The Practical Uses of Computational Fluid Dynamics – Not Just a Pretty Picture

Presenter: William Osley
Company: CALGAVIN Ltd
Email: william.osley@calgavin.com
Contents:

• Introduction

• **Case Study 1:** Air Cooled Heat Exchanger (ACHE)
  Problems related to bypass and flow distribution

• **Case Study 2:** Shell and Tube Heat Exchanger
  Maldistribution

• **Case Study 3:** Research and Development

• **Case Study 4:** Tube-side Flow stratification

• **Case Study 5:** Temperature Pinch

• Conclusion
Introduction

- Software used:
  - CFD: ANSYS CFX
  - Geometry: ANSYS DesignModeler

- Heat Transfer: Heating and Cooling Investigated

- Reynolds range: Laminar and Turbulent

- Turbulence Model: $k$-$\epsilon$ (when needed)
Case Study 1: Air Cooled Heat Exchanger (ACHE)
Problems related to bypass and flow distribution
Why use Computational Fluid Dynamics to Investigate Air coolers?

- Air coolers are designed using empirical correlations that use assumptions such as:
  - all the liquid entering the header subsequently flows through tubes
  - perfect air distribution over the bundle

- When built, the mechanical design and build quality / tolerances can have a profound effect on such assumptions

- CFD can be used to investigate those shortcomings and the effects on performance
Bypass Problem Description

- User of lube oil Air Cooled Heat Exchanger reports significant underperformance
- Measured 50% less pressure drop than design calculations
- Lower than expected tube side pressure indicates bypass around tube bundle
- Possible causes:
  - Vent hole in partition plate
  - Missing / broken welds between partition plate and header walls
Air Cooled Heat Exchanger Geometry

1. No Bypass, 2. 12 mm vent hole and 3. 12 mm vent hole and side gaps
Verification of CFD simulations

- By comparing the no bypass geometry with: tube side pressure drop (nozzles, header and tubes) results with heat exchanger design software

- CFD simulation results within 8% of calculated pressure drop from heat exchanger design software
No Bypass

12mm vent hole

12mm vent hole and 1 mm gaps

Inlet Temperature: 86 °C
Outlet Temperature: 59.0 °C

Outlet Temperature: 62.1 °C

Outlet Temperature: 65.9 °C
Results

- 12 mm vent hole = 20% mass flow bypass
  - 35% reduction in pressure drop
  - 11% reduction in duty

- 12 mm vent hole and 1 mm gaps = 42% mass flow bypass
  - 64% reduction in pressure drop
  - 27% reduction in duty
Simulation Scenarios

- API 661 Recommended 40% fan coverage

- Equations used to describe fan air flow
- Same total mass-flow for each scenario
- Three plenum depths for the three fan layout:
  - 500mm
  - 720mm
  - 1000mm
Verification and Results

• CFD results compared with Equation:

\[ \Delta P_A = \left( 2 \times f_b \times Nr \times (\rho \times V_{\text{max}})^2 \right) / \rho \]

Equation commonly used to calculated cross flow air pressure drop though a tube bundle (Serth and Lestian (2014))

• Eq 2 gives \( \Delta P_A = 86.4 \text{ Pa} \)
• CFD gives \( \Delta P_A = 88.4 \text{ Pa} \)
• CFD model accurately predicts the air flow

Effect of Increased Plenum Depth

- 500mm Plenum Depth
- 720mm Plenum Depth
- 1000mm Plenum Depth

Increase in Plenum Depth

More even Velocity Distribution
Effects of Maldistribution on Heat Transfer

Calculated Duty = 7.5 kW
Velocity Profile half way across domain

Calculated Duty = 7.3 kW  2.7% reduction
Velocity Profile half way across domain

Air IN Even Distribution Ave 3.5 m/s
3 inlets at 1.5 m/s and 2 (2x area) at 5 m/s
Conclusions

- Care should be taken in sizing vent holes and pass partition welds to avoid bypass to ensure correct performance
- Increased Plenum depth improved distribution
Case Study 2:
Shell and Tube Heat Exchanger
Maldistribution
Service:
• Heat recovery for hydro treatment reactor

Problem description:
• Calculated performance should be 60% higher
• No spare capacity of fired heater to increase throughput

<table>
<thead>
<tr>
<th>Shells:</th>
<th>TEMA: AES</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>3 in series; 2 in parallel;</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Bundle:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tubes:</td>
</tr>
<tr>
<td>2521; 1pass</td>
</tr>
<tr>
<td>20mm x 1.8mm x 9m</td>
</tr>
</tbody>
</table>

Calculated Exchanger Performance

<table>
<thead>
<tr>
<th>Tube side dp calc / allow.</th>
<th>2.5kPa / 45 kPa</th>
</tr>
</thead>
<tbody>
<tr>
<td>Shell side HTC</td>
<td>900 W/m²K</td>
</tr>
<tr>
<td>Tube side HTC</td>
<td>285 W/m²K</td>
</tr>
<tr>
<td>Duty</td>
<td>Measured 20 MW / real +60%</td>
</tr>
</tbody>
</table>
CFD Simulation of Bundle Maldistribution

Expected severe fluid maldistribution in the bundle on tube side
• Tube side pressure drop of 2.5 kPa, this is very low. 85% of which is in the nozzles (allowable tube side pressure drop 45 kPa!)

• Axial Tube side nozzles contribute to maldistribution

Higher tube side pressure drop would be beneficial
CFD Simulation of Bundle Maldistribution

side view  

plane
CFD Simulation of Bundle Maldistribution

<table>
<thead>
<tr>
<th></th>
<th>before (empty)</th>
<th>after (hiTRAN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tube pressure drop</td>
<td>2.5 kPa (&gt;85% nozzles)</td>
<td>20 kPa (~10% nozzles)</td>
</tr>
</tbody>
</table>

Plain empty

hiTRAN

© Copyright Cal Gavin 2010 www.calgavin.com
## hiTRAN installation and benefits

<table>
<thead>
<tr>
<th></th>
<th>before (empty)</th>
<th>after (hiTRAN)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tube pressure drop</td>
<td>25mbar (&gt;85% nozzles)</td>
<td>200mbar (~10% nozzles)</td>
</tr>
<tr>
<td>Tube side heat transfer</td>
<td>&lt;285 W/m2K</td>
<td>~980W/m2K</td>
</tr>
<tr>
<td>Shell feed outlet temp</td>
<td>240 °C</td>
<td>314 °C</td>
</tr>
<tr>
<td>Tube effluent outlet</td>
<td>115</td>
<td>82</td>
</tr>
<tr>
<td>Mass flow</td>
<td>27kg/sec</td>
<td>42kg/sec</td>
</tr>
<tr>
<td>Load on fired heater</td>
<td>4.2MW</td>
<td>2MW</td>
</tr>
</tbody>
</table>

Annual energy savings of $ 233000
Case Study 3: Research and Development
Fluid movement, cooling
Re 253, 70 °C inlet and 7 °C wall
CFD Simulation Plain empty tube

Simulation verified with experimental data for different Reynolds numbers
70 °C Inlet temperature; 7 °C Wall temperature, 2.5m tube length; Viscosity 12cP

Reynolds 253

70 °C Inlet

Outlet bulk
CFD 62 °C; measured 61.8 °C

Reynolds 935

70 °C Inlet

Outlet bulk
CFD 66.2 °C; measured 66.2 °C

- Stratified flow
- Long residence time
  at bottom of tube
- Low heat transfer
Verifying CFD Simulation results with experiments

Pr; 1150; LMTD: 50°C ; tube Inlet 15°C ; Shell 70°C
Dye Stream hiTRAN
Verifying CFD Simulation results with Cal Gavin heat transfer measurements for hiTRAN

<table>
<thead>
<tr>
<th>Reynolds number</th>
<th>Outlet Temperature (°C)</th>
<th>CFD</th>
<th>% dev</th>
</tr>
</thead>
<tbody>
<tr>
<td>190</td>
<td>60.98</td>
<td>60.61</td>
<td>0.6</td>
</tr>
<tr>
<td>496</td>
<td>62.08</td>
<td>61.94</td>
<td>0.2</td>
</tr>
<tr>
<td>1014</td>
<td>62.71</td>
<td>62.61</td>
<td>0.16</td>
</tr>
<tr>
<td>1993</td>
<td>63.28</td>
<td>63.17</td>
<td>0.17</td>
</tr>
</tbody>
</table>

65°C INLET Temperature, 40°C Wall temperature, 1000mm test section
Fluid movement hiTRAN

Re 253, 70 °C Inlet and 7 °C Wall

Velocity profile at outlet
CFD Simulation Plain empty tube compared to enhance hiTRAN flow

Example Simulation verified with experimental data:
70 °C Inlet temperature; 7 °C Wall temperature
2.5m tube length; Reynolds number 253; mass flow 195kg/hr; Viscosity 12cP

plain tube

70 °C Inlet

Outlet bulk
CFD 62 °C; measured 61.8 °C

hiTRAN tube

70 °C Inlet

Outlet bulk
CFD 50.7 °C; measured 49.9 °C

- Stratified flow
- Long residence time at bottom of tube
- Low heat transfer

- Good fluid distribution
- High heat transfer with low outlet temperature

Velocity profile plain

Velocity profile hiTRAN
Flow Stratification

Tube ID: 22 mm, Tube Length: 2500mm, Reynolds number 190, Inlet 65°C Wall 40°C

Empty Tube: Temperature Profile

hiTRAN: Low density

Outlet: 60.7°C

Outlet: 55.9°C

Highest velocity in centre of tube

Highest velocity towards tube wall
Residence time Distribution

Tracer at tube outlet, plain empty

Tracer at tube outlet, hiTRAN

Residence Time Distribution in laminar flow Reynolds 250; 2.5m tube length

Tracer injection inlet
tracer empty tube flow, outlet
tracer hiTRAN flow outlet
Static mixer Heat Transfer – Heating Experimental and CFD comparison

Fluid used: Glycerol

Viscosity: 350 cP at ~35 °C

Reynolds number Range: laminar 1 to 28

Inlet Temperature: ~30 °C

Wall Temperature: ~64 °C
Comparison of Experimental and CFD results
Static Mixer: Re 16, Inlet 30 °C and Wall 60 °C

hiTRAN: Re 14, Inlet 30 °C and Wall 51 °C
Case Study 4: Tube-side Flow stratification
Goal of Revamp is to increase polymer outlet temperature

AEL 4 pass, 372 tubes
25.4mm x 1.65mm x 4000mm

condensing steam
9.2bar
176 °C
12000W/m²K

38.2 °C

Viscous polymer
~2000cP / inlet
~800cP / outlet

47600kg/hr
~115W/m²K

101 °C / dp 2.7bar
<table>
<thead>
<tr>
<th></th>
<th>Plain 9.2bar</th>
<th>hiTRAN 6.3bar</th>
</tr>
</thead>
<tbody>
<tr>
<td>No of passes</td>
<td>4</td>
<td>2</td>
</tr>
<tr>
<td>Steam pressure [bar]</td>
<td>9.2</td>
<td>6.3</td>
</tr>
<tr>
<td>Steam temp. [C]</td>
<td>176</td>
<td>160</td>
</tr>
<tr>
<td>tube side HTC [W/m²K]</td>
<td>100</td>
<td>206</td>
</tr>
<tr>
<td>Tube side outlet [C]</td>
<td>101</td>
<td>124</td>
</tr>
<tr>
<td>Tube side dp [bar]</td>
<td>2.7</td>
<td>2.9</td>
</tr>
</tbody>
</table>
Mixed Convection causes flow stratification

hiTRAN
Temperature distribution; tube Inlet:

Plain empty

Temperature distribution; middle of tube

Temperature distribution; tube exit
Case Study 5: Temperature Pinch
190 kg/sec Wet crude ~ 1000 cP

83 kg/sec 103 °C Produced water ~ 15 MW

60 °C

65 °C

85 °C

Heat transfer

<table>
<thead>
<tr>
<th>Plain design</th>
<th>Plain design</th>
</tr>
</thead>
<tbody>
<tr>
<td>Tube side / Reynolds ~ 1800</td>
<td>400 W/m²K</td>
</tr>
<tr>
<td>Shell side</td>
<td>300 W/m²K</td>
</tr>
<tr>
<td>Overall</td>
<td>140 W/m²K</td>
</tr>
</tbody>
</table>

EMTD ~ 9 °C

<table>
<thead>
<tr>
<th>plain</th>
<th>plain</th>
</tr>
</thead>
<tbody>
<tr>
<td>No of shells [-]</td>
<td>2 parallel</td>
</tr>
<tr>
<td>Total tubes [-]</td>
<td>10348 x 12.8m long</td>
</tr>
<tr>
<td>Total area [m²]</td>
<td>7821</td>
</tr>
</tbody>
</table>
In tube temperature pinch in conventional design

HTRI warning message on plain tube design:

In tube temperature pinch predicted in tubepass 1: The localized temperature pinch could effectively nullify up to 55.6% of the tubeside heat transfer surface area in this tube pass.

Empty tube

hiTRAN

CFD simulation over 10.2m tube length with: water inlet 103 °C
In tube temperature pinch in conventional design

Water outlet temp: 74 °C
ΔT on tube cross-section ~10 °C

Empty tube

Water outlet temp: 69 °C
ΔT on tube cross-section ~2 °C

hiTRAN

No in-tube pinch
Conclusion

• This presentation has shown a variety of uses for CFD they include:
  ➢ Identification for the cause of an air cooled heat exchanger underperformance
  ➢ Investigation ACHE air-side flow distribution
  ➢ Shell and tube tube-side maldistribution
  ➢ Identification of flow stratification and temperature pinch
  ➢ Research and development

• There are many more possibilities to explore using CFD:
  ➢ New heat transfer enhancement geometries
  ➢ Turbulence flows
  ➢ 2-phase flow
CALGAVIN Limited, UK
Specialist Heat Exchange Engineers

• What we do?
  ➢ Provide thermal engineering solutions to:
  ➢ Optimize plant production
  ➢ Solve production limitation problems
  ➢ Reduce energy costs
  ➢ Enhancement technology (hiTRAN)
CALGAVIN: Solving Problems, Saving Costs

• **Study to revamp operations** - Providing consultancy advice through project engineering to improve plant operations.

• **Design Services** - Enhancing heat exchangers using various software such as HTRI, Aspentech and hiTRAN SP.

• **Analytical engineering services** - Analysing the performance and operation of existing heat exchangers, making comparisons between original designs and enhanced designs for improved efficiency.
Any Questions?

Email: william.osley@calgavin.com
Tel: +44 1789 400401
Fax: +44 1789 400411