Towards on-line simulation of fluidized bed combustors

Liekkipäivä, Lappeenranta, 26 Jan 2012

Sirpa Kallio, Timo Niemi, Juho Peltola, Veikko Taivassalo, Matti Tähtinen (VTT)
Alf Hermanson, Henrik Saxén (ÅA)
Markus Honkanen, Pentti Saarenrinne (TUT)
Tommi Kärkkäinen (JyU)
Overview

The presentation summarizes a TEKES project:

**CFD based on-line process analysis – applied to circulating and bubbling fluidized bed processes**

and shows results from the first year of research.

Contents:

- Project partners and research goals
- Background and plans
- Activities in 2011
- Time-scale and dispersion analysis of transient simulations
- Time-averaged simulation of Chalmesr boiler
- Fast Particle Size Distribution Modelling in CFB simulations
- Laboratory scale experiments at Åbo Akademi University
- Summary and outlook for 2012
Project partners and research goals

- Project duration: 2011-2013
- **Research partners:**
  - VTT Technical Research Centre of Finland, Tampere University of Technology, Åbo Akademi University, Jyväskylä University
- **International co-operation:**
  - Chalmers University of Technology, Tsinghua University
- **Industrial support:**
  - Fortum, Metso Power Oy, Numerola Oy, Etelä-Savon Energia Oy, Saarijärven Kaukolämpö Oy

The goal is to **develop a real-time process analysis method**, which:

- Opens up a window to the fluidized bed to **provide run time information** on the phenomena occurring inside the process
- Gives **information on possible unwanted process conditions** including changes in particle size distribution and bed behaviour
Background and plans: Fast CFD simulation of CFBs

- **TAveCFD_CFB TEKES project**: 
  - A fast CFD simulation approach for CFBs
  - Tested then only in small scale and simplified conditions, later extended to combustion and larger scales.

- The new approach should allow performing the simulations online with a **response time of 1-2 hours to process changes**.

- Extention and validation of the method for industrial conditions still required.

- Other online analysis tools should be combined with the CFD approach to get a fuller picture of the real-time process conditions in a FB combustor.
Background and plans: Current project

Combination of measurements and modeling to evaluate hydrodynamics of BFBs and CFBs
- The previously developed CFD models for fluidization processes are validated and extended.

Real-time diagnostic methods are developed for fluidized bed availability problems.
- Pressure and acceleration measurements

Information obtained with signal processing methods is utilized as part of the on-line CFD approach.
- Coupling of online process measurements, signal processing and fast response CFD simulations.
Activities in 2011 include:

- A new BFB cold model was built at Åbo Akademi.
- CFB and BFB cold model experiments were carried out at ÅA using image-based and high frequency pressure measurements.
- A measurement campaign in a 135 MWe CFB boiler in China in co-operation with Tsinghua University of Technology.
- Time-averaged validation simulation of the Chalmers CFB boiler.
- Evaluation of methods to include particle size distribution in the simulations.
  - Dense DPM simulation approach was tested in BFB and CFB simulations.
  - Novel implementation of a CFD size distribution model.
- Time scale and dispersion analysis of existing transient simulation results, basis of the model implemented in the time-averaged CFD model.
Turbulent dispersion coefficient is a product of velocity fluctuation standard deviation and Lagrangian length scale

\[ D_{T,i} = v_{\sigma,i} L_{L,i} \quad v_{\sigma,i} = \sqrt{v'_i v'_i} \]

The length scale can be calculated from Lagrangian time-scale

\[ L_{L,i} = v_{\sigma,i} \tau_{L,i} \quad \tau_{L,i} = \int_{0}^{\infty} \frac{v'_i(\tau)v'_i(\tau-t)}{v^2_{\sigma,i}} dt \]

If the time-scale is approximated with Eulerian time-scale, it is possible to calculate the dispersion coefficient from transient simulation data from a single monitoring point.

\[ D_{T,i} \approx u^2_{\sigma,i} \tau_{E,i} \]
Time scale and dispersion analysis of transient CFD simulations

- Strong anisotropy and location dependence, wide wall regions.
- Vertical time-scales are similar in large and small scale. The small CFB corresponds to the bottom 3 m of the large CFB.
- In the large CFB, the horizontal time-scales are larger than in lab scale by up to an order of magnitude.
Time-averaged CFD modeling: submodels

Balance equations are solved for:

1. Gas and solids momentum
   • Requires closures for drag, laminar and turbulent stresses, Reynolds stresses, pressure fluctuation and solids pressure terms.

2. Phase continuity

3. Gas phase local scale turbulence: dispersed $k$-$\varepsilon$ model

4. Solid phase Reynolds stresses
   • Gas phase stresses are calculated with algebraic correlations from the solid phase stresses
   • Fluctuation time scales are obtained from algebraic correlations

5. Energy: Specific enthalphy equations for both phases

6. Species equations for gas components
   • Closure for mass diffusion, reaction rates
Time-averaged CFD modeling: Chalmers boiler
(Taivassalo et al., FBC21, Italy 2012)

- Based on Åmand et al. (1997)
  - Coal 2: 8 MW, Polish coal.
- Flue gas recirculation with the primary air.
- Only the riser section and short outlet and solids return sections are in the computational domain.
- Recirculation of fuel and solids is modelled with boundary conditions.

### Table 1. Gas inflows.

<table>
<thead>
<tr>
<th></th>
<th>Primary</th>
<th>Secondary</th>
<th>Solids recirculation</th>
<th>Ash classifier</th>
<th>Fuel feed</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mass flow</td>
<td>2.66 kg/s</td>
<td>1.24 kg/s</td>
<td>0.079 kg/s</td>
<td>0.15 kg/s</td>
<td>0.018 kg/s</td>
</tr>
<tr>
<td>Temperature</td>
<td>100 °C</td>
<td>100 °C</td>
<td>775 °C</td>
<td>791 °C</td>
<td>100 °C</td>
</tr>
</tbody>
</table>

### Table 2. Inlet gas compositions.

<table>
<thead>
<tr>
<th></th>
<th>O2 [%wt]</th>
<th>CO2 [%wt]</th>
<th>H2O [%wt]</th>
</tr>
</thead>
<tbody>
<tr>
<td>Primary</td>
<td>18.6</td>
<td>4.92</td>
<td>1.36</td>
</tr>
<tr>
<td>All Other</td>
<td>23.3</td>
<td>0</td>
<td>0</td>
</tr>
</tbody>
</table>

### Table 3. Chemical composition of the fuel

<table>
<thead>
<tr>
<th></th>
<th>Volatiles [wt% daf]</th>
<th>Proximate analysis [wt% a.r.]</th>
<th>Ultimate analysis [wt% daf]</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Comb.</td>
<td>Ash</td>
<td>Moist.</td>
</tr>
<tr>
<td></td>
<td>40.2</td>
<td>74.2</td>
<td>8.9</td>
</tr>
</tbody>
</table>
Preliminary results: Volume fraction and mass weighted velocities

- Typical CFB velocity and volume fraction fields.
- Volume fraction is unrealistically low at the very top.

Figure 2. a) Time averaged solid volume fraction. b) and c) Time-averaged mass weighted vertical velocities for the gas and solid phases.
Preliminary results: Fuel particles, volatile release rate and char content

- A significant portion of fuel recirculates: returned immediately in the simulation.
- Fuel particle residence times up to one minute on one pass, or even longer for the largest particles.
- Most of the particles remain at the bottom, as evidenced char and volatiles release patterns.

Figure 3. a) and b): Two typical pathlines for a 1 mm fuel particle coloured by the residence time (a) and the particle diameter (b). c) Volatiles release rate (kg/m^3s). d) Char content (kg/m^3s).
Preliminary results: Comparison to the experiment

<table>
<thead>
<tr>
<th></th>
<th>Åmand et al.</th>
<th>Simulated</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solids circulation</td>
<td>29 kg/s</td>
<td>32 kg/s</td>
</tr>
<tr>
<td>Char content at bottom</td>
<td>2.4-4.3 %</td>
<td>0.3 – 2.0 %</td>
</tr>
<tr>
<td>Char content at outlet</td>
<td>0.5 %</td>
<td></td>
</tr>
<tr>
<td>Char content in cyclone leg</td>
<td>1.0 %</td>
<td></td>
</tr>
</tbody>
</table>

![Graph showing pressure drop vs. height](image1)

![Graph showing oxygen concentration vs. height](image2)
Fast Particle Size Distribution (PSD) Modelling in CFB simulations

- **Goal:** Implement a fast method to include a PSD in CFD simulation of fluidized beds

- **Method:** Based on the Direct Quadrature Method of Moments (DQMOM)
  - PSD approximated with delta functions, usually 2 or 3
  - Positions and amplitudes tracked with scalar transport equations
  - Each particle size class can have unique velocity

- **Typically** velocities are obtained by solving momentum balance for Eulerian phases => computationally slow

- In the new implementation velocities are approximated algebraically from local force balances
  - Only one set of momentum equations for the dispersed phase => relatively fast
Fast Particle Size Distribution Modelling in CFB simulations

- **Current status:** Implementation ready for 2D-cases without mass transfer mechanisms. Validation in process.

Particle Size Distribution and corresponding delta functions
Åbo Akademi Lab scale experiments 2011

Two cold models: CFB: 0.4 m x 3.0 m
BFB: 0.9 m x 1.95 m

The provide good optical access, allowing visualization and detailed hydrodynamic measurements.
Åbo Akademi Lab scale experiments 2011

Co-operation between VTT, Åbo Akademi and Tampere University of Technology

Image-based measurements:

- Simultaneous measurement of instantaneous:
  - Particle velocity: Particle Image Velocimetry (PIV), Particle Tracking Velocimetry (PTV)
  - Solids volume fraction: Absorbtion of light, correlations of image Greyscale value
  - Particle size distribution: Greyscale gradient direction matching with a circular mask

- Effects of particle size distribution? What are the Eulerian and Lagrangian time and lengths scales in different portions of the riser?
Åbo Akademi Lab scale experiments 2011

Co-operation between VTT, Åbo Akademi, University of Jyväskylä and Tampere University of Technology

Pressure measurements:

- High sampling frequency pressure measurements:
  - How to measure fluctuation characteristics in an industrial scale furnace?
  - How measuring point influences the pressure signal? How different measurement points correlate?

- Testing in lab-scale:
  - Effects of a change in particle size distribution or agglomeration in the pressure signal in CFB and BFB.
  - Can we predict changes in fluidization?

- Synchronous visualization, high-speed PIV and two high sampling frequency pressure measurements:
  - What is it that we see in the pressure signal? Can we use it to measure velocity?
Velocity measurement by correlating signals of two high frequency pressure probes?

High-speed PIV and visualization to
- help with interpretation of high speed pressure signal.
- to validate results of pressure signal analysis.

- Imaging frequency 250 Hz
- Imaging area 0.4 m x 0.4 m
- Pressure sampling frequency 5000 Hz
Summary and Outlook

- A real-time CFD-based analysis tool for CFB and BFB boilers is being developed.
- Improvements in the description of mixing and particle size distribution have been done.
- Validation simulation of a small boiler produced acceptable results.
- Extention and validation of the method for industrial conditions still required, in addition to further speed-up of the simulations.
- Other online analysis tools should be combined with the CFD approach to get a fuller picture of the real-time process conditions in a FB combustor.
- Real-time diagnostic methods based on analysis of pressure fluctuations are developed to detect fluidization conditions and to contribute to the online analysis.
VTT creates business from technology