Tridiagonal™ Solutions

compute | innovate | develop

Performance Enhancement & Profitability Improvements of Refinery Operations

✧ Moving Beyond Correlations & Rules of Thumb to Effective Use of 3-D Flow Modeling Techniques

By

Praveen Gonuguntla and Damo Vedapuri, Ph.D.

Table of Contents

Introduction

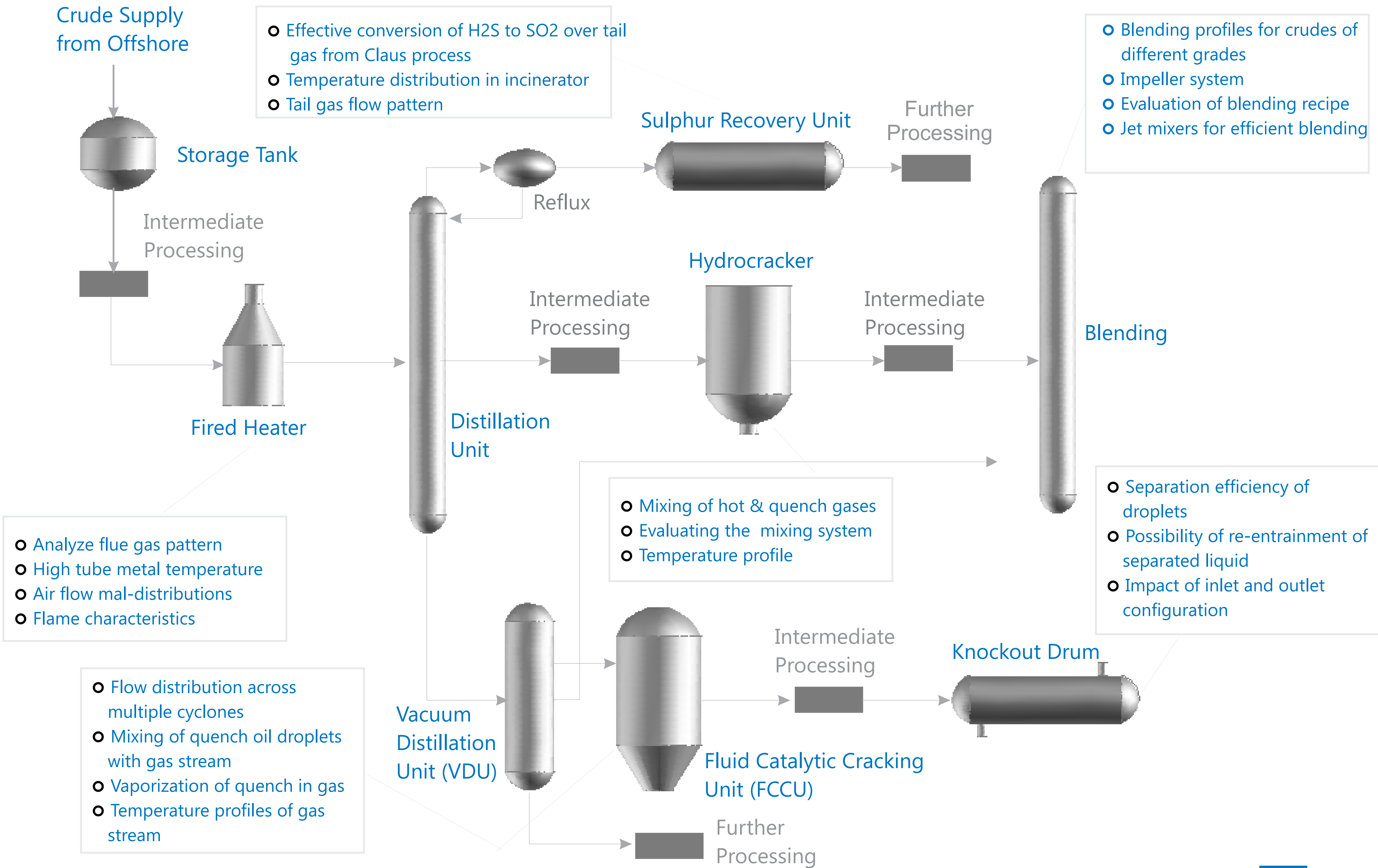
Mixing, Separation and Flow Distribution occurs in almost every refinery process operation starting from crude blending to distillation & cracking to product blending. Efficient operations of Refinery processes rely on optimal flow distribution, fluid mixing and phase separation. Poor flow, mixing or separation bottlenecks a process, reduces its efficiency and results in reduced operational times between shut downs. Optimizing flow can debottleneck a process, lead to an enhancement of the process performance and reduce the frequency of shut down. This then improves the profitability of the refinery operations.

Advancement in simulation technology has enabled building reliable computational models of refinery process units. Computational Fluid Dynamics (CFD) is one of the preferred engineering simulation techniques used for improving flow performance. CFD is a science which is based on the physical laws governing the fluid flow under various conditions. This technique can be applied for diagnosing operational issues, improving the performance of existing processes and design of new process equipment and components. CFD has been successfully applied to various applications involving blending, flow distribution, heat transfer, mixing, separation, erosion, reactions and particulate flows. CFD has enabled engineers to go beyond the normal, empirical correlations and expensive physical testing to optimize product & process design.

Tridiagonal Solutions has expertise in developing computational models that predict the unit operation performance close to the plant observations and evaluate multiple design options to improve the performance. Tridiagonal Solutions collaborates with clients to increase their performance, by providing unparalleled experience, comprehensive performance enhancements, as well as product development solutions across a multitude of industries.

The following is a list of examples to showcase how Tridiagonal has used CFD technology for process improvement in refineries.

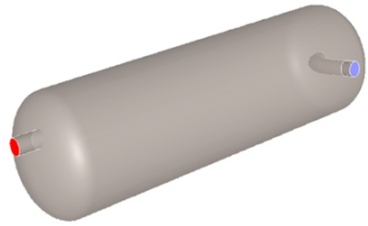
Tridiagonal : CFD Applications in Refining



Enhancing performance of a Knockout Drum

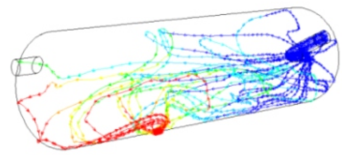
The Knockout drum (KOD) is one of the crucial pieces of equipment in a flare system. The KOD is expected to efficiently separate entrained liquid from vapor stream and prevent the liquid from being transported to the flare. Improper functioning of KOD could lead to a phenomenon known as flaming rain, which occurs when liquid droplets are blown out at the top of a flare stack, along with the ignited process gases. It is always desirable to have the highest separation efficiency of

The sizing of KOD is generally based on API standards. A key assumption in the API procedure to size a KOD is that the fluid stream achieves a plug flow profile once they enter the KOD. However based on the inlet configuration the flow profiles could deviate from the plug flow behavior. This reduces the separation efficiency since adequate residence time is not achieved. Further the flow profile could promote re-entrainment of liquid from the liquid pool at the bottom.

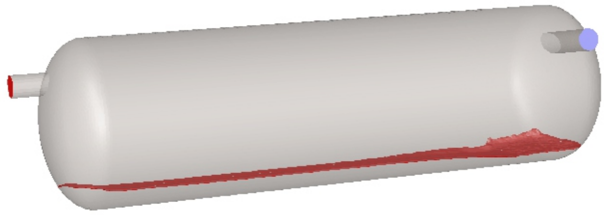


3D model of the Knockout Drum

When there are changes in the process condition, such as an increase in through put or a larger turn down ratio, it is possible that the actual operations of KOD will deviate from the design basis. While replacing the existing KOD with a new one to cope with the new process conditions is often discussed, it is possible to improve the performance of the existing unit through retrofits.



Droplet trajectories colored by residence time showing the



Separated liquid layer at the bottom of KOD was considered as initial condition to study the re-entrainment

Iso-surface of liquid volume fraction shows that the separated liquid layer is being re-entrained to the vapor phase.

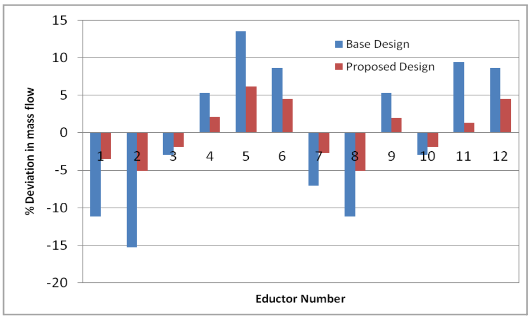
CFD analysis captures the detailed flow phenomena occurring for the actual operating conditions. The analysis helps visualize how the liquid droplets are transported in the drum. Flow features that cause entrainment and poor separation efficiency can be identified. Retrofits to improve performance can then be identified to improve efficiency. In a recent study several retrofit options were considered. Changes to inlet nozzle design, use of baffles to reduce entrainment, inducing a cyclonic flow instead of plug flow were evaluated. The CFD modeling framework provides an ideal environment to test a number of these concepts in a relatively short time frame and identify the most appropriate solution.

Using CFD for Evaluating Crude Oil Blending with Jet-mixers

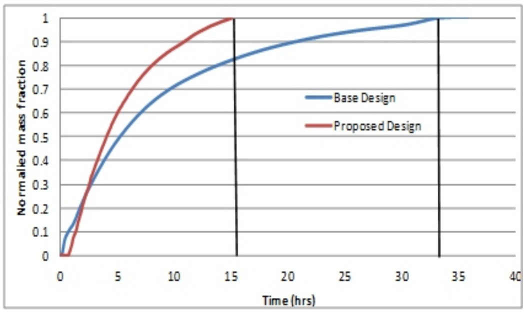
Crude oil blending is used to blend oils of different grades in different proportions in order to produce consistent crude of desired properties. It is important that the crude oil blending operation is able to produce crude of consistent properties that could be sent downstream for refinery operations. Blending is a very crucial operation. An inappropriate blend configuration could lead to blend errors resulting in substantial losses.

Multiple configurations are utilized to achieve the blending of crudes. Hot oil is passed through an arrangement of coils, which then creates a density differential. This helps in setting up natural circulation loops which in turn helps in the blending process. Certain tanks have side entry impellers forcing the oils to the bottom and then forming circulation loops to blend the crudes. An efficient way of blending could be to use a circulation pump and pass the flow through a specific arrangement of jet-mixers. These jet-mixers have suction to motive fluid ratio of 3 to 4. Hence, the effective circulation rate is enhanced. The key to this approach is the layout and orientation of the jet-mixers in order to achieve the desired blending within a stipulated time.

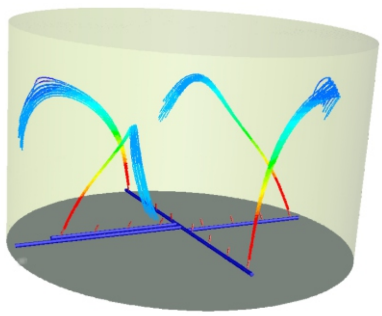
These tanks are huge, having approximate diameters in the range of 30-50 meters and height of about 20 meters. It is important to understand the flow patterns and mixing of crudes within the tank based on the layout of jet mixers.



Comparison of mass flow distribution across the jet-mixers for base and proposed design



Comparison of blend time for base and proposed configuration



Liquid jets through few mixers giving a uniform flow distribution

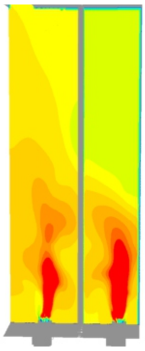
CFD based simulation has been utilized to evaluate multiple configurations of the jet-mixers to understand their impact on the blending process. As in real time operations, the grades of oil and the corresponding proportions will keep on changing depending on the market conditions. Simulations have also been used to study the blending process over a wide range of operating conditions for a fixed configuration of jet-mixers. Based on these simulations an operational

Tridiagonal Solutions has developed a modeling frame work to simulate the blending process and provide valuable inputs in designing the blending system for large tanks.

Improving Fired Heater Operations using Computational Fluid Dynamics Modeling

Fired heaters, furnaces and process heaters account for a majority of the energy costs involved in the refinery operations. Significant savings could be achieved even with a marginal improvement in heater operations. Some potential issues in a fired heater operation involve flame impingements on the tubes, inappropriate flue gas circulation patterns and flame-flame interaction between adjacent burners. All of these situations can lead to elevated tube metal

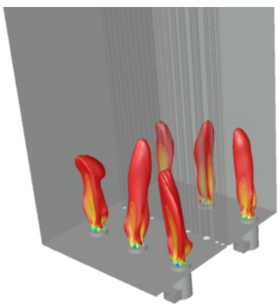
Process heaters are simulated using tools like FRNC, etc. These tools provide understanding about the heat-flux profiles and the temperature distribution within the tubes, assuming the heater is maintained at a uniform temperature based on the firing and heat generation rates. However the burner design, its arrangement and orientation with respect to the tubes and its proximity to wall or tubes impact the flame characteristics and the temperature profiles within the heater. CFD modeling of burners with detailed combustion models, considers intricate details of fuel tips and the orientation of burners. This helps to understand the heater operation in great detail.



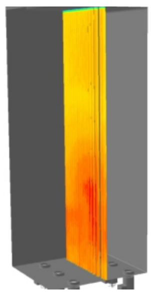
Temperature profiles in the radiant section

CFD modeling is capable of predicting the shape and size of flame, heat transfer within the radiant section, flame impingement on the tubes, tube metal temperature distribution, heat flux profiles and flue gas flow patterns. Key flow features such as the presence of any re-circulation loops near the burner and the interaction of flames between two burners are identified.

Flue gas flow pattern in the radiant section, path lines colored by temperature.



Flame shape colored by temperature, shows the orientation with respect to radiant tubes.

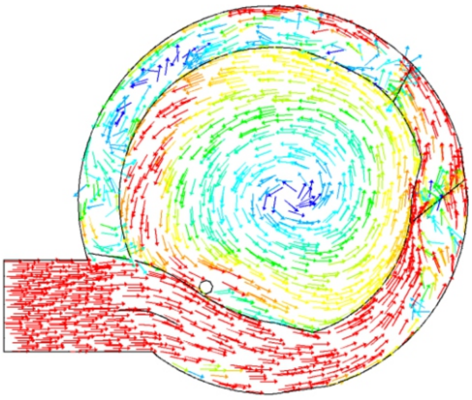


Radiant tubes colored by temperature shows the high tube metal temperature for few tubes in the lower region

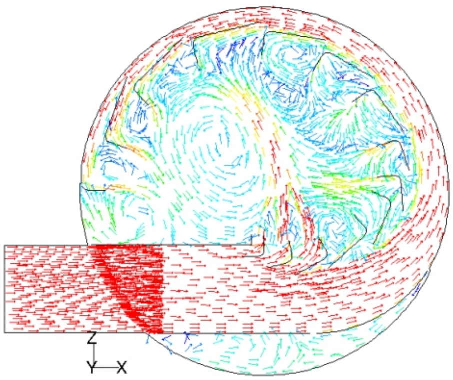
Tridiagonal Solutions has simulated a number of process heaters and provided detailed analysis based on which modifications were identified to improve the heater performance. The model predictions have been validated against the field observations for the fuel and air side pressure drop, flame length and tube metal temperatures.

Modeling of Vapor Horn using Computational Fluid Dynamics (CFD)

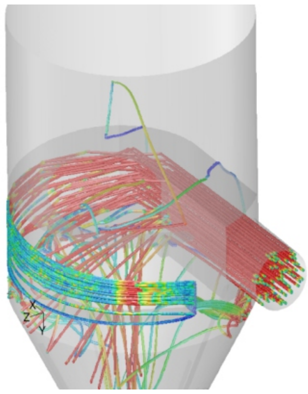
Refinery fractionators that have a flashing feed require pre-separation of the any free liquids entering the fractionator. The two-phase vapor-liquid mixture enters the unit at a very high flow rate and it is essential to have good separation of the vapor and liquid for enhanced performance of the unit. A vapor horn is one of the widely used inlet device for this process. The measure of performance of the vapor horn is based on its ability to adequately separate the vapor and liquid fed into the unit. There are multiple variations in the vapor horn configuration to obtain good centrifugal flow pattern for separation. Spacing is limited and after separation the vapor needs to flow axially upwards in a uniform manner. Performing cold flow testing at the real scale would require substantial efforts and are time consuming. CFD based modeling provides the opportunity to review and understand the flow patterns within the tower along with the vapor and liquid distribution. Radial and tangential flow generated by the various configurations of vapor horn internals could be analyzed from the CFD simulations.



Flow pattern across the cross section of vapor horn showing the swirling flow pattern for the base design.



Flow pattern for the proposed design with low velocity at the central region improving the separation of droplets.



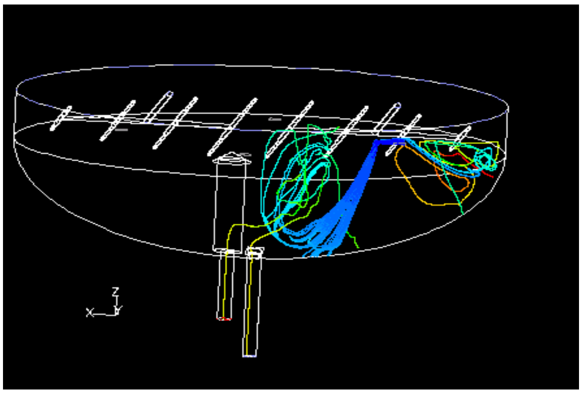
Droplet trajectories for the proposed design showing improved separation

Droplet trajectories are predicted to determine the separation efficiency for various designs over a range of droplet diameters. The complete analysis of the droplet trajectories provides an insight into the operations of the unit and help select an appropriate vapor horn design.

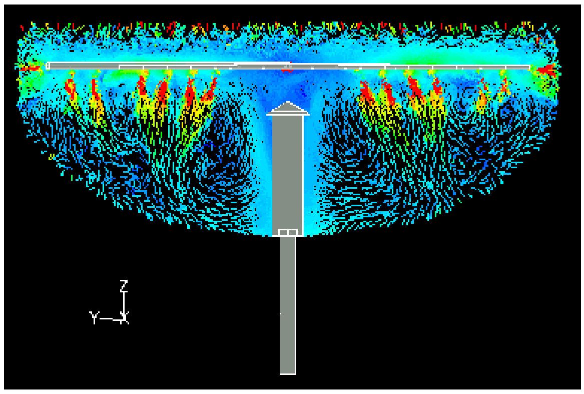
Tridiagonal Solutions has simulated multiple designs of vapor horns with radial and tangential entry, and evaluated the flow patterns and effectiveness of the separation in the towers. CFD proves to be a very efficient simulator in designing the inlet devices and is able to account for the effect of tower design including internals like tray, baffles etc in the system.

Trouble shooting coking issues in Fractionator using Computational Fluid Dynamics

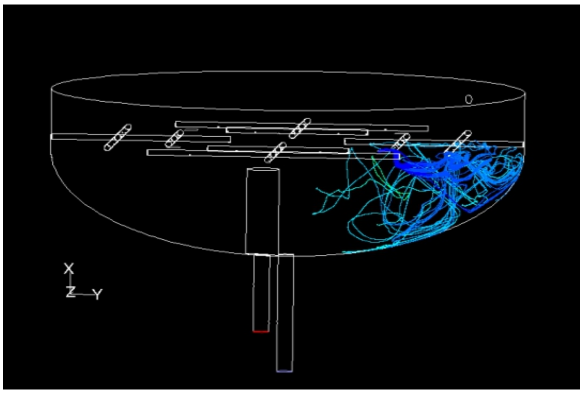
Coking is a critical issue in most of the fractionators installed in refineries. Coking leads to build of pressure due to blocking of pipes and nozzles, and subsequently, intermittent shut-downs. At the bottom of the fractionator it is desirable to avoid dead zones and have sufficient velocities so that the coke particles don't settle and agglomerate. For some fractionators, liquid from the bottom is continuously recycled through a distributor to address the coking issue. In the design of a liquid distributor it is critical to generate jet plumes such that desired flow pattern is achieved and the probability of coke particles blocking the outlet is minimized. CFD based simulations provide insights in evaluating the flow patterns generated by the distributor. Analysis provides details on the extent of jet circulation loops and if any low velocity region exists.



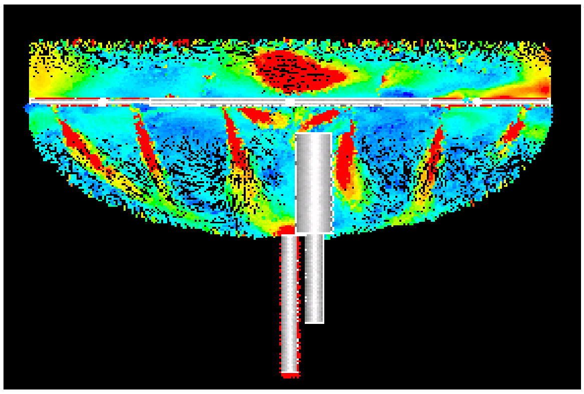
Particle trajectory showing the suspension and settling of particles



Velocity vectors showing the penetration of jets creating flow circulation



Particle trajectory showing the suspension and settling of particles for an alternative design



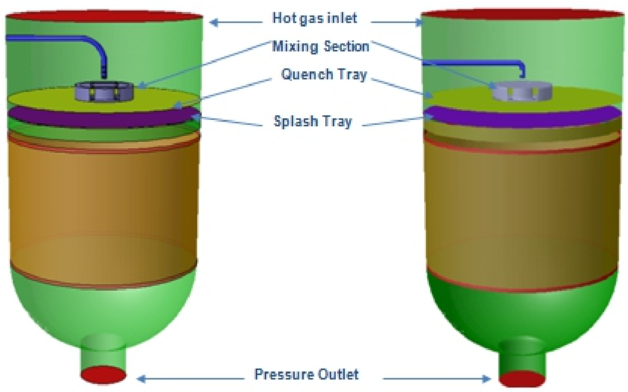
Velocity vectors show much more penetration of jets with high velocity helping in suspension of particles

Tridiagonal Solutions have evaluated multiple designs of liquid distributors and provided detailed analysis comparing different parameters such as velocity field, circulation patterns, and particle trajectories. These simulations help in design of the distributor configuration to address the coking issues in the fractionators.

Improving Temperature Profiles in a Fixed Bed-Reactor using Computational Fluid Dynamics

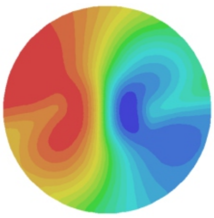
Fixed bed reactors are utilized in refineries for multiple processes such as hydrocracking, hydrodesulfurization, middle distillate hydrodewaxing etc. These are typically exothermic processes and the reactor configuration generally consists of multiple catalyst beds stacked on top of each other with a quench system in between each bed to control the temperature and redistribute the gases uniformly across the bed cross section. For enhanced performance of the processes it is desirable to have uniform temperature profile across the bed cross section. A typical quench system consists of a distributor at the top of the reactor to inject quench gas, a mixing zone to mix the hot and cold gases and then a redistributor on the top of the catalyst bed to insure flow is uniform across the bed cross section. CFD based simulations provide valuable insight into the effectiveness of the design of distributor and the mixing zone, along with the distribution of temperature in the catalyst bed.

The performance of a quench system in a fixed bed reactor was evaluated. The operator was observing poor performance and wanted to understand the flow and temperature distribution in the catalyst bed. A CFD analysis revealed that the temperature distribution across the bed cross section was non-uniform. To improve its performance the quench system needed to be modified. A number of modifications were evaluated and a suitable design that improved the temperature distribution in the bed was selected. Adjacent image shows the temperature distribution for the existing configuration.

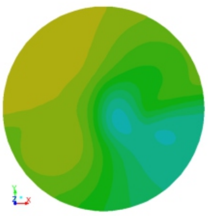


Geometrical details for the base and proposed design

Instead of a single opening for the quench, a design with multiple holes on the periphery of the pipe was evaluated and it showed a reduced variation of temperature in the bed.



Non-uniform temperature distribution across the cross section for base design

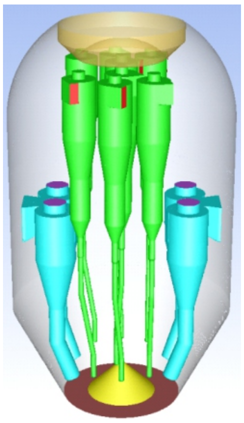


Improved temperature distribution for the proposed design

Tridiagonal Solutions has performed analyses of multiple fixed bed reactor systems. The study involves evaluating the existing distributor and mixing system and providing possible design modifications to improve the mixing and flow uniformity across the cross section in order to improve temperature distribution profiles within the reactor.

Analyzing the performance of a Fluid Catalytic Cracking cyclone unit

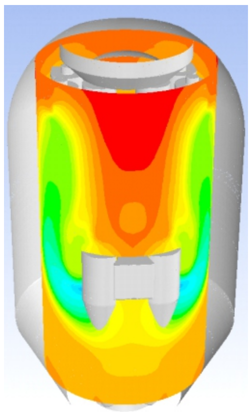
Fluid catalytic cracking (FCC) is the most important conversion process used in petroleum refineries. The reactor and regenerator are considered to be the heart of the fluid catalytic cracking unit. The preheated high-boiling petroleum feedstock consisting of long-chain hydrocarbon molecules is combined with recycled slurry oil from the bottom of the distillation column. This is then injected into the catalyst riser where it is vaporized and cracked into smaller molecules of vapor in the presence of catalyst. In the FCC process, the spent catalyst is rapidly separated from the hydrocarbon product in the riser effluent in the reactor. The hydrocarbon product is immediately quenched to an un-reactive temperature. Effective quenching is provided in a catalytic cracking unit to increase product yield and to decrease thermal cracking. CFD simulations provide greater insight for the flow analysis, and temperature distribution within the FCC unit.



FCCU internal geometry details

Velocity distribution across the FCCU

Path lines colored by temperature showing evaporation of the quench spray



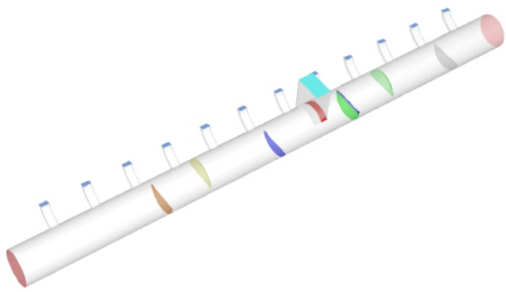
Temperature distribution on a cylindrical plane under the influence of quench spray

The current case study focused on using CFD analysis to determine the uniformity of flow distribution between the six second stage cyclones and the effectiveness of quenching by oil injection in the reactor vessel. Details of the velocity field and the temperature distribution in reactor are analyzed. Quenching of the hydrocarbon stream by injecting oil droplets considerably reduces the temperature in the reactor to create unlikely conditions for thermal cracking. High velocity is observed near the cyclone outlet. The hydrocarbon vapor is well distributed in the unit.

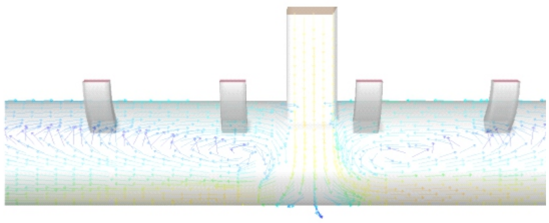
The analysis of Fluid Catalytic Cracking Cyclone unit was performed at Tridiagonal Solutions. The analysis evaluated the mixing of the quench oil droplets with the gas stream leaving the rough cut cyclones, the uniformity of distribution of gas between the second stage cyclones

Improving air flow distribution across multiple burners

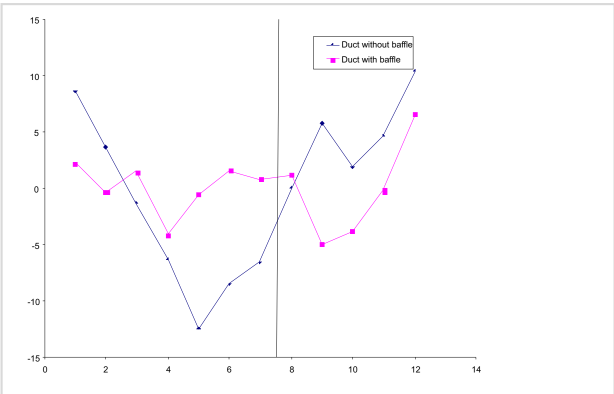
The overall combustion efficiency for a process heater or furnace is a strong function of the fuel and air flow distribution across the burners. Usually a manifold is used to distribute the air flow across multiple burners and there is a single inlet to the manifold. As these manifolds have a length of the order of 20-30 feet, usually there exists a variation in air flow distribution across the burners. The air flow mal-distribution across the burners leads to improper combustion characteristics within the heater, leading to non-uniform temperature distribution and reducing the overall efficiency of the unit. CFD analysis of the manifold or the ducts provides a detailed flow analysis along with the mass flow distribution across all the burners. Based on these results, possible design modifications on the manifold interior are suggested to improve the flow distribution across all the burners.



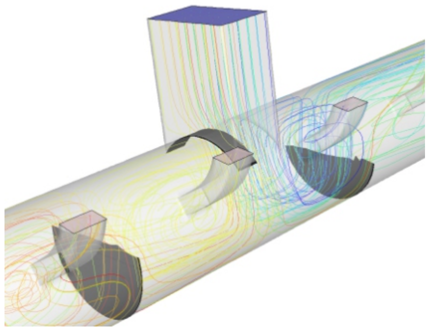
Geometry of duct along with internal baffles



Flow distribution across the two sides of the duct showing uniform flow on either side



Comparison of deviation in mass flow rate for all the outlets for base and proposed duct configuration



Path lines showing the flow pattern within the duct

Tridiagonal has performed analysis of many manifolds and duct systems for multiple process heaters with the number of burners in the range of 12-20. Analysis was performed for existing configurations and possible design modifications were suggested that improved the air mass flow distribution across all the burners.

About Tridiagonal

Tridiagonal Solutions, founded in July of 2006, is a company formed by experts from national laboratory and experienced CFD software professionals from the industry. Commencing from a humble beginning, it has now grown into an organization with more than forty five team members, approximately 85% of whom are highly qualified engineering professionals. Tridiagonal employees have 75 years of combined experience. We are committed to increasing our clients efficiency and profits, by developing innovative engineering solutions based on a true understanding of industry requirements.

Tridiagonal Solutions, with its wide range of expertise and experience provides solutions and services for various industries like:

- Power generation
- Chemicals and Process
- Oil and gas
- Pharmaceuticals
- HVAC

Tridiagonal's contribution in achieving your business goals

Engineers & Scientists at Tridiagonal could help your company be more efficient and increase your bottom line by using simulation technology. We could evaluate potential modifications which will enhance the process performance in your business. We operate a flow lab and have expertise in conducting cold-flow experiments for a wide range of process sytems. Experiments could be performed to collect data that will be used to better understand the system and provide input and close the gap between modeling and actual operations of the unit. The combination of simulations along with experiments is used to provide practical solutions which have been implemented and verified to enhance the overall performance of the unit.

Reach us at:

12703 Spectrum Drive, Suite 101
San Antonio, TX 78249 USA
Phone: 210 372 0816
Fax: 210 468 0699

For enquiries: write to info@tridiagonal.com