SCR DESIGN OPTIMIZATION

Abstract:

Selective Catalytic Reduction (SCR) is a proven technology for deNOx of flue gas streams. SCR is used for coal, oil, and natural gas fired combustion systems. The performance of the SCR system, however, requires careful design consideration related to process conditions such as the flue gas velocity patterns, temperatures, ammonia injection technique, and the proper mixing of ammonia and NOx. For coal fired boilers, the presence of flyash in the flue gas stream can cause erosion and/or pluggage of the catalyst. If these parameters are not properly optimized, deNOx performance can be degraded and the plant can experience increased O&M costs for ammonia usage, pressure drop, and catalyst life.

This paper will discuss best practices in SCR system design and operation related to the key factors of:

- Ammonia injection methods
- Static mixer design
- Achieving optimal deNOx performance
- Minimizing pressure drop
- Avoiding ash pluggage and erosion of catalyst
- Flow modeling goals, methodology, and accuracy of results

Examples from various past SCR designs will be presented, comparing the performance of each. The applicability of the various designs to SCR systems in India will be discussed and summarized.

Authors(s)

Robert G. Mudry, P.E.
Airflow Sciences Corporation
Livonia, Michigan, USA

Table of Contents

1 Introduction

A Selective Catalytic Reactor (SCR) for a coal-fired power plant involves many engineered components. In order to ensure optimal operation, long life, and maximum De-NOx of the flue gas, careful design of all these components is critical. This includes such factors as general arrangement, structural design, construction materials, catalyst chemistry, safety considerations, maintenance planning, chemical process design, and fluid dynamic design. All become equally important for optimal SCR operation.

This paper focusses on the fluid dynamic design for SCRs. At the basic level, the NOx molecules in the flue gas must be brought into contact with the catalyst surface. This may be simple from a theoretical view, but in an operating plant there are many design tradeoffs, including:

- Capital cost of the system
- Ongoing costs of ammonia and catalyst
- Operational costs of pressure drop and flow uniformity
- Maintenance costs due to erosion, corrosion, and pluggage

This paper explains the importance of the various design factors based on the author's experience in designing SCR systems for coal and gas fired plants over the past 15 years. Design objectives, engineering methods (flow modeling) and best practices are discussed in detail.

2 SCR Design Objectives

There are number of goals related to flow patterns within the SCR system that are intended to optimize performance. As is typical with engineering design, these objectives often work counter to each other, and the design process requires a careful balance between the goals. Tradeoffs between competing goals must be considered when making the final decisions for system design. The sections below lists the typical goals that must be considered when designing an SCR for a coal-fired power plant.

2.1 Uniformity at the Catalyst

The majority of the design focus for SCRs is achieving uniformity of the flow through the system and particularly through the catalyst layers. The DeNOx reactions of the catalyst material are governed by the surface contact and residence time of the NOx and ammonia (NH3) molecules with the catalytic materials. Thus, the system will operate most effectively when there is a uniform distribution of velocity, NOx, and NH3 across the large volume of the catalyst. In addition, temperature distribution can be important. It plays a small role in the efficiency of the catalytic reactions, but more importantly the temperature profile must be uniform to avoid low temperature zones, which can result in the formation of solids (ammonium bisulfate for instance) that cause localized pluggage of catalyst.

Typically the uniformity is quantified in a statistical manner using one of the following criteria:

- Deviation from average
    - This is a simple variation from the average at a plane
    - Most frequently this is applied to temperature, where the focus is on avoiding extreme high and low conditions

    - For example “The temperature distribution at the catalyst at all unit loads should be within ±20 ^{o}C from the average temperature”
- Percent area deviation from average
    - Most often this reference is based on a percentage of the catalyst cross sectional area that is within a specified velocity tolerance
    - For example “100% of the flow area at the catalyst must be within ±15% of the mean velocity”
    - This implies that if one measures the velocity at many points across the face of the catalyst, the velocity should not vary by more than 15% at any of the measured points.
- %RMS or CV
    - This is a measure of flow uniformity based on the statistical standard deviation of a measured data sample
    - %RMS is referred to as “Percent RMS” or “Percent RMS deviation”
    - CV is the “Coefficient of Variation”
    - Both of these terms are used interchangeably by industry and are equal to the standard deviation divided by the average velocity at a given test plane
        - $\sum(v-v_{avg}) / n / v_{avg}$
            - v=velocity at each point
            - n= number of points
            - v_{avg} =average velocity over the test plane
    - For example “the ammonia and NOx should be well mixed such that the ammonia-to-NOx ratio at the catalyst inlet face is within 5%RMS at full load operation”

2.2 Ash Pluggage and Erosion

For a coal fired system, one must contend with the ash that passes through the boiler and to the catalyst. For “high dust” SCRs (located upstream of any particulate control equipment such as an ESP), there is a potential for both large particle ash (often called LPA) and for fine flyash to cause pluggage issues to the catalyst. Erosion can also be a concern in areas where high velocity exists and the erosive ash impacts either duct walls, internal elements, or the catalyst itself.

The precise mechanisms of LPA formation are not completely known to the industry. Some plants have no LPA at the boiler outlet, while others have tremendous quantities. The most likely scenario is that LPA forms in the furnace and backpass where ash fusion temperatures are high enough to allow agglomeration and formation of large particles. This can be exacerbated by poor combustion conditions, coal type, moisture conditions, and other parameters. Also, backpass pluggage and sootblowing can contribute to periodic LPA release into the flue stream. The hoppers at the exit of the boiler are intended to capture a majority of LPA (>500 micron in size) in order to protect the Air Heater and SCR, but oftentimes these hoppers do not capture enough LPA to protect a downstream catalyst.

It has been well documented in SCR experience in the USA that some form of LPA protection of the catalyst is required in the system design. A non-zero percent of the LPA exiting a boiler is larger in size than the catalyst pitch, and thus if allowed to pass through the system and to the catalyst, pluggage will ensue. This pluggage can occur rapidly depending on the operating conditions, and without LPA protection in the design, blockage and pressure drop across the SCR catalyst layers can increase quickly. Not only does this cause issues for the plant fan capacity, but ash can get packed into the catalyst, damaging it permanently in many instances.

Based on the author's experience, every SCR system should consider LPA protection of the catalyst. Most often this involves an LPA screen located at a favorable position in the upstream ductwork. Hoppers must be present below the screen to capture the deflected ash. The screen design itself must balance the considerations of size, material type, erosion resistance, pressure drop, pluggage potential, and maintenance.

In some cases it may be possible to capture LPA without a screen, but by using very carefully designed aerodynamic baffles. This can only be determined through flow modeling as described in later sections.

Based on experience with over 100 LPA studies, the author strongly recommends a combination of aerodynamic baffles and an LPA screen as the best approach for maximum catalyst protection and long life of the system.

Besides catalyst pluggage due to LPA, another design consideration is erosion due to fine flyash impacts. This is an issue for "high dust" and even for "low dust" SCRs (located downstream of an ESP for instance). Erosion is dependent on the ash hardness, mass loading, velocity of impact on a surface, and angle of impact on a surface. Thus, it is very important to examine the flow properties in detail in areas where ash may impact the catalyst, and also internal elements such as the LPA screen, turning vanes, trusses, mixers, and ammonia injection systems.

A further issue with SCR systems is deposition of flyash on horizontal and pluggage of the catalyst due to fly ash. An overall goal of the design should be to minimize the potential for dead flow regions, or low velocity zones, where flyash may accumulate. Most important is the area near the reactor inlet, where ash can build up and then avalanche onto the catalyst surface. The build up may occur due to low load operation, or in the flow recirculation zones or wakes during high load operation. Regardless of operating condition, the design should be aerodynamically streamlined in nature, with well-designed flow control devices (vanes, baffles, straighteners, mixers, baffles, etc.) that minimize the potential for fly ash accumulation at all unit loads. Cleaning equipment should be considered at the catalyst surface especially to aid in the operation. Ash sweepers, sonic horns, and sootblowers have been used successfully at plants to keep the catalyst as clean as feasible.

2.3 Typical Design Goals

Typical criteria for an SCR system are shown in Table 1. These are often negotiated and agreed upon by the system owner, SCR provider, catalyst supplier, LPA screen vendor, and flow modeler. The table provides a range of performance for "minimum" requirements and for "stringent" systems. Clearly, the more stringent the goal, the better the SCR will operate, but also the costs will generally be higher in terms of mechanical engineering, flow modeling, pressure drop, quantity/complexity of design features, etc. Like all engineering, tradeoffs exist between all these goals, and a careful balance of the objectives should be considered in order to focus priorities on the items of key importance.

Criteria	Target Value – Minimum	Target Value – Stringent
Velocity Distribution		
- Upstream LPA screen*	≤25 m/s peak velocity	≤18 m/s peak velocity
- Upstream ammonia lances	≤20% RMS	≤15% RMS
- Upstream 1st catalyst layer	70% of points within ±15% of mean	80% of points within ±10% of mean
- Upstream 1st catalyst layer	100% of points within ±25% of mean	100% of points within ±20% of mean
-Air Heater Inlet	≤20% RMS	≤15% RMS
-Air Heater Inlet	flow split < ±10% of theoretical share	flow split < ±5% of theoretical share
NH_3/NO_x Distribution		
- Upstream 1st catalyst layer	≤5% RMS	100% of points within ±5% of mean 95% of points within ±6.5% of mean
Temperature Distribution		
- Upstream 1st catalyst layer	within ± 17 °C of mean	within ± 11 °C of mean
Erosion and Pluggage		
- Peak velocity near walls and internal features*	Minimize peak velocities > 25 m/s Select appropriate abrasion resistant materials	Minimize peak velocities > 18 m/s Select appropriate abrasion resistant materials
- Upstream 1st catalyst layer*	60% of points within ≤10° from vertical	80% of points within ≤10° from vertical
- Upstream 1st catalyst layer*	80% of points within ≤15° from vertical	100% of points within ≤15° from vertical
- LPA capture	Maximize using LPA screen and baffles	Maximize using LPA screen and baffles
- Ash deposition on duct floor, vanes, trusses, reactor inlet, etc.	Minimize, provide appropriate cleaning devices	Minimize, provide appropriate cleaning devices
Other General Goals		
- Total Pressure Drop	minimize	minimize

Table 1. Typical SCR Design Criteria

3 Flow Modeling Methods

Fortunately, there are engineering techniques that allow for detailed analysis of all these design objectives and tradeoffs. Flow modeling, using both experimental and computational methods, is ideally suited to aid in SCR design. Laboratory scale modeling, referred to as Physical Flow Modeling, is often used to study aerodynamic principles. Computational Fluid Dynamics (CFD) uses advanced software and high speed computers to simulate flow, thermal mixing, species injection, and other fluid dynamic behavior.

The author has designed over 100 SCRs using both these modeling techniques, and recommends that both be utilized for coal-fired SCRs. Each modeling method has its strengths and shortcomings, and the combination of the two methods provides assurance in the final design. That said, both modeling methods are very complex, and if done incorrectly can result in poor designs that do not meet performance expectations.

Modeling should be conducted by fluid dynamics experts with SCR-specific experience. There are many possible errors in the modeling process if not performed properly. High quality laboratory modeling techniques are required for physical modeling or the results will not scale up properly to actual plant conditions. Likewise, the choice of correct CFD software, meshing approach, and modeling technique are critical for accurate results.

4 Physical Flow Modeling

4.1 Theory

The primary principal behind physical scale modeling is Fluid Dynamic Similarity. Lindeburg describes similarity well in *The Mechanical Engineering Reference Manual* [Professional Publications, 1995]:

> "Similarity between a model and a full-sized object implies that the model can be used to predict the performance of the full sized object. Such a model is said to be mechanically similar to the full-sized object. Complete mechanical similarity requires geometric and dynamic similarity. Geometric similarity means that the model is true to scale in length, area, and volume. Dynamic similarity means that the ratios of all types of forces are equal. These forces result from inertia, gravity, viscosity, elasticity, surface tension, and pressure."

To ensure the applicability of model test results to the full scale plant, geometric and dynamic similarity rules must be fulfilled during the test procedure. Geometric similarity is achieved by ensuring that the model is correctly scaled to the full scale dimensions. Dynamic similarity is achieved by making sure that the Reynolds number is of the same magnitude in both the model and the full scale plant. This dimensionless number is the ratio of inertial forces to viscous forces and is defined in Equation 1:

$$Re = (\rho * v * d) / \mu \qquad \text{(Equation 1)}$$

where

Re = Reynolds Number

ρ = fluid density

v = fluid velocity

d = characteristic length

μ = fluid viscosity

As long as the Reynolds number is large enough in every part of the cross-section to guarantee a completely turbulent flow, the flow characteristics are largely independent of the Reynolds number, so exact Reynolds number matching is not necessary [Gretta, W.J., and Grieco, G.J., *Consideration of Scale in Physical Modeling of Air Pollution Control Equipment*, International Joint Power Generation Conference, 1995]. Modelers of pollution control equipment have generally found that by matching the full scale velocity or velocity head, adequate fluid dynamic similarity is achieved, though attention must be paid to avoid erroneous results.

Other fluid dynamic relationships should also be taken into consideration to ensure proper modeling results and accurate scale-up of model results to the real-world situation. These include Prandtl Number (Pr), Schmidt Number (Sc), Barth Number (B), Bagnould Number (Ba), and Momentum Ratio (K).

4.2 Model Scale, Construction, and Flow Conditions

Typical model scales used for SCR physical modeling range from 1:8 to 1:16. Some smaller models are occasionally used, as small as 1:40 scale. For a majority of SCRs in the USA, Europe, and Asia, a scale of 1:10 to 1:12 is prevalent. This allows accurate representation of small details, such as ammonia injectors and mixers, while keeping costs and schedule reasonable.

The major components of the physical model exterior walls should be constructed of clear acrylic. Some of the internal flow control devices may be constructed of sheet metal, plastic, and/or other materials if they do not interfere with flow visualization experiments. The model will generally begin in the boiler backpass at the economizer. The model will include the SCR inlet/outlet ductwork, the reactor itself, and end at the air heater inlet or other downstream location.

For all modeled regions, internal flow control devices such as vanes, gas distribution devices, ammonia lances, mixers, LPA screen, and dampers should be included. Internal bracing and trusses are to be considered and included in the model where deemed critical to flow distribution, temperature, or pressure loss effects. Bracing and trusses not included in the model should be considered in pressure loss calculations. The catalyst layers will be represented with an appropriate material to provide the proper straightening effect. Perforated plate can be used to obtain the specified pressure drop per layer.

Construction tolerance should target accuracy within 1.5mm of the scaled design drawings to ensure geometric similarity is adhered to. Tighter tolerance are required for intricate details such as ammonia injection nozzles.

Detailed flow calculations should be performed to ensure fluid dynamic similarity. The model flow rate should be correctly scaled from the unit design flow rates to ensure the Reynolds Number regime is matched. The critical Reynolds Number for determining dynamic similitude in the turbulent regime is 3200. In general the model should target a Reynolds number above 8000 where the turbulent regime exists to provide a margin. This is to be maintained over the load range of SCR operation. Typical practice is for the model flow rate to be set such that either the full scale velocity or the full scale velocity head (dynamic pressure) is matched in a 1:1 ratio.

Typically, more than one unit load is analyzed in the modeling. Often, three distinct operating conditions are suggested (such as 100%, 75%, 50% or as specified by the plant based on operating experience).

4.3 Instrumentation and Data Collection

Accurate instrumentation should be utilized to collect velocity, pressure, gas species, and other model data. The probes are usually traversed within the model by hand, but automated, robotic probe actuation can be used for increased accuracy. It is more important to automate the data collection, however, in order to obtain accurate, repeatable, and technically defensible results. This involves a computerized data acquisition system with a high data sampling rate (500 Hz or more). This should apply to all instrumentation as discussed below.

For velocity, it is preferred to use hot wire anemometry versus pitot tube readings when the velocity is below 6 m/s, such as in the SCR reactor. Hot wire systems with 1-2% accuracy are readily available. For static pressures or differential pressures associated with pitot tubes, digital pressure transducers are recommended. These are readily available as well with 2% or better accuracy for the range of interest.

The physical model is run with ambient temperature air in the laboratory, and thus corrections for the actual SCR density must be made after measurements are completed in the laboratory. Modelers should ensure accurate calculation based on detailed recording of ambient weather conditions during the testing (barometric pressure, temperature, etc.). Between two points in the model, the pressure difference (dp) is given by the following equation:

$$dp_{model} = (P_{static,\,B} + ½ * \rho_{model} * v_B^2) – (P_{static,A} + ½ * \rho_{model} * v_A^2)$$

where

P_{static} = static pressure
v = fluid velocity
ρ = fluid density
$_A$ = location A
$_B$ = location B

In physical models, ammonia, NOx, and temperature tracking are generally simulated at ambient temperature by using a pressurized tracer gas. The tracer gas is injected into the model to represent molecules of NH3 or NOx, for instance, and the mixing behavior of the tracer gas mimics the mixing behavior of the actual gas species in the full scale. Certain fluid dynamic properties must be maintained such as the Schmidt and Prandtl numbers for accurate representation.

An accurate gas analyzer is then used to measure the tracer gas distribution within the model. Gas analyzers with better than 1% accuracy are readily available.

For the ammonia injection, the tracer gas is metered to the model through the injection lances in a very precise manner. The objective is to match the momentum ratio (injected species to freestream) of the actual lance operation. This is essential to obtain a fluid-dynamically accurate representation of the injection and subsequent mixing in the scale model. For NOx and temperature, generally the tracer gas is injected at the boiler outlet plane, to simulate non-uniform boiler exit conditions, often matching actual plant test data.

4.4 Dust Drop-out and Re-entrainment Testing

In laboratory modeling it is not wise to use actual plant flyash due to safety issues given the dangerous chemical composition of most flyash. Also, actual ash makes the model interior very dirty and affects future testing and visualization. Thus, most modeling is performed by using a surrogate dust particulate to represent actual flyash. A fine dust such as salt, cork, or sand is often used.

The procedure for Dust Drop-out Testing is to run the model at a low load condition and inject the dust at the model inlet. In low velocity zones, below the transport speed of the ash, the dust fallout patterns can thus be observed. These should be documented via photos, video, and depth measurements.

Dust Re-Entrainment testing can then be conducted where the model is turned off and loaded with a reasonable amount of model dust on all horizontal surfaces. This is intended to simulate plant operation at a low load for an extended period. The depth of dust to load can depend on plant operational history and duct velocity patterns; a depth of ~25mm is often used but can vary by plant. The model flow rate is then increased incrementally to 50%, 60%, etc. until 100% flow is established. The gradual re-entrainment of the ash into the gas stream should be observed and video recordings made. Any remaining dust when 100% load is reached should be considered a target for design optimization with further baffles/vanes/etc. There are some instances, however, where small amounts of remaining dust are not economical to avoid, another SCR design trade off to consider.

It is important to note that some amount of aerodynamic testing should be conducted to ensure that the model dust behaves similarly to the actual flyash. This can involve wind tunnel testing of the dusts and comparison of saltation and re-entrainment velocities.

4.5 Flow Visualization

Flow visualization provides a means to understand the flow profiles from a qualitative perspective. Most often a smoke generator with a point-source injection want is used to visualize flow within the system. Inserted string tufts or helium bubble generators can also be used. Flow visualizations should be performed at several unit loads to ensure proper aerodynamic flow exists. Videos and still photos should be taken as appropriate to document the flow characteristics.

5 Computational Fluid Dynamics (CFD) Modeling

5.1 Theory

CFD is a method of simulating fluid flow behavior via computer that has been used in industry since the mid-1970s. The three-dimensional geometry is divided into a number of control volumes, or cells. Models can contain millions of these cells depending on complexity of the geometry or the desired resolution of the flow field. The equations of fluid motion (conservation of mass, momentum, and energy) are solved for each of these control volumes using sophisticated software. The result is a prediction of the flow velocity, pressure, turbulence, temperature, and chemical species concentrations. Heat transfer effects may be calculated, so thermal mixing, chemical reaction, and other processes can be simulated.

Since the entire geometry can exist virtually within the computer, there is no geometric scaling. Also, actual flow conditions (temperature, density, viscosity, etc.) are implemented so matching of important flow parameters, such as Reynolds Number, is attained.

5.2 CFD Software

For SCR modeling, the author suggests using a well-proven CFD software package. There are many, many CFD packages in the world, but to the author's knowledge the top two solver packages used for SCRs over the past 10 years have been Azore™ and Fluent™. A third package, Star CCM+™, is also used to a lesser extent. All these software packages are fully three-dimensional and offer accurate calculation of velocities, pressures, conjugate heat transfer, and chemical species mixing. They are based on the control-volume solver technique and offer parallel processing for faster simulations.

5.3 Model Geometry and Boundary Conditions

The CFD models generally have the same domain as the physical model, and begin in the boiler backpass at the economizer. The model will include the SCR inlet/outlet ductwork, the reactor itself, and end at the air heater inlet or other downstream location. Like the physical model, internal flow control devices such as vanes, gas distribution devices, ammonia lances, mixers, LPA screen, and dampers should be included. Internal bracing and trusses are to be considered and included in the model where deemed critical to flow distribution, temperature, or pressure loss effects. The catalyst layers and air heater should be represented with an appropriate distributed flow resistance to provide the proper straightening effect and the specified pressure drop per layer. The LPA screen is generally modeled as a porous media, providing the proper flow resistance coefficients as specified by the screen supplier.

NOx and temperature profiles can be set at the model inlet location or at a plane where actual test data exists. The ammonia tracking is performed by using mass sources to inject ammonia as a gas species into the model at the appropriate location and velocity from the lances. In some cases, direct injection of liquid ammonia is used for SCRs. In this event, the CFD model can track the droplets from the ammonia atomization nozzle, calculate the liquid evaporation, then track the gaseous ammonia to the catalyst.

Typically, more than one unit load is analyzed in the modeling. Often, three distinct operating conditions are suggested (such as 100%, 75%, 50% or as specified by the plant based on operating experience).

5.4 CFD Mesh Requirements

The quality of the CFD mesh is a critical factor in determining the accuracy of the results. The CFD community knows quite well that mesh-dependent solutions can lead to incorrect design decisions. It is well established in the industry that the solution is degraded by “false diffusion” when the mesh topology is poor. The author’s experience is that hand-built meshes created by an experienced meshing expert are superior to automatically-generated tetrahedral meshes. The preferred mesh would be predominately flow-aligned, using a maximum quantity of hexagonal cells with some polyhedral cells filling the more complex portions of the domain. The benefit of a flow-aligned mesh topology is well documented as a means for minimizing solution degradation due to errors in gradient calculation or false diffusion typically associated with automated tetrahedral mesh topologies. By using the more advanced mesh topology, higher fidelity predictions of gas velocity, temperature, pressure, and ammonia/NOx distributions can be achieved with lower computational resources and shorter simulation run times.

A typical 600-800 MW SCR model should have a high cell count in order to resolve all the flow details properly. A typical length scale might be on the order of 75-100mm, with further detail in areas of key focus. This may include the ammonia injection zone, where finer mesh on the order of 25mm or smaller may be necessary, through small channels such as the flow rectifier, or near the main areas of interest. In general, with today’s computers and software, the author would expect a CFD model to be on the order of 20,000,000 to 40,000,000 computational cells in order to offer the latest predictions.

5.5 Results Analysis

Results of CFD models are presented in a number of ways. Color contour plots are most typical and provide a quantitative depiction of velocities, pressures, and ammonia mixing within the model. Usually these contours are shown on two-dimensional sections within the three-dimensional model. Flow streamlines can also be plotted to depict pathlines through the domain. These are very helpful in visualizing the flow directionality, though actual flow vectors can also be plotted to show exact features.

A key aspect of CFD is that literally millions of data points are available. The potential for data review is thus orders of magnitude beyond the physical model. At a typical catalyst inlet plane, for instance, the CFD model will examine over 1000 data points. This compares to perhaps 30-60 traverse points in a physical model.

Another key output of the CFD simulation are the uniformity statistics such as %RMS deviation or the percentage of data points that lie within a certain tolerance of the average, as noted in Section 2 above.

These results are compared to the project goals and allow the CFD engineer to iteratively improve the design to meet the goals, and assess the tradeoffs between competing goals.

5.6 Flow Visualization

In CFD, flow visualization can be created as animated movies of flow streamlines or data planes. Like smoke flow in a physical model, these can be informative from a qualitative standpoint to understand the flow characteristics and look for areas that require optimization.

6 Example SCR Design Project

The GETS 2016 conference presentation will include actual CFD and physical model results for an 800 MW coal-fired power plant design project. The system includes ammonia injection, static mixers for ammonia, NOx, and temperature, LPA screen, and velocity distribution optimization.

7 Summary

The performance of an SCR system is highly dependent on the quality and uniformity of the flow through the catalyst. Using both physical flow modeling and CFD modeling, an SCR design can be optimized to maximize the DeNOx rates while also minimizing ammonia consumption and pressure drop. In addition, important performance factors such as LPA carryover, catalyst pluggage, erosion potential, and temperature stratification can be analyzed and optimized using the modeling tools.

Best practices for design include a very detailed analysis using both physical and CFD modeling. The experience of modeling personnel should be a key consideration to ensure quality results. Automated data acquisition should be used with highly-accurate instrumentation for lab testing. Well proven CFD software such as Azore^{TM}, Fluent^{TM}, or Star CCM+^{TM} should be used. Extreme care should be taken during the meshing stage to ensure proper cell distribution, quantity, topology, and flow alignment. This affects results significantly. Flow modeling of SCRs is an engineering specialty that can make or break an SCR design.