Upgrading a Biomass Furnace: A Simulation Driven Approach

By D. Hasevoets, Cofely Fabricom E&E & Wouter Remmerie, Patrick Vlieger, Voxdale

Cofely Fabricom GDF SUEZ is a leading multi-disciplined engineering, project management and construction organization in the oil, gas, power and allied industries. Part of the global energy group GDF SUEZ, Cofey Fabricom employ over 5,500 people worldwide who strive for the improvement of energy efficiency, durable development and quality of life.

The Cofely Fabricom, Energy & Environment Division specializes in turnkey Engineering, Procurement, Construction and Commissioning (EPC) projects in the renewable energy sector emphasizing on balance-of-plants, flue gas treatments, and cogeneration plants.
In April 2013, Cofely Fabricom E&E was awarded a turnkey revamping contract to improve the reliability and performance of the biomass furnace of the cogeneration unit in Sart-Tilman, Liège, Belgium, which is part of the central heat production and distribution across the Université de Liège (ULG) and the CHU University Hospital. The project was centered around the improvement of a number of failures experienced by the two institutions, specifically:

- Insufficient instrumentation for proper combustion control;
- Reduced performance due to a lack of recirculation of smoke gases in nominal operation;
- Incomplete coverage of grid with pellets, leading to primary air bypass non-uniform combustion; and
- Refractory damage caused by extremely high local temperatures.

In order to address these issues, a completely new combustion control philosophy was investigated.

The approach would involve:

- A mixture of 50% re-circulated smoke gases into primary and secondary air to improve performance;
- Optimizing, by means of Computational Fluid Dynamics (CFD) calculations, secondary air injections to respect legal emission limits;
- Installing new fans, frequency drives and electrical cabinets; and
- Implementing superior instrumentation, including ultrasonic flow meters and infrared cameras.

Cofely Fabricom E&E first performed the necessary process calculations for optimized combustion, accounting for flow rate, temperature, oxygen content and pressure drop, which sized the primary, secondary and recirculation fans as well as the duct diameter and valves.

The next step was to ensure the optimal configuration of the secondary air injections. Since the main objective of the project was to achieve an increase in performance, the oxygen concentration at the furnace outlet had to be reduced from its current value of 13% while respecting the legal emission limits. The secondary air injection therefore had to ensure that all produced CO was recombined to CO_2. This would be achieved by obtaining an optimal level of turbulence inside the narrowest section of the furnace, referred to as the "furnace neck".

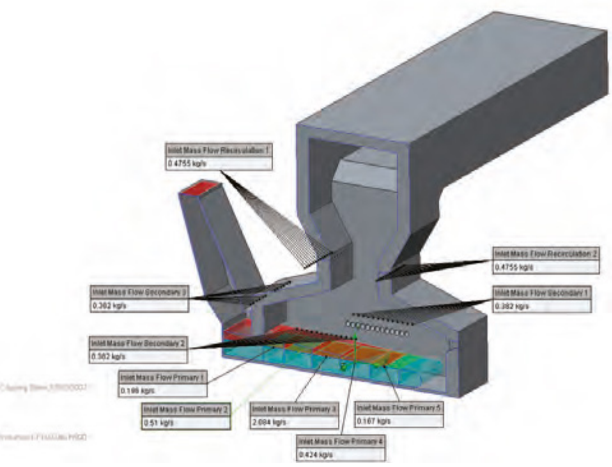


Figure 1. FloEFD simulation geometry

To gain sufficiently high velocities at the tube outlet, the secondary air injection nozzles were divided into two rows: an upper and lower ramp of 2 x 14 tubes each at the front and back end of the furnace, 112 tubes in total. At lower flow rates, the two lower rows were isolated in order to maintain a minimum velocity of 20m/s at the remaining tube outlets.

Physically building and testing different prototypes of an installation of this size would be time consuming and prohibitively expensive. In order to understand the response of the system and arrive at the optimum configuration of nozzles, Computational Fluid Dynamics (CFD) simulations were carried out using Mentor Graphics' FloEFD by project partners Voxdale.

The first step was to create a model of the existing furnace and simulate its behaviors. This approach would allow simulation parameters and boundary conditions to be adjusted to obtain a good match between a well-known system and the simulation model. During this phase confidence in the simulation approach is established and the first insights into the system are generated.

Combustion processes weren't directly modeled in the simulation, instead, air temperatures and flow rates were adjusted to obtain comparable behavior. This reduced the time required for each simulation without appreciably affecting accuracy, which in turn allowed for CFD to cover a large number of design iterations in a reasonable time.
Figure 1 illustrates the original setup of the furnace with wood pellets that are fed into the system, forming a layer on the moving grid. Below

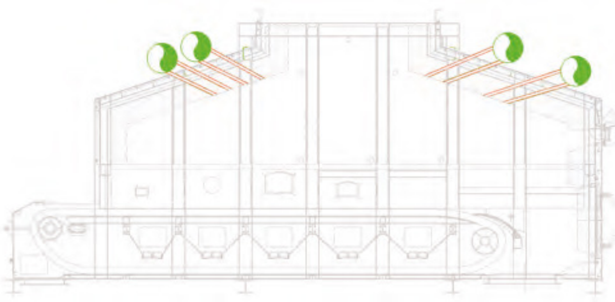


Figure 2. Secondary air nozzle detail results

this grid, five distinct supplies of primary air, provide the oxygen required for the combustion process. In addition, secondary air is supplied through nozzles on both the front and lateral walls. At the furnace neck, exhaust gases are injected into the furnace, providing the system with recirculation air.

In the second phase, different design proposals were set up and simulated in FloEFD. Three-dimensional turbulence plots and velocity patterns were used to assess the various configurations. One of the earlier setups is illustrated in Figure 2, in which the secondary air nozzles were moved closer to the furnace neck and the number of nozzles was greatly increased. Although this solution offered some benefits, it resulted in insufficient turbulence intensity across the furnace neck.

Based on the insight offered by such simulation and analysis, further design iterations could intelligently evolve. For example, it became apparent that offsetting the recirculation nozzles slightly with respect to each other created the desired amount of turbulence in the furnace neck.
The positioning and layout of the secondary air jets were similarly optimized in order to ensure sufficient turbulence in the adiabatic chamber (Figure 3). As the simulations were performed without combustion, it could reasonably be assumed that an even higher level of turbulence, higher velocities and better mixing would occur in practice. The resulting design is shown in Figure 4.

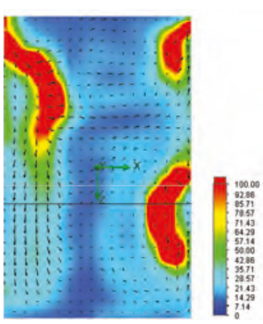


Figure 3. Turbulence intensity in the adiabatic chamber

Disassembly of the old furnace began in May 2013, with installation works running from June to September 2013. Start-up and combustion optimization took place in October - November 2013. During cold commissioning of the new installation the simulated velocity distribution was verified inside the furnace and comparable flow patterns were observed.

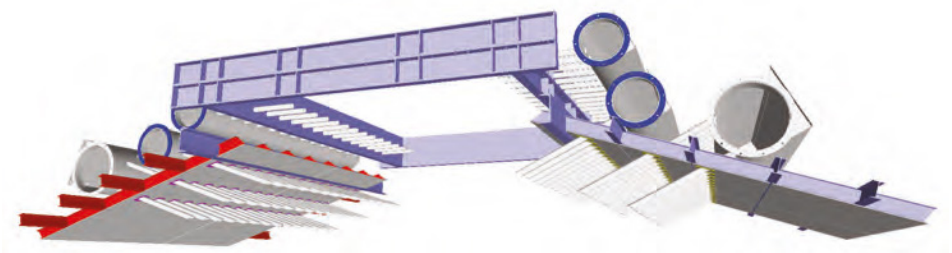
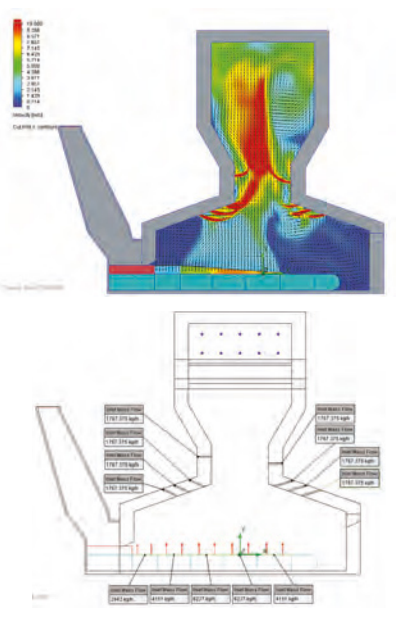


Figure 4. Furnace neck resulting design

Integrating CFD in to the design process helped Cofely Fabricom E&E deliver the project successfully and within a short time frame. Virtually prototyping various designs in FloEFD provided a reliable and inexpensive means by which different parameters could be assessed and adjusted. Ultimately, all project objectives have been fulfilled.

- Performance was increased by 8.4% (actual efficiency 89.3%);
- The furnace represents a considerable improvement over the previous design (see Table 1) and operates well below legal limits, e.g. for both carbon monoxide and nitrous oxides.

The process followed demonstrated the value of fluid simulation for such projects. It is therefore an approach which will be used for future projects, where appropriate. The installation has been in reliable operation since November 2013.



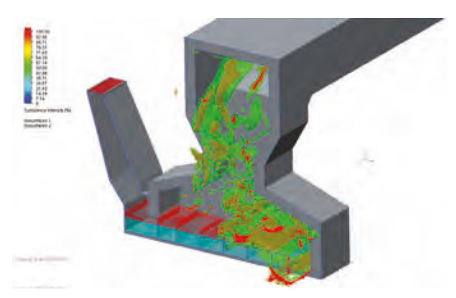


Figure 5. FloEFD visualizations

Table 1 Emission figures

Species	Old Furnace (Nm³)	New Design (Nm³)
CO	106	11
NO_x	406	203