IMPROVING ENERGY EFFICIENCY IN CEMENT PLANTS

USING

COMPUTATIONAL FLUID DYNAMICS (CFD)


Date: 23rd June 2015

Jodhpur, Rajasthan
ABOUT US

Established in 1983, Mechwell is a multi-discipline applied R&D oriented engineering firm.

Mechwell was established to provide solutions for Cement, Power & Allied industries for flow, emissions & energy consumption problems.

Business:

- Turnkey Solutions to Cement, Power & Allied Industries including custom engineering, specialty systems design (EFIP), manufacturing and implementation, and support to industry in emissions control to achieve SPM<30mg, heat and mass transfer, and applied energy systems.

- Design, Manufacture & Supply of Expansion Joints, Dampers & Gates and EFIP’s

- CFD & FEA Consultancy Services

Mechwell’s team consists of the engineering experts from IIT & Other reputed engineering institutes having vast experience.
SOFTWARE FACILITIES

Mechwell has the latest Computer Aided Engineering (CAE) systems to help you create superior products more effectively than ever before.

3D Modeling  -  ANSYS Design Modeler, Solid Works
Meshing  -  ICEM CFD, Gambit, HyperMesh.
CFD Analysis  -  ANSYS Fluent, ANSYS CFX, AcuSolve, OpenFoam
FEA Analysis  -  ANSYS Mechanical, Radios.
Boiler Software  -  Boiler Expert

The company has adequate resources to press into service additional machine power, man power & additional software licenses required then and there.
Computer-aided flow simulation (CFD) is a highly recommendable technology to use in the process optimization. It is especially an alternative to costly, conventional development processes, consisting of constructive development, prototype construction and experimental validation (in several cycles).

CFD (Computational Fluid Dynamic, CFD), is a new technology with the emergence of high-speed electronic computer. The technology allows the numerical calculation method for solving fluid flow problems as possible, in solving practical engineering flow, heat transfer problem in practice has been continuously mature and complete, and has an increasing number of engineering practices verification.
CFD APPLICATION AREAS IN CEMENT PLANTS

- Cyclone
- Ducts
- Electrostatic Precipitators (ESP)
- Baghouse
- Raw Mill/ Coal Mill
- Kiln/ Calciner
- Gas Conditioning Tower
GENERAL PROBLEMS FACED BY CEMENT INDUSTRIES

**PH system**: Improper Heat Transfer, High Pressure Drop, Improper Material Distribution, Material Accumulation

**Cyclone**: Low Collection Efficiency, Improper material distribution, Improper Heat Transfer

**Ducts**: High Pressure Drop, Erosion

**Raw Mill, Cement Mill**: Erosion, High Pressure Drop

**ESP, Bag House**: Improper flow distribution, Low Collection Efficiency

**Gas Conditioning Tower**: High Pressure Drop, Improper Heat Transfer

**Fan**: High Power Consumption, Lesser Efficiency

**Kiln**: Castable Erosion, Flame Propogation, Improper Combustion

**Calciner**: Improper Combustion
COMPUTATIONAL FLUID DYNAMICS : APPROACH

- Identification of problem
- Data Collection
- Modelling
- Mesh Generation
- Boundary Condition
- Evaluation
- Modification/Analysis
- Simulation
- Verification
SUCCESS STORY

COMPUTATIONAL FLUID DYNAMIC ANALYSIS OF RAW MILL CIRCUIT
DATA BY CLIENT:

- Detailed Drawings
- Operating parameters like temp., Flow rates, Pressure etc.

6/30/2015
MECHWELL INDUSTRIES LTD. | Driven By Innovation
DETAILED DRAWING:
CAD MODEL OF RAW MILL CIRCUIT
MESHING : GRID FOR RAW MILL CIRCUIT

➤ Types of Meshing :
  • Hexa Mesh
  • Tetra Mesh
  • Structured Mesh
  • Unstructured Mesh
CFD RESULTS OF EXISTING RAW MILL CIRCUIT
CFD RESULTS OF EXISTING RAW MILL CIRCUIT
### CONCLUSION:

<table>
<thead>
<tr>
<th>Area of Work</th>
<th>Existing</th>
<th>Modified</th>
</tr>
</thead>
<tbody>
<tr>
<td>Rawmill Outlet Duct</td>
<td>Not Uniform flow due to immediate Bend</td>
<td>Guide Vanes Provided &amp; flow optimised</td>
</tr>
<tr>
<td>Cyclone Inlets</td>
<td>Flow not uniform due to ____</td>
<td>Flow distribution improved due to modification in</td>
</tr>
<tr>
<td>Cyclone Outlets</td>
<td>Flow distribution was not uniform, highly turbulent &amp; swirling</td>
<td>Flow distribution is optimized &amp; sufficient reduction in pressure drop is achieved.</td>
</tr>
<tr>
<td>CA fan inlet</td>
<td>Non Uniform flow distribution</td>
<td>Flow distribution is made uniform</td>
</tr>
<tr>
<td>Pressure drop</td>
<td>153 mmWC</td>
<td>Pressure drop reduced by 42mmWC(&gt;25%) than existing system.</td>
</tr>
</tbody>
</table>
Reasons for High PH Exit Temperature

- Improper material distribution
- Improper flow distribution
- High velocity

Heat Transfer Analysis in Riser Duct for Flue gas with Particles elaborates the Temperature
Modification in Spreader box & Feed Pipe for Heat transfer Improvements

So by CFD Modification, PH Exit Temperature can be reduced from 5 Deg C to 25 Deg C
CFD STUDY FOR PH SYSTEM TO REDUCE PRESSURE DROP

Case Study for JK Cements, Nimbahera
CASE STUDY JK CEMENT WORKS, NIMBAHERA PH SYSTEM

Benefits After CFD Modification & Implementations

<table>
<thead>
<tr>
<th>Reduction in Pressure Drop</th>
<th>Power Saving</th>
</tr>
</thead>
<tbody>
<tr>
<td>60 mm WC</td>
<td>More than 100 KW</td>
</tr>
</tbody>
</table>

Streamline Plot in PH system
MODIFICATION TO PH SYSTEM TO AVOID MATERIAL ACCUMULATIONS

Case Study for Bheema Cements
MODIFICATION TO PH SYSTEM TO AVOID MATERIAL ACCUMULATIONS

Inlet to 6th cyclone

CFD results of Material Deposition study in PH System
From CFD results (Multiphase flow analysis), it can be observe that for 75 micron & Above, not a single particle is escaping through cyclone, means Cyclone have 100% efficiency.
### PARTICLE TRACKING STUDY AND IMPROVEMENT IN PH SYSTEM

**Efficiency Comparison of Testing Data (Provided) & CFD Results to 1st Stage Cyclone**

<table>
<thead>
<tr>
<th>Sr No.</th>
<th></th>
<th>10 micron</th>
<th>30 micron</th>
<th>52 micron</th>
<th>75 micron</th>
<th>104 micron</th>
<th>209 micron</th>
<th>Increase in Pressure Drop</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>% Retain (given Data by Client)</td>
<td>17.82</td>
<td>16.38</td>
<td>10.72</td>
<td>27.63</td>
<td>20.02</td>
<td>1.05</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Exist Efficiency (From % Retain Data) (%)</td>
<td>6.38</td>
<td>24.2</td>
<td>40.58</td>
<td>51.3</td>
<td>78.93</td>
<td>100</td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>Exist Efficiency (From CFD) (%)</td>
<td>6.0</td>
<td>17.7</td>
<td>28.7</td>
<td>96.6</td>
<td>100.0</td>
<td>100.0</td>
<td></td>
</tr>
<tr>
<td>4</td>
<td>MOD1_Deep Tube dimension change</td>
<td>7.2</td>
<td>35.3</td>
<td>90.9</td>
<td>100.0</td>
<td>100.0</td>
<td>100.0</td>
<td>12 mmWC</td>
</tr>
<tr>
<td>5</td>
<td>MOD2_Deep Tube dimension change</td>
<td>6.24</td>
<td>45.74</td>
<td>98.11</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>15 mmWC</td>
</tr>
<tr>
<td>6</td>
<td>MOD3_Deep Tube dimension change</td>
<td>8</td>
<td>54.43</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>19 mmWC</td>
</tr>
<tr>
<td>7</td>
<td>Mod 4_Entry Duct dimension change</td>
<td>14.4</td>
<td>23.5</td>
<td>100.0</td>
<td>100.0</td>
<td>100.0</td>
<td>100.0</td>
<td>12 mmWC</td>
</tr>
<tr>
<td>8</td>
<td>Mod 5_same as MOD4 with bottom Surface dimension change</td>
<td>22.0</td>
<td>21.0</td>
<td>99.0</td>
<td>100.0</td>
<td>100.0</td>
<td>100.0</td>
<td>13.5 mmWC</td>
</tr>
<tr>
<td>9</td>
<td>Mod 6_same as MOD5 with Top Surface dimension change</td>
<td>28.53</td>
<td>21.11</td>
<td>93.87</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>12.5 mmWC</td>
</tr>
<tr>
<td>10</td>
<td>Mod 7_MOD1+MOD5</td>
<td>13.825</td>
<td>33.73</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>12.0 mmWC</td>
</tr>
<tr>
<td>11</td>
<td>Mod 8_MOD1+MOD6</td>
<td>36.47</td>
<td>17.85</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>100</td>
<td>18.9 mmWC</td>
</tr>
</tbody>
</table>
CFD STUDY TO PREDICT THE EROSION PATTERN

Case Study for Saurashtra Cements

Cooler ESP Inlet Duct
CFD STUDY TO PREDICT THE EROSION PATTERN

Max. Erosion rate of the duct is 3 mm/year and CFD predicted duct erosion surface area is higher.

Existing Model

Max. Erosion rate of the duct is 1 mm/year at few area and erosion area is has optimized.

Modified Model
CFD STUDY TO PREDICT THE EROSION PATTERN

Max. Erosion rate of the duct is 3 mm/year and CFD predicted duct erosion surface area is higher.

Max. Erosion rate of the duct is 1 mm/year at few area and erosion area is has optimized.
Minimum pressure drop reduction of 12 mmWc

Payback period less than 5 months
CFD STUDY OF BAG HOUSE OUTLET DUCT

Case Study for Ramco Cements
CFD STUDY OF BAG HOUSE OUTLET DUCT

Minimum pressure drop reduction of 20 mmWc
Payback period less than 4 months
CFD STUDY FOR FAN OUTLET DUCT OF CEMENT MILL 2

Case Study for Ramco Cements
CFD STUDY FOR FAN OUTLET DUCT OF CEMENT MILL 2

Flow separation
CFD STUDY FOR ELECTROSTATIC PRECIPITATOR (ESP)
CFD STUDY FOR ELECTROSTATIC PRECIPITATOR (ESP)
Computational Model of Bag house
CFD STUDY OF BAG HOUSE

(Premodified Baghouse)

(Modified Baghouse)

Flow Distribution Study
CFD STUDY OF BAG HOUSE

(Premodified Baghouse)  (Modified Baghouse)

Flow Distribution Study
## BENEFITS ACHIEVED BY CFD:

<table>
<thead>
<tr>
<th>Sr.No</th>
<th>Project</th>
<th>Benefits</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>Thermal Efficiency Improvement in PH system – JK Cement</td>
<td>Improved Production, Ph system collection efficiency improved</td>
</tr>
<tr>
<td>2.</td>
<td>Thermal Efficiency Improvement in PH system – Sagar Cement</td>
<td>100 KWh Saving &amp; Heat Rate Improvement by 5 deg celcius</td>
</tr>
<tr>
<td>3.</td>
<td>Duct Design for Maihar Cement</td>
<td>50 KWh Saving</td>
</tr>
<tr>
<td>4.</td>
<td>Deflector Design in Down comer For Dalmia Cement Tamilnadu</td>
<td>30 KWh Saving</td>
</tr>
<tr>
<td>5.</td>
<td>Duct Design in Raw Mill- Shree Cement</td>
<td>Pressure drop reduction by 18 mmWC</td>
</tr>
</tbody>
</table>
Combustion analysis using CFD allows:

- to improve the combustion air flow pattern to the burners in order to smooth the combustion.
- to compare the influence of different burner designs on flame shape.
- to understand the influence of several inlet parameters concerning either the fuel used, or the kiln geometrical dimensions, the position of the burner in the kiln, the various velocities.
Thermal Efficiency Improvement in Calciner

CFD analysis in the Calciner

- Distributions of Velocities, Temperature
- Concentrations of the reactant products and trajectories of coal particles
- Particle interaction with the gases can be predicted
- Particle residence time

All the above factors are assessed for CFD analysis in a Calciner
By CFD Analysis in the Kiln Hood & Kiln,

- Secondary/Tertiary air Splits
- Distributions of Velocities, Temperatures
- Flow through the Hood
- Size & shape of the flame in the kiln
- Erosion pattern in the kiln Hood
Benefits by CFD analysis in the Kiln,

- Cold CFD Simulation: to predict the flow pattern & Improvement in flow for combustion
- CFD simulations in combustion: To predict the Flame length propagation
- Reduction in Kiln hood Erosion
These Customers Trusts US:

Cement Plants

Thermal Power Stations

Others

6/30/2015
MECHWELL INDUSTRIES LTD. | Driven By Innovation
R&D – CENTER FOR INNOVATION

Contact Details: Mr. Akshay Shah
Engineer - Marketing

Email id: akshay@mechwell.org; nasik@mechwell.com

Landline: 0253-2453556