FLOWNEX®
SIMULATION ENVIRONMENT

CASE STUDY

EVALUATING BOILER WALL CIRCULATION USING A SYSTEM-LEVEL FLOWNEX MODEL

This case study demonstrates how Flownex was used to model flow distribution within the wall-tube circuits of a John Thompson MicroGen boiler, providing insight into circulation behaviour across a burning furnace.

GENERAL INDUSTRIES

CHALLENGE:

To accurately predict flow distribution in a system that experiences boiling in a large network of vertical tubes.

BENEFITS:

Flownex allowed the user to:

- Quickly deliver reliable results with little effort compared to conventional 3D CFD analysis.
- Develop a model that accounts for the hydraulic losses associated with the phase changes that occurs during boiling.
- Develop a model that can serve as a digital asset to investigate the resulting flow distribution for different scenarios like non-uniform heat loads for example.

SOLUTION:

To combine detailed tube-level heat transfer models with a system-level circulation network to predict flow distribution throughout the boiler.

THE JOHN THOMPSON MICROGEN® BOILER

The John Thompson MicroGen® is a modular watertube boiler developed for biomass and coal-fired cogeneration applications. A single boiler can produce between 20 and 25 t/h of steam, while higher steam outputs can be achieved by combining multiple units. The boiler is designed to generate steam at pressures between 31 and 61 bar, allowing it to be paired with commonly available steam turbines that offer good cycle efficiency at reasonable cost.

The MicroGen can use a wide range of fuels, including coal, bagasse, wood waste, wood chips, sawdust, agricultural residues and other biomass products, making it suitable for a broad range of industrial applications.



Figure 1: typical MicroGen installation (Source: John Thompson MicroGen® Biomass-/Coal-fired Boiler brochure.)

MODELLING THE BOILER CIRCULATION NETWORK

To evaluate flow distribution within the MicroGen boiler, a detailed circulation model was developed in Flownex. The objective was to predict how flow is distributed throughout the furnace wall circuits under a specified heat load.

Accurately predicting this flow distribution requires more than a simple hydraulic analysis. As water absorbs heat and begins to evaporate, the fluid properties and flow resistance within the tubes change significantly. These changes directly influence how flow is distributed between the parallel circuits, making it necessary to account for both fluid flow and heat transfer within the model.

Since the amount of evaporation depends on the heat transferred to the fluid, the heat transfer process must also be represented in sufficient detail. The simulation therefore captures both the thermal and hydraulic behaviour of the boiler wall circuits, allowing the interaction between heat absorption and flow distribution to be evaluated simultaneously.

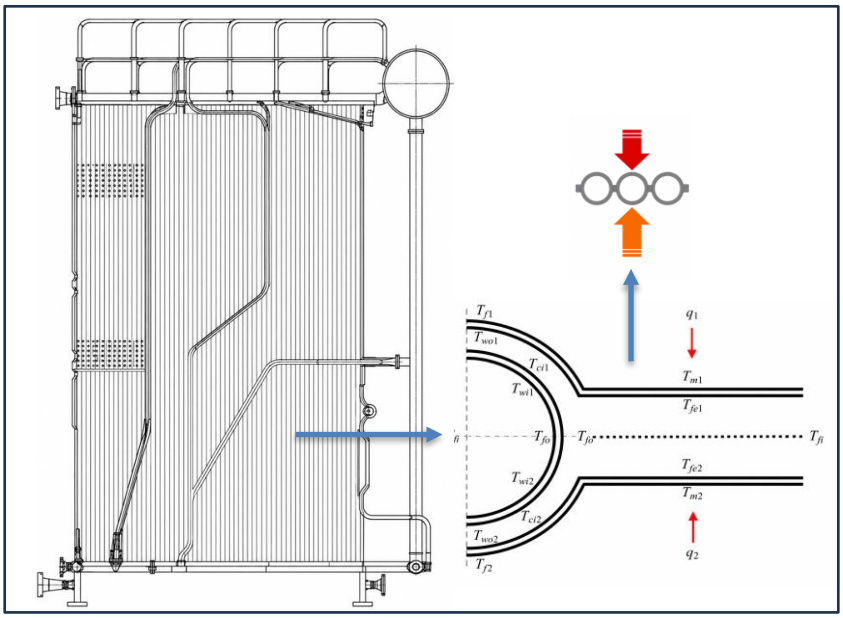

Figure 2: Boiler Wall Geometry and Representative Membrane Wall Segment

Figure 2 shows a front view of the furnace wall and a top view segment of a membrane wall made up of a tube-and-fin geometry. The repeating membrane wall segment will be modelled as a building block that can be replicated for parallel tube sections.

FLOWNEX MODEL OF THE MEMBRANE WALL SEGMENT

The model shown in Figure 3, represents a single vertical tube and fin segment of the boiler wall. Like building blocks, multiple segments will be connected in parallel to represent the complete boiler wall geometry. These tube models are then incorporated into the larger circulation network used for the system-level analysis.

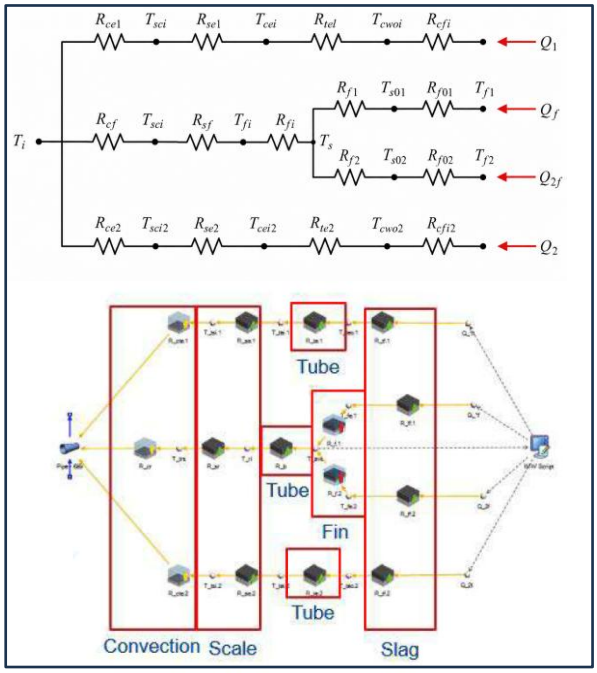

Figure 3: Flownex model of the membrane wall segment

As shown in Figure 3, the model explicitly accounts for thermal resistances associated with slag deposits, scale formation, tube material and fin material. These influence the rate at which heat reaches the working fluid and therefore have a direct impact on evaporation and flow distribution.

A prescribed heat load is applied through a script-driven boundary condition, representing the thermal energy supplied by the furnace. This heat load is transferred through multiple conduction and convection layers in the thermal resistance network before it is ultimately absorbed into the fluid flowing through each tube segment.

FLOW DISTRIBUTION RESULTS

Once the detailed tube models were assembled into the full circulation network, the overall flow distribution within the whole boiler circulation network could be evaluated. The detailed tube models contribute the correct hydraulic losses and thermal effects, while the larger network determines how flow is shared between the various parallel circulation paths.

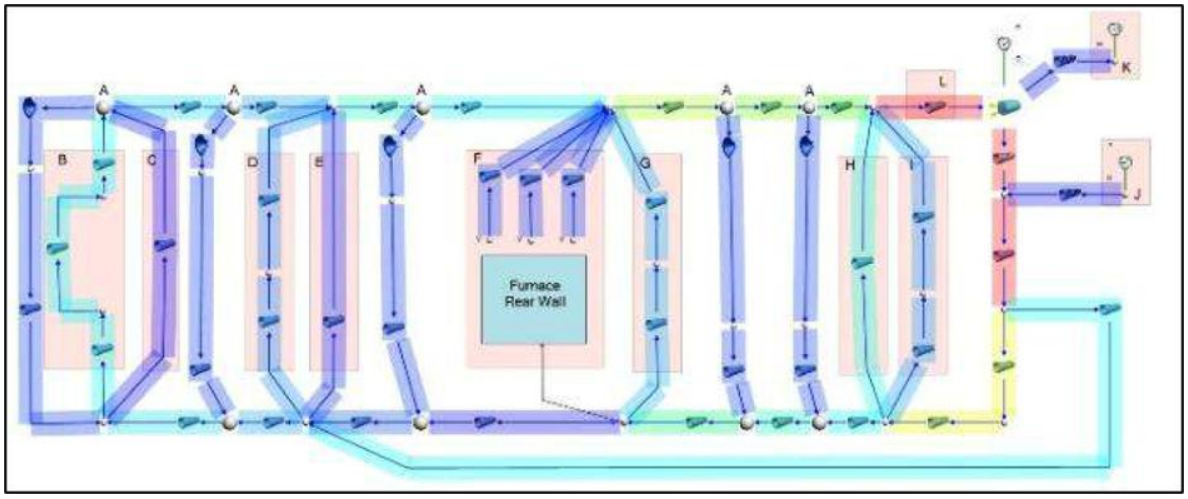

Figure 4: Predicted Flow Distribution in the Boiler Circulation Network

Figure 4 presents the Flownex model of the whole boiler circulation network. The detailed membrane wall models are linked to the circulation network through View nodes, allowing the hydraulic and thermal behaviour of the wall tubes to be reflected in the system-level results.

The colour contours indicate the relative flow distribution throughout the network, highlighting regions that carry higher and lower mass flow rates.

The results show some variation in flow distribution throughout the circuits, which is expected in a complex interconnected boiler network. The simulation nevertheless provided clear insight into the circulation behaviour of the furnace wall circuits and demonstrated how Flownex can be used to evaluate both the thermal and hydraulic performance of complex boiler systems using a 1D modelling approach.

CONCLUSION

By combining detailed heat transfer modelling with system-level flow analysis, the Flownex model provided visibility into circulation behaviour across the furnace wall circuits while avoiding the complexity of a full CFD analysis. The approach enabled tube-level thermal effects to be captured within a complete boiler circulation network, demonstrating how Flownex can be used to evaluate large thermofluid systems efficiently.