CFD SIMULATIONS TO IMPROVE THE PERFORMANCE OF FIRED HEATERS

Introduction

Computational Fluid Dynamics (CFD) is a simulation tool which solves Navier-Stokes equations for conservation of mass, momentum, and energy. CFD is proven and effective technique widely used to simulate various design configurations and understand heater insights.

CFD modeling can be used to perform detailed combustion analysis of burners, to study the flow and temperature profiles along with the flue gas circulation pattern in the heater. The CFD simulation helps to understand the heater insights in terms of flue gas flow pattern, heat flux distribution, tube metal temperatures, tube side flow distribution, temperature distribution in the heater, etc.

CFD modeling is used to design plenums and internals to achieve uniform air flow distribution across the burners. Optimum design of baffles, turning vanes is derived to reduce system pressure losses for ID fan suction and discharge ducts.

CFD METHODOLOGY

Figure 1 exhibits typical steps for CFD modeling and analysis.

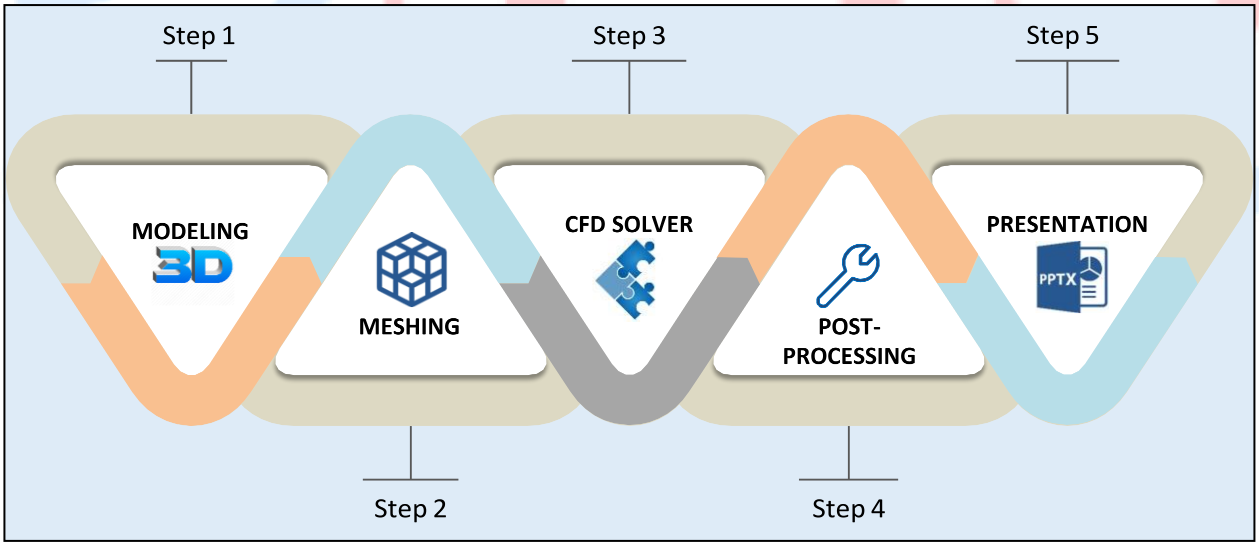


Figure 1: Typical Steps of CFD Modeling

Introduction

Step 1: Detailed 3D model of fluid system to accurately represent the domain to be analyzed.
Step 2: Meshing of the 3D model with appropriate mesh resolution and refinement
Step 3: Setup of models, boundary conditions, computational parameters and running the simulation
Step 4: Analysis of converged solution with the help of contours, vectors, fluxes, plots and animations
Step 5: Combined details from each step to present a simple structure of the methodology and CFD results

FIS has developed best practices to perform CFD simulations for:

- Evaluating burner configurations, to improve heater performance
- Design of plenum and internals to achieve uniform air flow distribution across the burners
- Design of baffles and turning vanes to reduce system pressure losses for ID fan suction and discharge ducts

Typical case study for each of the application mentioned above is presented here.

EVALUATING BURNER CONFIGURATION

FIS was approached by a refinery in Texas to improve the performance of an Atmospheric Charge Heater. The heater had issues with high tube metal temperatures which required prompt shutdown of the heater almost every 3 months.

FIS carried out combustion modelling of burners using CFD to understand flue gas flow patterns, flame characteristics, heat flux distribution and tube metal temperatures. Following were major observations from CFD results:

- Long flue gas recirculation loops, long flames almost 30' length as shown in Figure 2.
- Non-uniform heat flux distribution, top section of heater had higher heat transfer

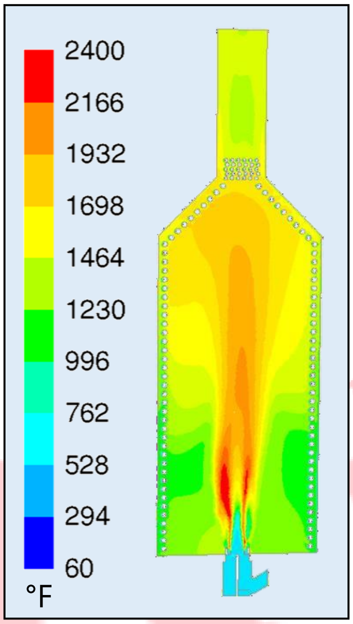


Figure 2: Temperature Contours on a Plane for Existing Heater

FIS evaluated multiple design modifications using CFD analysis. Based on the CFD results it was recommended to replace the existing high capacity burners with more number of smaller capacity burners to reduce the heat release per burner.

Figure 3 portrays the contours of flame colored by height for the proposed burner configuration.

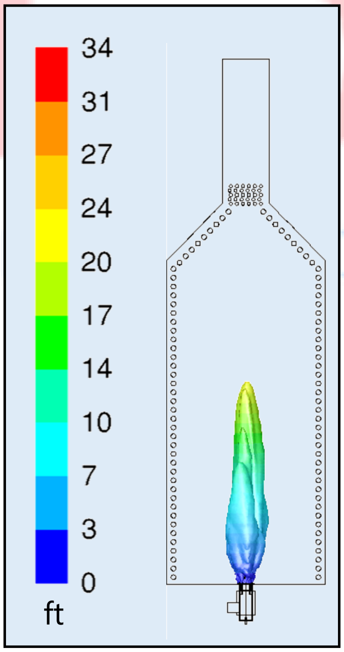


Figure 3: Contours of Flame for Proposed Option

The proposed modification provided following benefits:

- Shorter flame length and improved flue gas recirculation patterns
- Uniform heat flux distribution in the radiant section
- Elimination of hot flue gases and flame impingement, reduced localized coking
- Increased run length of the heater

The heater was successfully commissioned in 2012 with FIS recommended option. The heater run length increased to 18 months. Client is extremely happy with the heater performance.

Case Study

UNIFORM AIR FLOW IN BURNERS

In 2016, FIS conducted the revamp of a vertical cylindrical crude heater for a refinery in Texas. In this heater, the existing burner plenum had maldistribution of air flow across the burners. Extensive CFD simulations were performed to evaluate various modifications in the plenum configuration along with internals to achieve deviation in air flow across all burners to be within ±3 %. Figure 4 portrays the velocity configuration for existing plenum configuration.

Plenum configuration was modified, internal baffles and vanes were utilized in the duct to improve air flow distribution across the burners. Figure 5 shows the comparison of deviation in air flow for existing and proposed configurations.

Heater was successfully revamped with the proposed modifications of burner plenum. In addition, flame profiles improved with the proposed configuration.

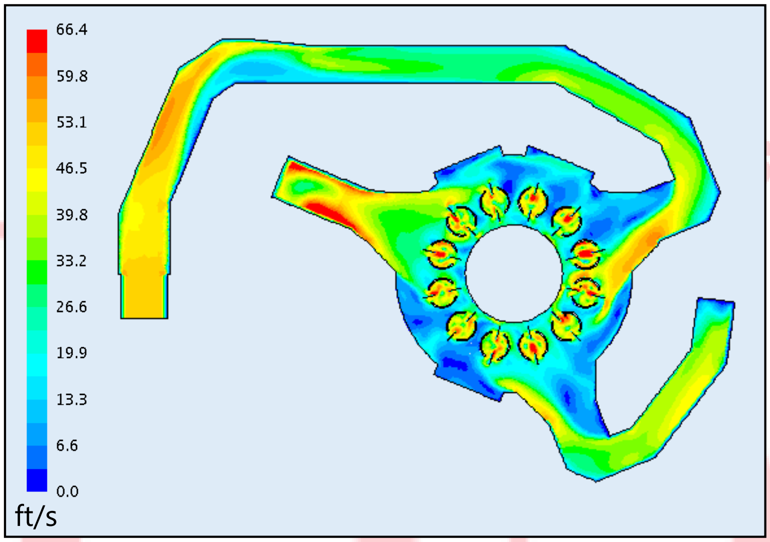


Figure 4: Velocity contours for Existing Plenum Configuration

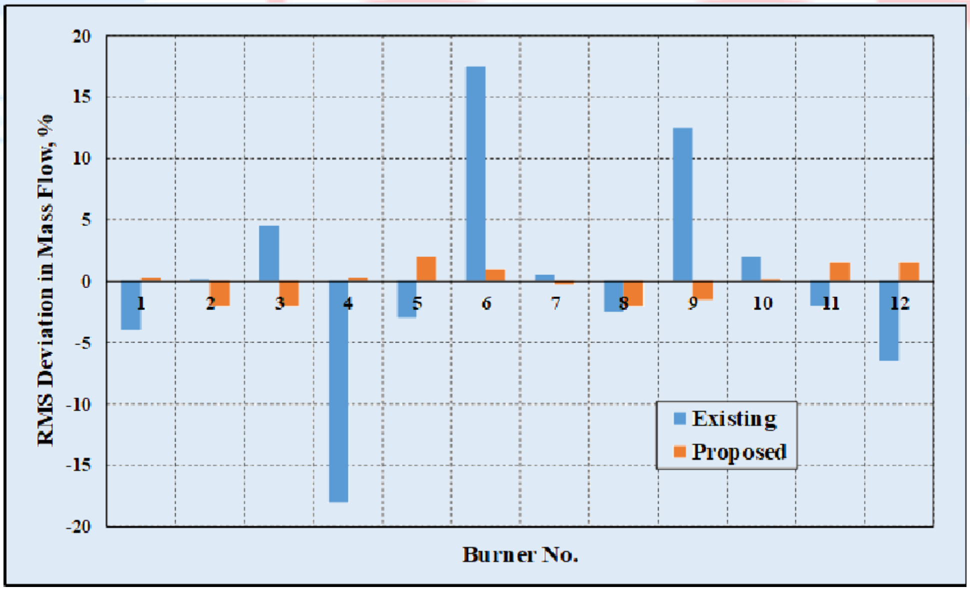


Figure 5: Comparison of Mass Flow Deviation for Existing & Proposed Cases

REDUCING SYSTEM PRESSURE LOSSES

The existing ID fan suction and discharge ducts of the same vertical cylindrical crude heater had higher pressure drop. This was also limiting the flow of flue gas through the heater.

Following were major issues:

- Existing ID fan suction duct had a sharp 90 deg bend. This caused non-uniform velocity profile at the fan inlet.
- The discharge duct connection to stack resulted into a long recirculation loop in the stack.

FIS carried out detailed CFD analysis to evaluate different design modifications to reduce the pressure drop in the ducts.

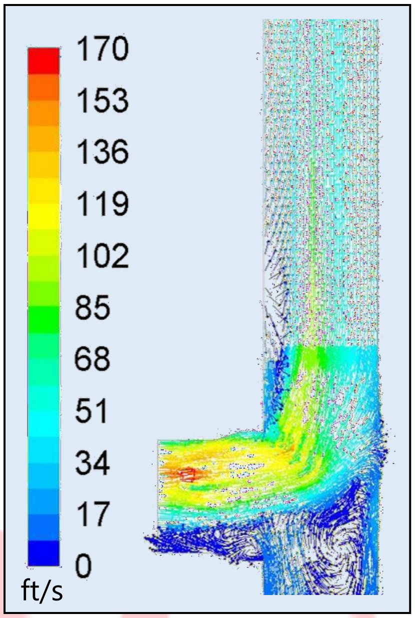


Figure 6: Velocity Vectors on a Plane for Proposed ID Fan Discharge Duct

- ID fan suction duct was modified along with appropriate internals
- Flue gas velocity profile at ID fan inlet was very uniform, with no backflow
- There was a reduction of 0.5 inch WC pressure drop in the suction duct
- Discharge duct profile was also modified with proper internals at specific locations
- Recirculation loop in the stack was eliminated
- Pressure drop in the discharge duct was reduced by 0.25 inch WC

Figure 6 depicts the velocity profile in the ID fan discharge duct. While, Figure 7 exhibits the pressure contours for proposed ID fan suction duct. The heater was successfully revamped in 2016, along with the internals in ID fan suction and discharge duct as suggested by FIS. Lower flue gas duct pressure losses also reduced the ID fan horsepower.

REDUCING SYSTEM PRESSURE LOSSES

cont....

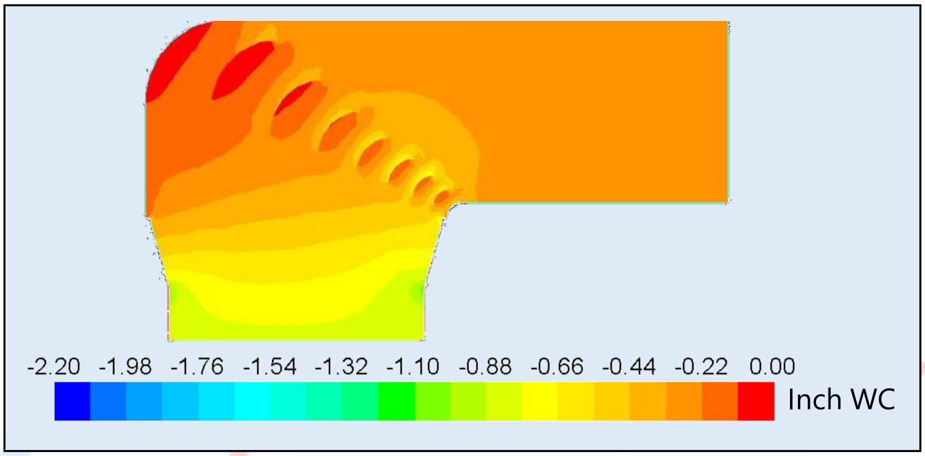


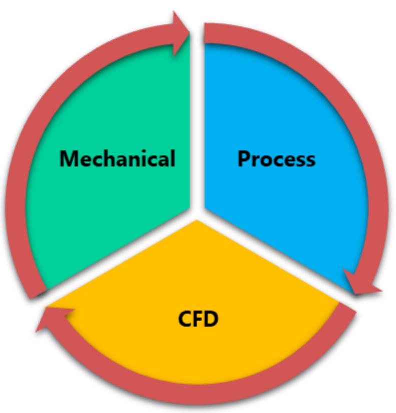
Figure 7: Pressure contours on a plane for proposed ID fan suction duct

Each of the above case study demonstrates how CFD simulations are effective in evaluating the possible modifications and selecting the most optimum design. FIS CFD team has a vast experience of CFD modeling for various applications of Fired Heaters and Boilers.

CFD Modeling

Furnace Improvements has vast experience in Process Design and Computational Fluid Dynamics (CFD) Modeling of fired heaters and boilers. Successful solutions for various types of Crude, Coker, Reformer, Vacuum Charge and Hot Oil heaters have been implemented at various refineries for over the last 23 years.

Application Type	Description	Outcome
Combustion	Evaluate various burner configuration to improve flame characteristics and radiant heat flux profiles	Number of burners and the corresponding layout
	Demonstrate advantages of inclined firing system for reduction in radiant tube metal temperatures	Improved flue gas recirculation pattern, and flame characteristics
Flow	Evaluate design of hot air duct/ burner plenums, to ensure uniform air flow distribution across all the burners	Identify placement of turning vanes, baffles and perforated plates
	Reduce system pressure losses and achieve uniform flow distribution for ID/FD fan ducts	Improved duct design, with turning vanes and baffles
Ammonia Injection Grid (AIG) Design	Design of AIG for uniform mixing of flue gas and ammonia flow at the inlet of SCR unit	Improved AIG grid design, number of lances and holes
Manifold Distribution	Evaluate design of process fluid transfer lines for uniform flow distribution across all the passes	Improved design of manifolds for process fluid



- FIS uses an **integrated approach** working in close coordination with Process and Mechanical groups
- All the 3D models and simulation inputs are verified by Mechanical and Process group respectively
- All the final fabrication GA drawings are verified by CFD group to ensure they are in line with the proposed modifications

Furnace Improvements Services

Uniform Heat Transfer in Fired Heaters

Furnace Improvements Services
1650 Highway 6, Ste. 440
Sugar Land, TX 77478
(281) 980-0325 Fax: 1-877-395-0760

AG Furnace Improvements Pvt. Ltd.
A-74, Ground Floor, Sector-2, Noida.
Uttar Pradesh-201301
Phone:+91-120-4146823

For more information

For more information please contact FIS by email at
info@heatflux.com
or
Visit us online at www.heatflux.com