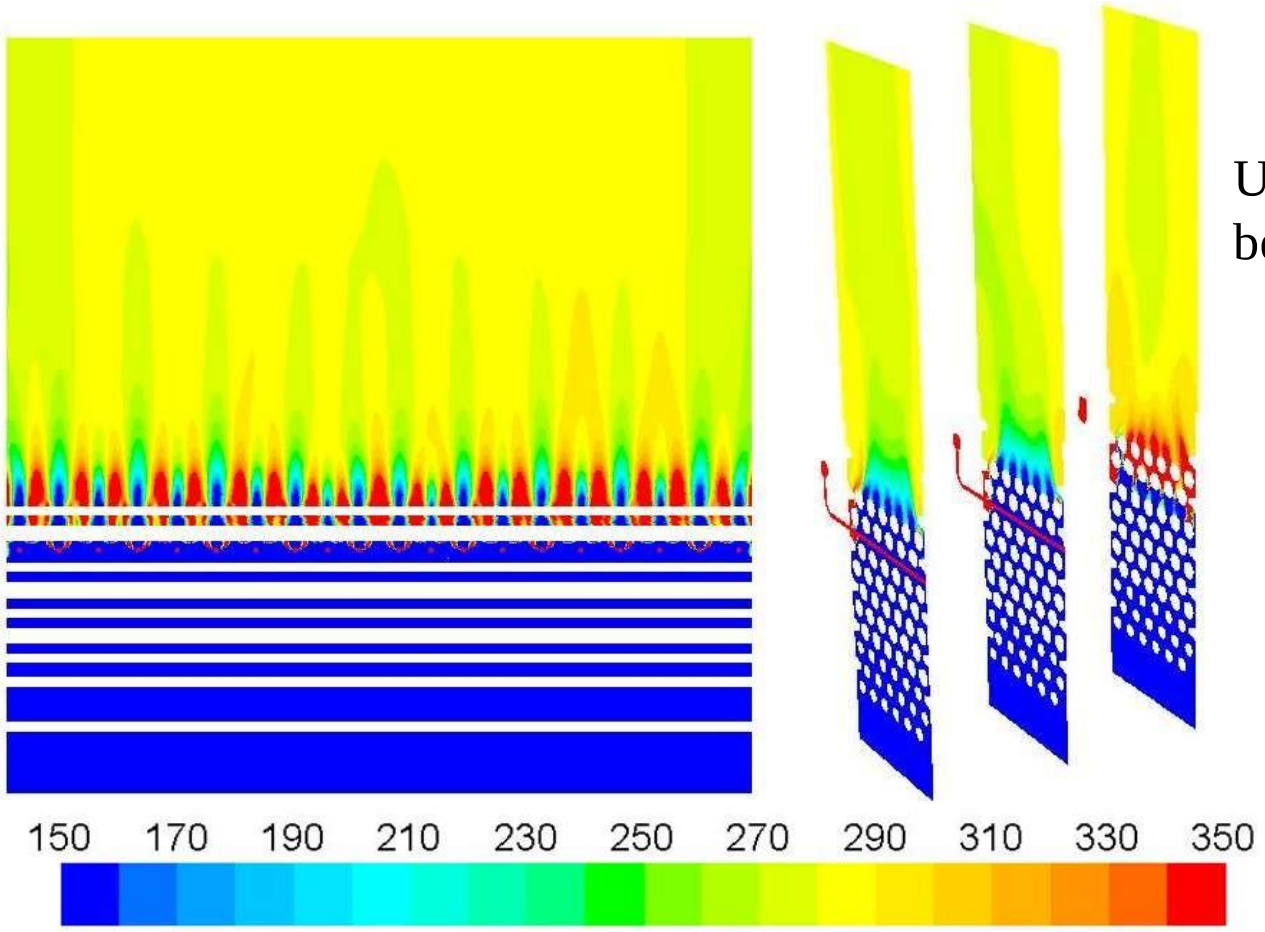
Project #: FIS-439
Scope of Work: Design of Ammonia Injection Grid



Introduction

- New design of Ammonia Injection Grid (AIG) to distribute ammonia-air mixture evenly in the flue gas stream
- Uniform mixing of ammonia-air and flue gas stream was required at inlet of SCR catalyst to ensure proper reduction of NOx
- SCR unit is located above the convection section in this heater
- CFD simulations were carried out for various designs of AIG lances and injection nozzle patterns to achieve uniform mixing

Contours of Ammonia Mass Fraction

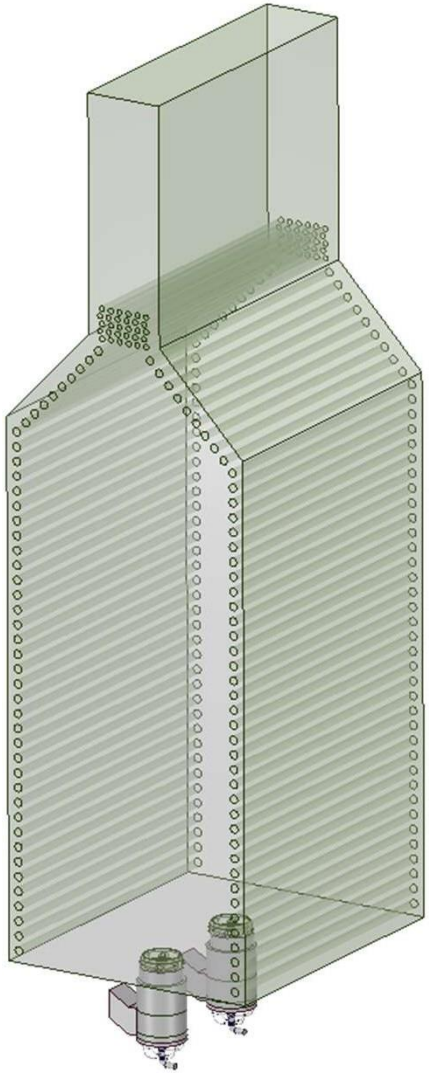


Uniform mixing is achieved before SCR inlet

Summary

- AIG lances were located below two rows of convection tubes
- Design of AIG lances and its location provided uniform mixing within short distance after convection section
- RMS deviation of ammonia concentration at SCR inlet was within ± 2.5%
- RMS deviation of normal velocity at SCR inlet was within ± 4.9%

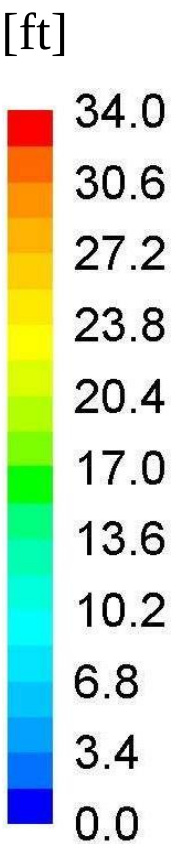
Project #: FIS-402
Scope of Work: Crude Heater Burners

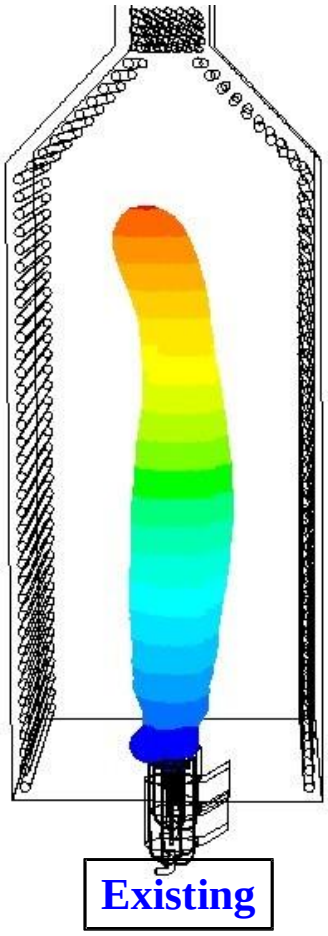


Introduction

- Longer flames and high tube metal temperatures were observed in the radiant section of the heater. This lead to coking of tubes, shorter run length of heater and frequent shutdowns
- CFD analysis was done for existing burner and heater with radiant tubes to know the cause for high TMT. It was proposed to use more number of burners to distribute the heat evenly within the heater
- Proposed burner configuration along with FIS patented inclined firing system was used to eliminate flame impingement and overheating

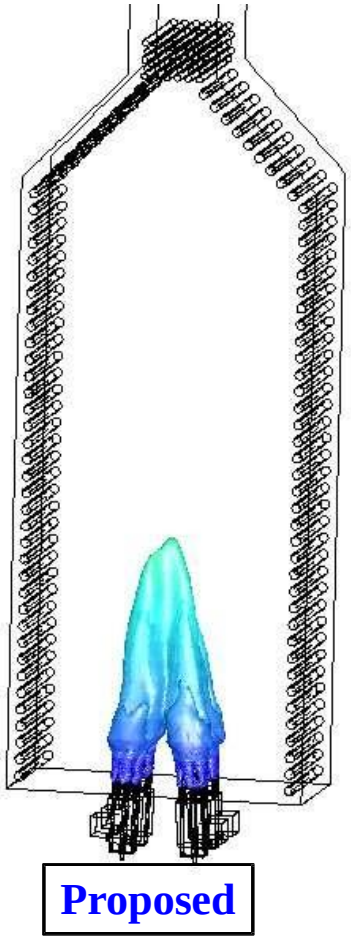
Flames colored by Height





Existing

Flame Height:
34 ft



Proposed

Flame Height:
14 ft

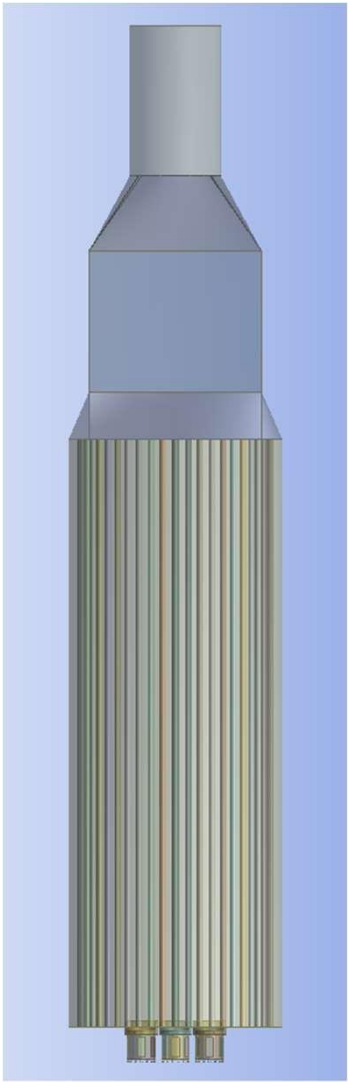
With proposed burner configuration flames are shorter and away from radiant tubes

Summary

- The proposed modification of using more number of burners was implemented
- Post implementation, the flame height was reduced considerably also the maximum tube metal temperatures were reduced
- The heater run length was increased by more than 2 years

Project #: FIS-400

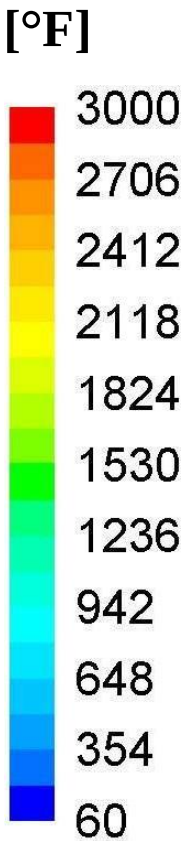
Scope of Work: Hydrocracker Stabilizer Reboiler

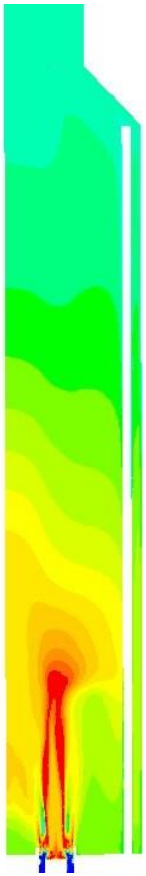


Introduction

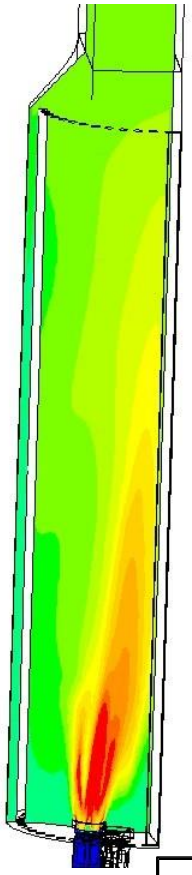
- Inadequate burner clearance is causing radiant tubes to be in the path of the hot flue gases and causing high tube metal temperatures
  - Flue gas flow recirculation due to swirling effect near the burners
- CFD analysis was done for the existing and proposed configurations
  - It was proposed to install spider jet tip inclined towards centre
  - Proposed simulation results showed reduction in tube metal temperatures, improved flue gas flow pattern and external recirculation

Flue Gas Temperature Contours





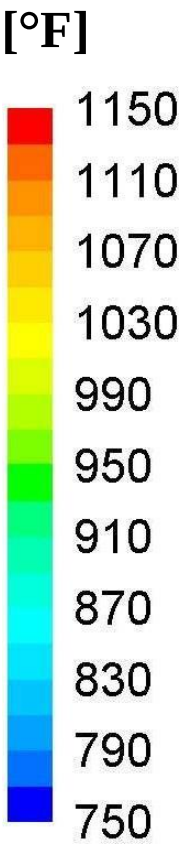
Existing

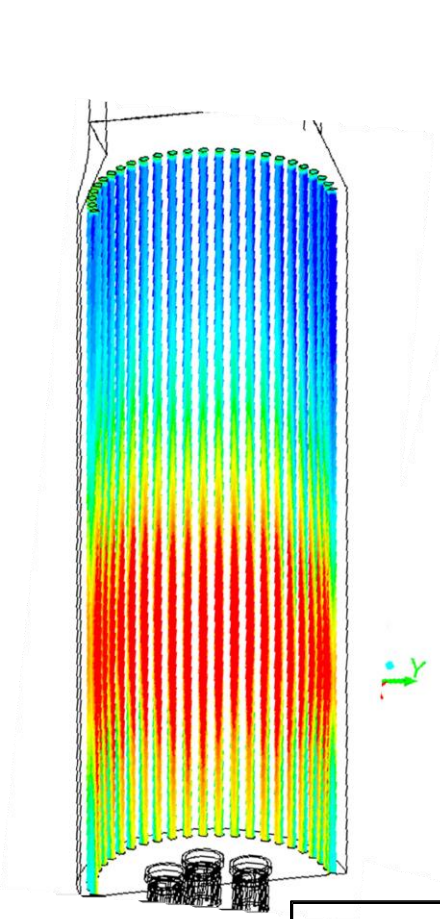


Proposed

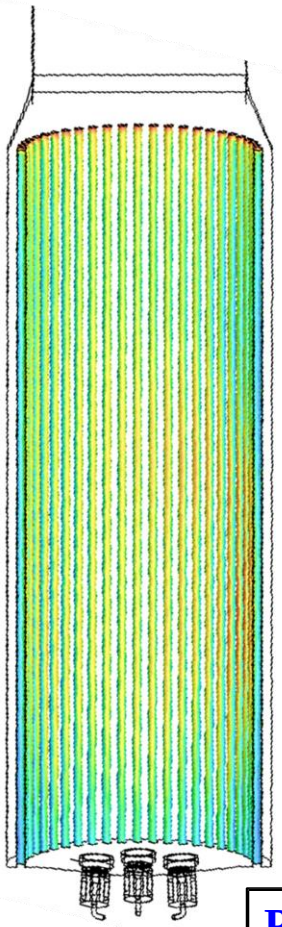
Inclined burner configuration have flames inclined towards the center of the furnace and reduced flue gas temperature around the radiant tubes

Radiant Tube Metal Temperature Profiles





Existing



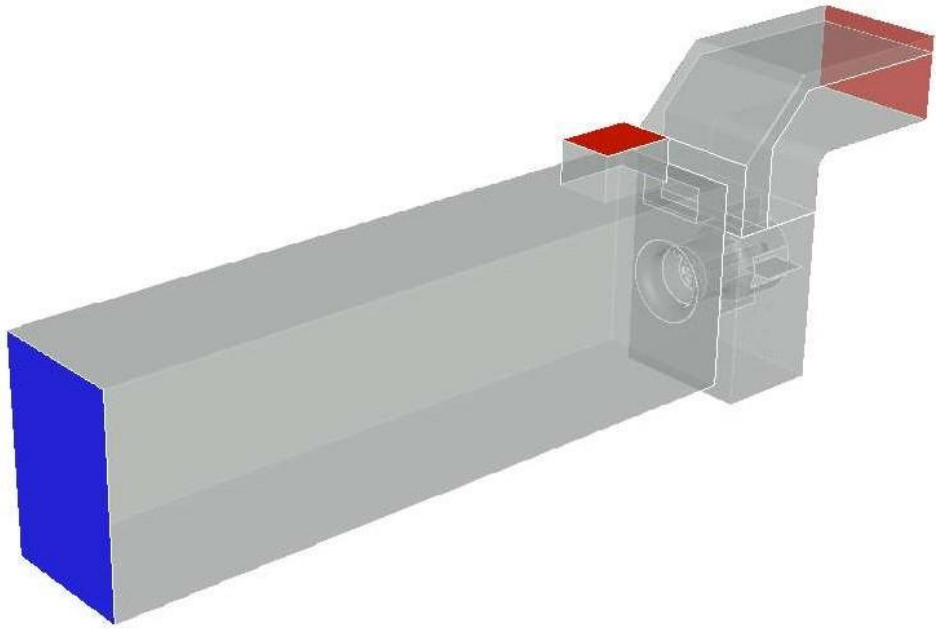
Proposed

Proposed heater configuration has uniform tube metal temperatures, eliminating all the hot spots

Summary

- The proposed inclined burner configuration was implemented in the heater
- Post implementation with external recirculation showed uniform tube metal temperature on radiant tubes
- Reduced flame height
- Increased tube life
- Increased run length

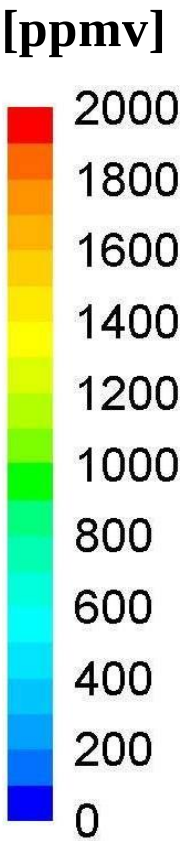
Project #: FIS-399
Scope of Work: CO Heater Burners



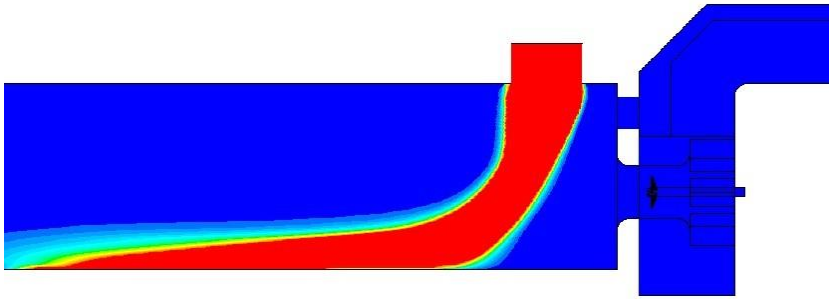
Introduction

- There is a significant amount of CO slippage in the duct causing combustion to be incomplete
- CFD analysis was done for the existing and proposed configurations
  - It was proposed to change the burner and reduce the size of air duct opening
  - Proposed case simulation results showed CO slippage has reduced considerably, causing combustion to occur completely

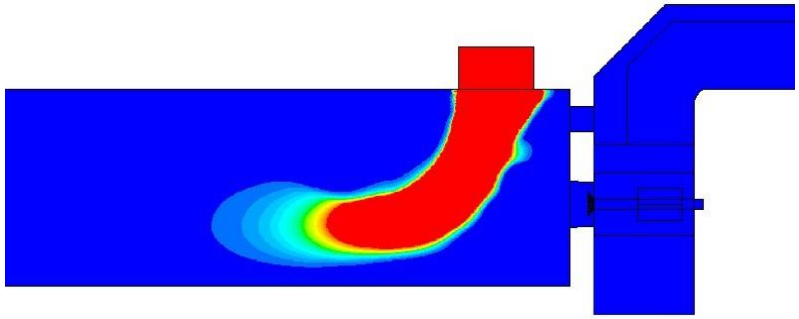
CO Mass Fraction Contours



Modified burner provided better combustion of CO



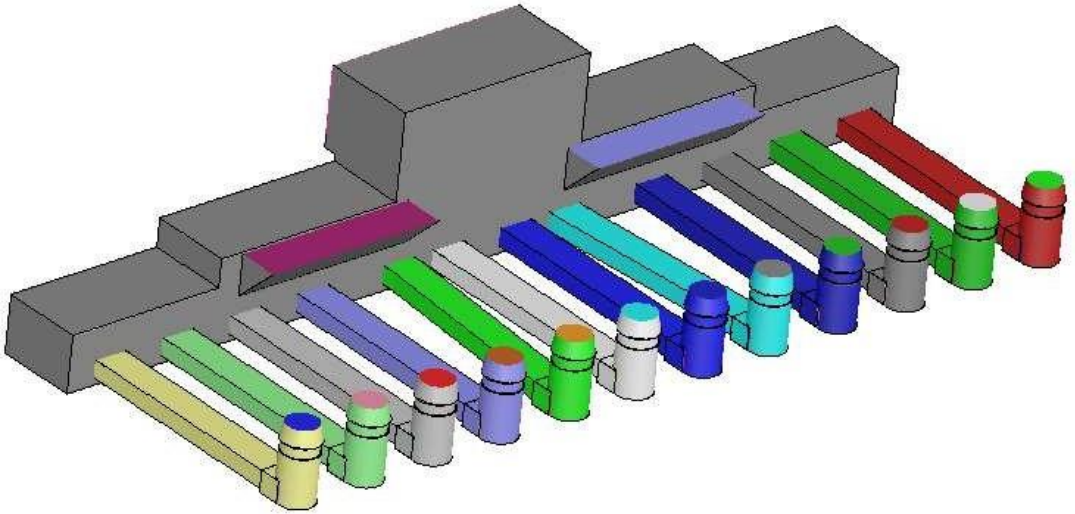
Existing



Proposed

Summary

- For existing case, combustion of CO is not complete which leads to slippage of unburnt CO through the exit of boiler
- With proposed modifications, combustion of CO is complete eliminating any CO slippage at the exit of boiler



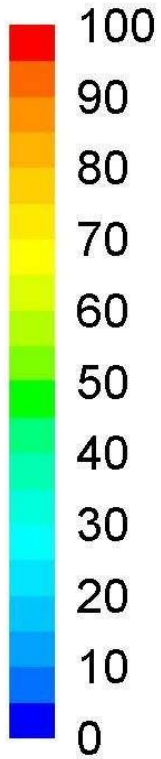
Project #: FIS-396C

Scope of Work: Improving Air Flow Distribution Across all Burners

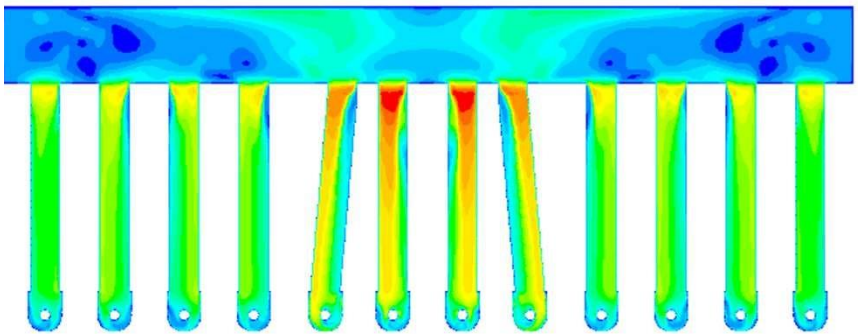
Introduction

- For the existing configuration, there is a significant deviation in air flow across all the burners in the heater
  - High pressure drop variations in manifold and high air velocities in burner duct
- CFD analysis was done for the existing and proposed configurations
  - It was proposed to install splitter vanes at the entry of the duct for forced draft case
  - Proposed simulation results showed there is a uniform air flow distribution across all the 12 burners

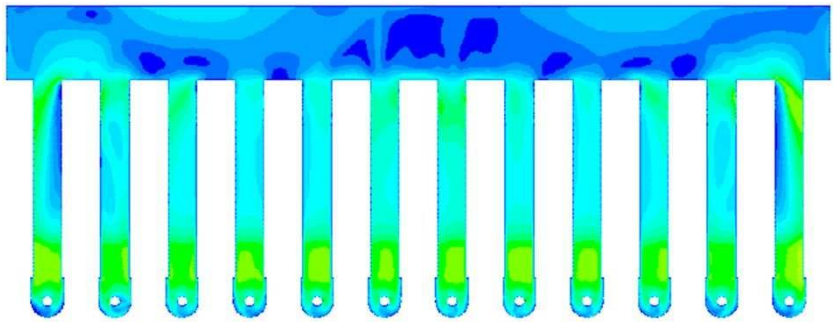
Air Flow Velocity Contours



Proposed duct is having more uniform air distribution among all the 12 burners



Existing



Proposed

Summary

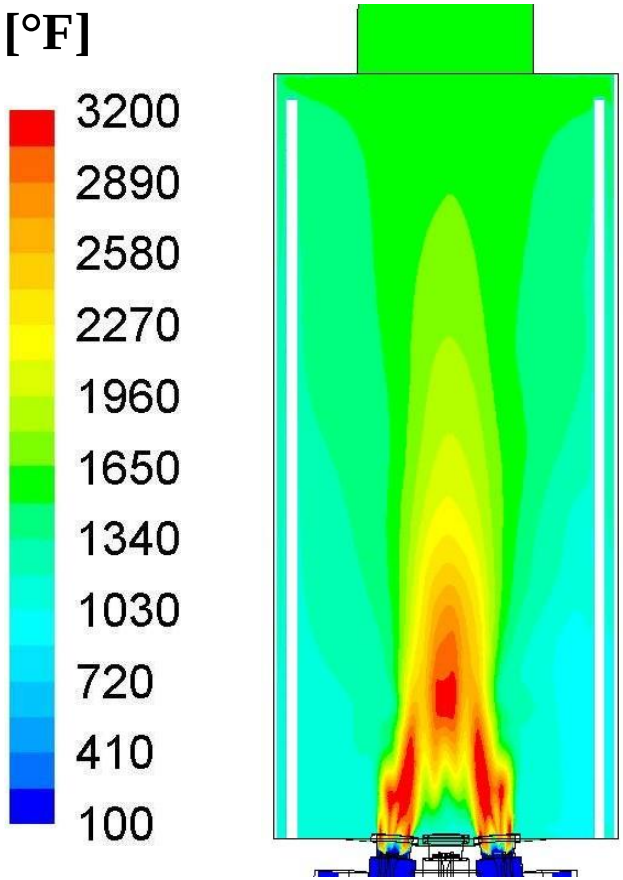
- Proposed modifications were implemented in the burner plenum
- Modifications showed uniform air flow distribution across all burners both for natural draft and forced draft operations

Project #: FIS-392
Scope of Work: Hot Oil Heater Burners

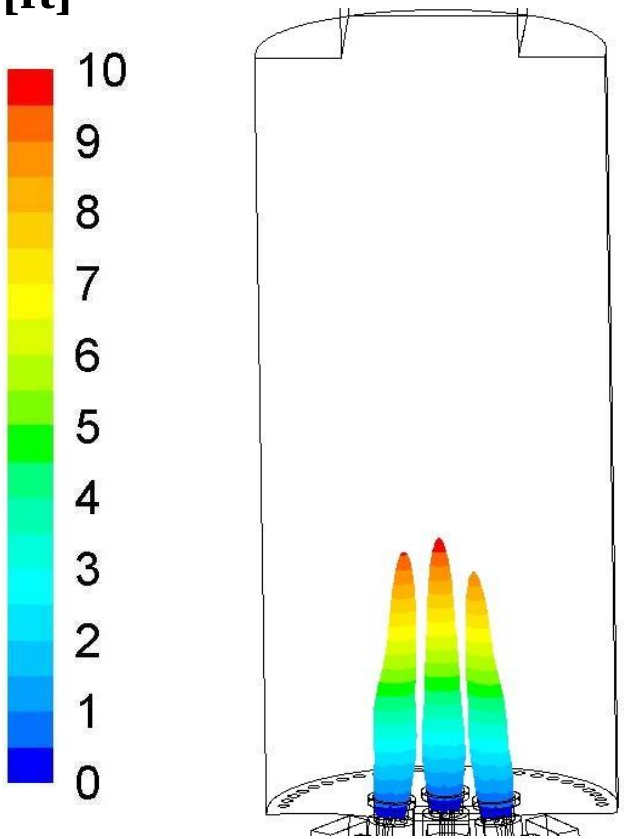
Introduction

- Objective: To study the effects of inclined firing in the heater
- CFD analysis was done to study the flame behaviour and its effects on tube metal temperature
  - 5° inclination of the spider jet burner towards the centre of the heater resulted in reduction of maximum TMT and heat flux in radiant section of the heater

Temperature Contours & Flame Height



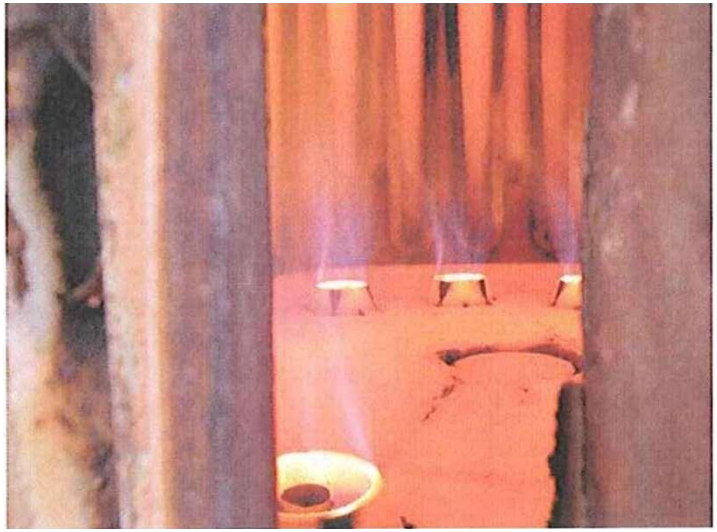
Inclination of the burner has reduced the TMT of the radiant tube with an increase in flame height to 10 ft



Summary

- FIS patented inclined firing system was implemented in the heater
- It showed improved the flame patterns and uniform heat flux distribution across all the radiant tubes

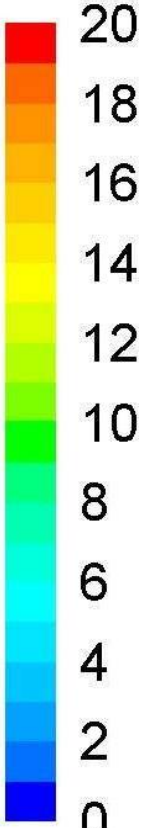
Project #: FIS-356
Scope of Work: Crude Charge Heater

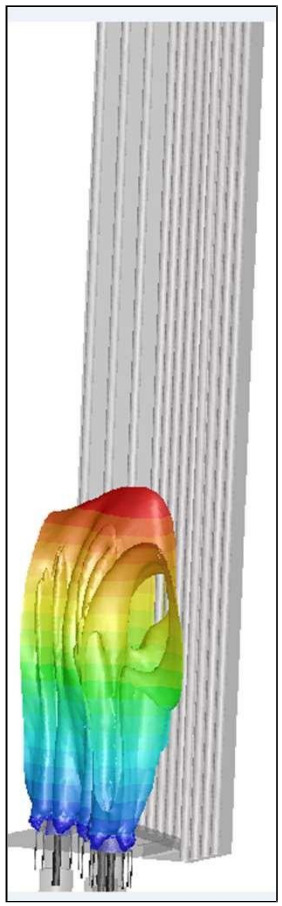


Introduction

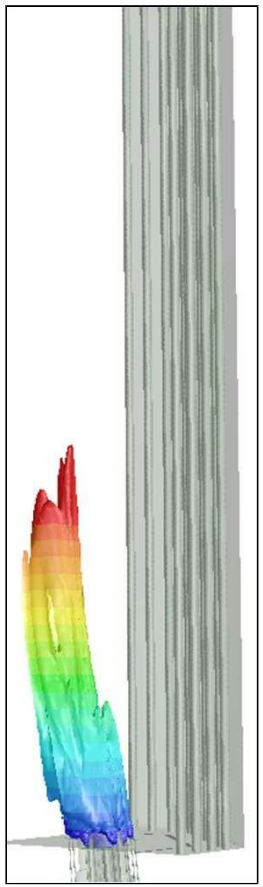
- Client faced flame impingement and high tube metal temperatures in radiant section
- FIS recommended inclined burner firing system to eliminate flame impingement
- CFD modeling of the burners and radiant section was carried out to demonstrate the benefit of the inclined burner firing system

Flame Profiles





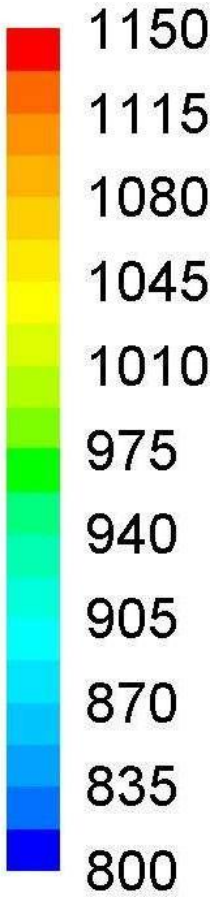
Existing

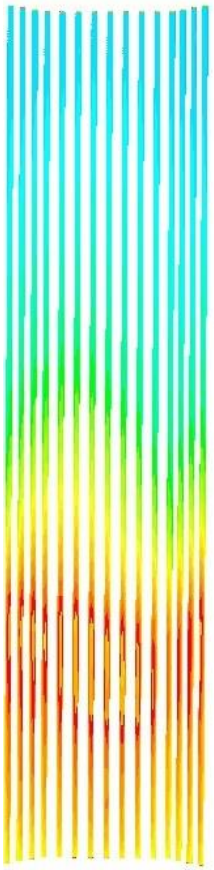


Proposed

Proposed burner configuration improved the flame characteristics. The flames are more concentrated in the center of the heater and no impingement on radiant tubes.

Radiant Wall Temperatures





Existing

Proposed

Proposed burner configuration had more uniform heat flux profiles on the radiant tubes. The maximum radiant tube metal temperature was reduced by 150 °F

Summary

- The proposed inclined burner configuration was implemented in the heater
- Heater performance was improved post implementation
- Client reported almost 150 °F reduction in maximum radiant tube metal temperature

Conclusions

- FIS has a vast experience of analysing various types of heaters using CFD
- FIS engineers are very well versed to utilize CFD simulations to improve the heaters performance and provide solutions to operational issues or limitations
- Please get in touch to discuss improving your heater performance
  - Email: info@heatflux.com