Numerical Study of Industrial Scaled RFCC Cyclones Efficiency and Performance

Tommy Firmansyah, Mohammad Abdur Rakib, Syed Basheer, Hala Kelani, and Mohamed Al Musharfy
Outline of Presentation

1. RFCC Main Process and Key Equipment
2. Key Parameters for RFCC Optimum Performance
3. Importance of Computational Models and Roles of Computational Fluid Dynamics (CFD)
4. Use of Zoom-In Method
5. CFD Analysis Process
6. Results and Validation
7. Future Work
8. Summary
RFCC MAIN PROCESS
Key Parameters for Optimum Performance of RFCC

- Feed Composition
- Residence Time
- Hydrocarbon Partial Pressure
- Temperature
- Catalyst-to-Oil (CTO) Ratio
- Catalyst Properties
- Riser Hydrodynamics

Optimization
Solution Approaches

- Theoretical/Analytical
- Fluid Flow & Heat Transfer Problems
- Experimental
- Numerical/Computational
## Approaches: Pros and Cons

<table>
<thead>
<tr>
<th>Approach</th>
<th>Advantages</th>
<th>Disadvantages</th>
</tr>
</thead>
<tbody>
<tr>
<td>Experimental</td>
<td>Capable of being most realistic</td>
<td>Equipment required, scaling problems, measurement difficulty, expensive operating cost</td>
</tr>
<tr>
<td>Theoretical/Analytical</td>
<td>General information in formula form</td>
<td>Restricted to simple geometry and physics, usually linear problems</td>
</tr>
<tr>
<td>Numerical/Computational</td>
<td>Actual scale/ flow domain, details of flow (in space and time), cheaper than experiment</td>
<td>Numerical errors, computer costs, boundary conditions</td>
</tr>
</tbody>
</table>

Numerical/Computational model can never replace experiment (measurements), but the amount of experiment and the overall cost can be significantly reduced.
What is CFD?

Computational Fluid Dynamics (CFD)

“CFD is the analysis of systems involving fluid flow, heat transfer and associated phenomena, such as chemical reactions by means of computer-based simulation”

Versteeg & Malalasekera, “An Introduction to Computational Fluid Dynamics: The Finite Volume Method”
Complements experimental and theoretical fluid dynamics by providing an alternative cost effective means of simulating real flows.

**Insight**

High spatial resolution of the computed variables.

⇒ Better visualization of flow-fields and engineering parameters thus facilitating enhanced understanding of designs.

**Foresight**

Testing many variations until you arrive at an optimal result before physical prototyping and testing. Practically unlimited level of detail of results at minimum added expense.
Include all relevant phenomena
- Effects of inlets, outlets, internals and other geometric details captured
- Scale independent
- Qualitative and quantitative information
- Insights on off-design performance
RFCC Modeling Using CFD

Reactor Cyclones

Riser Separation System (RSS) Model
Why Zoom-In method is required?
- Reduce modeling complexities
- Reduce computational time

Divide the concerned RFCC section into smaller components

“Zoom-In” Components of RFCC Reactor Cyclones:
- Single Reactor Cyclone
- Dual Reactor Cyclones (Inner and Outer Rings)
- Inlet Distributor and Reactor Plenum
## Use of Zoom-In Method

<table>
<thead>
<tr>
<th>ZOOM-IN 1</th>
<th>ZOOM-IN 2</th>
<th>ZOOM-IN 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single Reactor Cyclone</td>
<td>Dual Reactor Cyclones (Inner and Outer Rings) + Inlet Ducts</td>
<td>Reactor Plenum and Upstream Inlet Ducts</td>
</tr>
</tbody>
</table>

![Single Reactor Cyclone](image1)

![Dual Reactor Cyclones](image2)

![Reactor Plenum and Upstream Inlet Ducts](image3)
CFD Analysis Process

Pre-processing
- Problem formulation
- Mathematical model
- Geometry and Meshing

Processing
- Solution method
- Iteration

Post-processing
- Results analysis
- Visualization

CFD Analysis Process Diagram
- Define geometry of the domain and operating conditions.
- Define type of flow:
  - Turbulent.
  - Multiphase (gas-solid particles) flow.
- Define objectives of the CFD analysis to be performed:
  - Detailed flow pattern.
  - Cyclone efficiency (catalyst loss evaluation).
Typical Cyclone Flow

Discretization Process

- Discretize the computational domain into a number of control volumes, i.e. **Meshing**
  - Structured Hexahedral Cells
  - Mesh generation tool (ANSYS Mesh)
  - Mesh size/resolution: ~ 600,000 cells for single cyclone
- Discretization Method:
  - Finite Volume based Software (ANSYS Fluent)
  - High order approximations (QUICK method)
CFD Solver Set-Up and Simulations

- Multiphase flow:
  - Eulerian model for gas phase
  - Lagrangian model (Discrete Phase Model) for catalyst particles

- Turbulence model:
  - Seven equation RSM (Reynolds Stress Model)
  - Suitable for streamline curvature, swirl, rotation and high strain rates, e.g. cyclone flow
Results Visualization

Contours of Pressure and Velocity

<table>
<thead>
<tr>
<th>Pressure (Pa)</th>
<th>Velocity Magnitude (m/s)</th>
<th>Axial Velocity (m/s)</th>
<th>Tangential Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.00e+03</td>
<td>5.00e+01</td>
<td>1.00e+01</td>
<td>6.00e+01</td>
</tr>
<tr>
<td>7.40e+03</td>
<td>4.75e+01</td>
<td>9.00e+00</td>
<td>5.65e+01</td>
</tr>
<tr>
<td>6.80e+03</td>
<td>4.50e+01</td>
<td>8.00e+00</td>
<td>5.30e+01</td>
</tr>
<tr>
<td>6.20e+03</td>
<td>4.25e+01</td>
<td>7.00e+00</td>
<td>5.00e+01</td>
</tr>
<tr>
<td>5.60e+03</td>
<td>4.00e+01</td>
<td>6.00e+00</td>
<td>4.60e+01</td>
</tr>
<tr>
<td>5.00e+03</td>
<td>3.75e+01</td>
<td>5.00e+00</td>
<td>4.25e+01</td>
</tr>
<tr>
<td>4.40e+03</td>
<td>3.50e+01</td>
<td>4.00e+00</td>
<td>3.90e+01</td>
</tr>
<tr>
<td>3.80e+03</td>
<td>3.25e+01</td>
<td>3.00e+00</td>
<td>3.55e+01</td>
</tr>
<tr>
<td>3.20e+03</td>
<td>3.00e+00</td>
<td>2.00e+00</td>
<td>3.20e+01</td>
</tr>
<tr>
<td>2.60e+03</td>
<td>2.75e+00</td>
<td>1.00e+00</td>
<td>2.85e+01</td>
</tr>
<tr>
<td>2.00e+03</td>
<td>2.50e+01</td>
<td>0.00e+00</td>
<td>2.50e+01</td>
</tr>
<tr>
<td>1.40e+03</td>
<td>2.25e+01</td>
<td>-1.00e+00</td>
<td>2.15e+01</td>
</tr>
<tr>
<td>8.00e+02</td>
<td>2.00e+00</td>
<td>-2.00e+00</td>
<td>1.80e+01</td>
</tr>
<tr>
<td>2.00e+02</td>
<td>1.75e+00</td>
<td>-3.00e+00</td>
<td>1.45e+01</td>
</tr>
<tr>
<td>-4.00e+02</td>
<td>1.50e+00</td>
<td>-4.00e+00</td>
<td>1.10e+01</td>
</tr>
<tr>
<td>-1.00e+03</td>
<td>1.25e+00</td>
<td>-5.00e+00</td>
<td>7.50e+00</td>
</tr>
<tr>
<td>-1.60e+03</td>
<td>1.00e+00</td>
<td>-6.00e+00</td>
<td>4.00e+00</td>
</tr>
<tr>
<td>-2.20e+03</td>
<td>7.50e+00</td>
<td>-7.00e+00</td>
<td>5.00e+01</td>
</tr>
<tr>
<td>-2.80e+03</td>
<td>5.00e+00</td>
<td>-8.00e+00</td>
<td>-3.00e+00</td>
</tr>
<tr>
<td>-3.40e+03</td>
<td>2.50e+00</td>
<td>-9.00e+00</td>
<td>-6.50e+00</td>
</tr>
<tr>
<td>-4.00e+03</td>
<td>0.00e+00</td>
<td>-1.00e+01</td>
<td>-1.00e+01</td>
</tr>
</tbody>
</table>
Results Visualization

Velocity Vectors

Post-Processing
2D PLOT – Axial Velocity

- $y = -0.5 \text{ m}$
- $y = 0.0 \text{ m}$
- $y = 0.5 \text{ m}$
- $y = 1.0 \text{ m}$

Axial Velocity (m/s)

Position (m)

Post-Processing

We Refine Right

ANSYS Fluent Release 16.2 (3d, pbns, RSM)
2D PLOT – Tangential Velocity

Post-Processing

Tangential Velocity (m/s)

Position (m)

Tangential Velocity

Post-Processing

We Refine Right

TAKREER

شركة أبوبكر لتصنيع النفط

Nov 08, 2016

ANSYS Fluent Release 16.2 (3d, pbns, RSM)
Animation

Post-Processing
## Resulting Pressure Drop

<p>| | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Design Pressure Drop (Pa)</strong></td>
<td>8238</td>
</tr>
<tr>
<td><strong>Predicted Pressure Drop (Pa)</strong></td>
<td>8308</td>
</tr>
<tr>
<td><strong>% Difference</strong></td>
<td>1.0</td>
</tr>
</tbody>
</table>

### Post-Processing

![Pressure (Pa)](image-url)
# Resulting Cyclone Efficiency

<table>
<thead>
<tr>
<th>CFD Model</th>
<th>Predicted Catalyst Loss (kg/hr)</th>
<th>Predicted Cyclone Efficiency (%)</th>
<th>Design Efficiency (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Single Cyclone</td>
<td>1.19</td>
<td>99.995</td>
<td></td>
</tr>
<tr>
<td>Dual Inner Cyclone 1</td>
<td>1.18</td>
<td>99.995</td>
<td></td>
</tr>
<tr>
<td>Dual Inner Cyclone 2</td>
<td>1.68</td>
<td>99.993</td>
<td>99.987</td>
</tr>
<tr>
<td>Dual Outer Cyclone 1</td>
<td>0.50</td>
<td>99.998</td>
<td></td>
</tr>
<tr>
<td>Dual Outer Cyclone 2</td>
<td>2.86</td>
<td>99.989</td>
<td></td>
</tr>
</tbody>
</table>

### CFD Model Comparisons

- **Single Cyclone**
- **Dual Inner Cyclones**
- **Dual Outer Cyclones**

---

**Graphs:**

1. **Single Cyclone**
2. **Dual Inner Cyclones**
3. **Dual Outer Cyclones**
Resulting Inlet Distribution Flow

Standard deviation from averaged flow rate

Streamlines colored by velocity
Future Work

Reactor Cyclones

Riser Separation System (RSS) Model

Animation
Takreer Research Centre (TRC) has been actively engaged in research and development activities in RFCC modeling primarily to support refineries in troubleshooting its operational problems and upgrading its process performance.

A reliable Computational Fluid Dynamics (CFD) model to study efficiency and performance of an industrial scaled RFCC reactor cyclone has been developed and validated.

The CFD model will be improved and refined further using the results of RSS model.
The authors gratefully acknowledge Abu Dhabi Oil Refining Company (TAKREER) and TAKREER Research Centre (TRC) management for their support and guidance.
Thank you