

IMPROVED EVALUATION OF RECOVERY BOILER WATER CIRCULATION DESIGN WITH THE HELP OF STATE-OF-THE-ART CFD-BASED HEAT FLUX DATA

Antti Sirainen^a, Jukka Röppänen^b, Viljami Maakala^b, Jari Lappalainen^c, Esa Vakkilainen^d

a) Andritz Oy, Kymminlinnantie 6, 48601 Kotka, Finland

b) Andritz Oy, Tammasaarenkatu 1, 00180 Helsinki, Finland

c) VTT Technical Research Centre of Finland, Vuorimiehentie 3, 02044 Espoo, Finland

d) Lappeenranta University of Technology, Skinnarilankatu 34, 53850 Lappeenranta, Finland

ABSTRACT

Modelling and simulation is an important method to evaluate process design of water circulation systems for recovery boilers. Dynamic process simulation software Apros was used in this work. Besides 1D modelling tools, another simulation methodology, namely Computational Fluid Dynamics (CFD), can be used in the water circulating studies.

In this paper, a universal method to combine the results of CFD simulations with the 1D water circulation model. The data processing of the intermediate and final results of the simulations is managed by a specific Excel tool. The new generic heat fluxes were based on heat flux maps that were made with the state-of-the-art CFD models. As a summary of the benefits, a substantial improvement to the accuracy of the water circulation models, as well as enhanced speed and user independence was achieved.

Key Words: Water circulation, CFD model, Apros, Recovery boiler, Heat flux

INTRODUCTION

Computational water circulation studies for recovery boilers are carried out to ensure that the design provides adequate water circulation. Natural circulation boilers do have some circulation challenges that have to be considered. Because these boilers do not have circulation pumps, the water-steam circuits must be designed properly. With good engineering these issues can be solved and natural circulation boilers are easy and safe to operate. Boiler manufacturers use this method to reduce risk for any unwanted phenomena such as dryouts, unstable flows, too high or low flow velocities. With system wide water circulation simulation the effects of different dimensions, pressures and capacities are easy to test.

Critical heat flux (CHF) is the most important phenomenon to consider in the water circulation design. The key factors for CHF are pressure, steam mass fraction and mass flux. At a low steam mass fraction CHF can occur at very high heat fluxes and cause a rapid temperature rise in tube walls. The tube wall temperature can become high enough for a rupture to occur. In recovery boilers CHF can be prevented with proper water circulation engineering and by ensuring constant and sufficient cooling water flow in tubes. The highest probability for CHF appears in horizontal and inclined tube parts e.g. bull nose and roof tubes. The most risky situation is, when the flow is stratified and heat is transferred from above, i.e. from the dry side. This dryout can occur when the steam volume fraction increases so high that the water flow is not enough for cooling. The burnout can occur for example in the floor tubes when the cooled smelt layer is locally removed. The flow velocities are usually very modest in the floor tubes and thin layer of steam can stratify to the upper part of the tube. If water-steam flow is unstable, CHF can occur also in vertical tubes in the furnace. The most critical areas in the recovery boilers are located in the lower part of the furnace, where a tube damage may result in a smelt-water explosion [1].

Traditionally, water circulation studies are made by using simplified heat flux distribution that is based on less accurate Computational Fluid Dynamics (CFD) models and practical experiences. This approach has been accurate enough to reach acceptable results. However, it appeared that using the new heat flux values from the state-of-the-art CFD simulations and improving the related data exchange and processing could give better tools for the water circulation studies. The main weaknesses in the previous water circulation calculations have been inaccuracy in certain areas and a large amount of work. More accurate CFD models can be used to calculate for example local heat flux values. In this paper, modern and more

realistic CFD models are used to create the heat flux values for heat transfer surfaces of a recovery boiler. The phases of the old and new heat flux feeding methods are shown in Figure 1. The improvements for the heat flux value accuracy increase the number of steps, but by automatization manual work and spent time can be decreased.

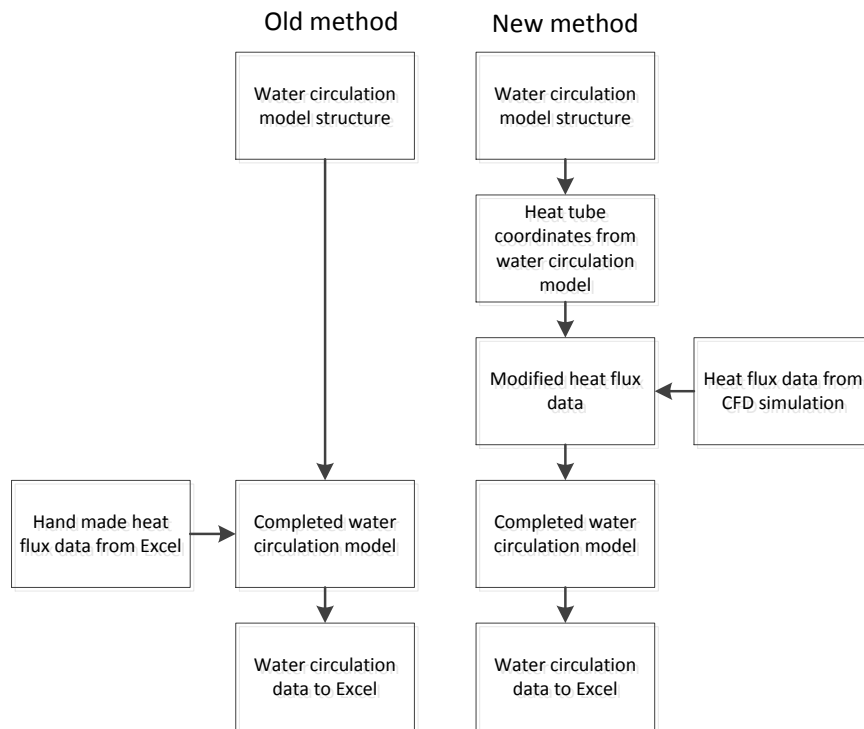


Figure 1. The steps of the old and new heat flux data feeding methods. The old method on the left and the new method on the right

MODELLING METHODS

Heat Flux Fields generated with CFD

Ansys Fluent software is used for CFD simulations in this work. State-of-the-art CFD provides a realistic way to model combustion, flow fields and thermodynamics that take place in the recovery boiler furnace. With these models, it is possible to examine multiple situations with different capacities, fouling rates and air flows. Consequently, these in-company CFD submodels offer a competent way to accurately estimate the heat fluxes to the tubes.

The simulation model was made for a modern 3000 tds/d recovery boiler furnace design. From this CFD model, all the heat fluxes were collected from the lower furnace to the end of the boiler generating bank. These state-of-the-art models can accurately solve the heat flux values also in the superheater area (Figure 2) which is a great improvement compared to the earlier models [2].

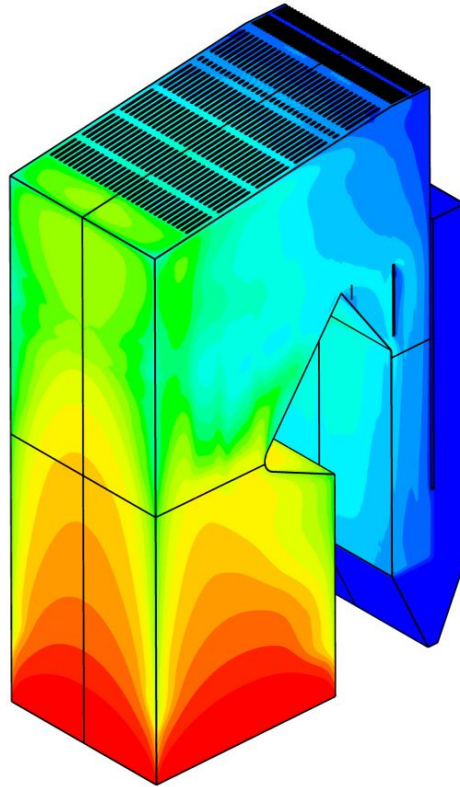


Figure 2. CFD model of the superheater region including the upper part of the furnace. A typical heat flux intensity distribution is shown for illustration (red: high values, blue: low values)

The CFD simulation produces heat flux values with very high resolution. Those have to be modified to a proper form to better act as boundary conditions in the 1D water circulation simulations. It is important to have accurate representation of the heat flux values. However, as the number of heat flux values in the CFD result is numerous, it is more practical to calculate average values using a grid that aligns with the heat structure nodalization of the 1D model.

Small CFD based heat flux squares are combined to larger squares, and these square sizes are determined for each tube based on its location. Average values for every heat flux square in the final 1D input data is formed from dozens of smaller points. This averaging process also takes into account the regions that are caused by airports and black liquor burners, where there is no heat flux solution available.

The shape of each heat flux square in the furnace wall grid is a rectangle with a height of about 2 % of the total furnace height. The squares are narrower near the corners i.e. the heat flux values are more specific in areas. The reason for this is that circulation problems are more probable in the corner tubes. In the central part of the furnace, heat flux values are noticeably larger and the circulation is therefore stronger.

The thermal power and the heat transfer surfaces are very seldom identical in different boilers. Accordingly, if there are not specific CFD simulations made from the studied boiler, the general heat flux data can be adjusted to the right size. When the thermal power of the examined recovery boiler and the general heat flux distribution are known, it is easy to calculate correction factors for every part of the boiler.

Water Circulation Model

Apros is developed for system-wide modelling and dynamic simulation of process, automation and electrical systems. When constructing a model, the user graphically selects, sets parameter values and connects suitable components, such as pipes, valves, pumps, controllers, etc. At the same time the system generates a calculation-level structure, which is directly used by the simulation solvers. Several thermal hydraulic fidelity levels are available, and in this study, the six-equation model was used. It applies the conservation equations of mass, momentum and energy to both liquid and gas phases [3].

In this work, Simantics Constant Language (SCL) was used to update parameter values and retrieving data and simulation results from the model. The environment provides a powerful script language for configuring model structures, modifying model parameters, running simulation experiments etc.

The model of this study represents the piping network and equipment used in the water-steam circulation of the recovery boiler. The model includes all the components that belong to the real water/steam circulation loops, such as tubes, tanks and valves. All these components are graphically connected to each other forming a realistic model of circulation loops, see Figure 3. For the heat transfer simulation, the pipes with heat structure (Heat Pipe) are the most important components in the model. They are used to form all the heat transfer surfaces of the recovery boiler, and consequently they receive the heat flux as a boundary condition.

Appropriate heat structures, heat nodes, are searched with the SCL commands. These heat nodes represent the outer surface of the tube wall where heat is transferred. By giving x, y, z coordinates for the connection points of the pipes and choosing appropriate nodalization in respect of axial and radial directions, all generated heat nodes in the calculating level have their own names and valid coordinates. This way the heat flux values can be properly transmitted despite of the discretization used. In case of changing the model layout or nodalization, only a re-run of the SCL command script is needed to find the updated coordinate information.

In the developed method, MS Excel is used as a support tool in data exchange and processing. Heat node data is imported to Excel via SCL commands. In Excel heat node coordinates are matched with the rectangles of the heat flux grid at the heat transfer surfaces. In other words, heat flux data from the CFD simulation's squares is linked to corresponding heat nodes. This operation is repeated to all heat nodes using Visual Basic for Applications (VBA) macros in Excel. The number of heat nodes for the whole recovery boiler simulation model was over 8000 in this example case. After matching the heat node names and the heat flux values they are transferred to the model using a SCL script. Consequently, all the heat nodes in the calculation level of the pipes have been updated with correct heat flux values. Heat flux values can be updated at any time, and the transient influence to water circulation can be monitored accordingly.

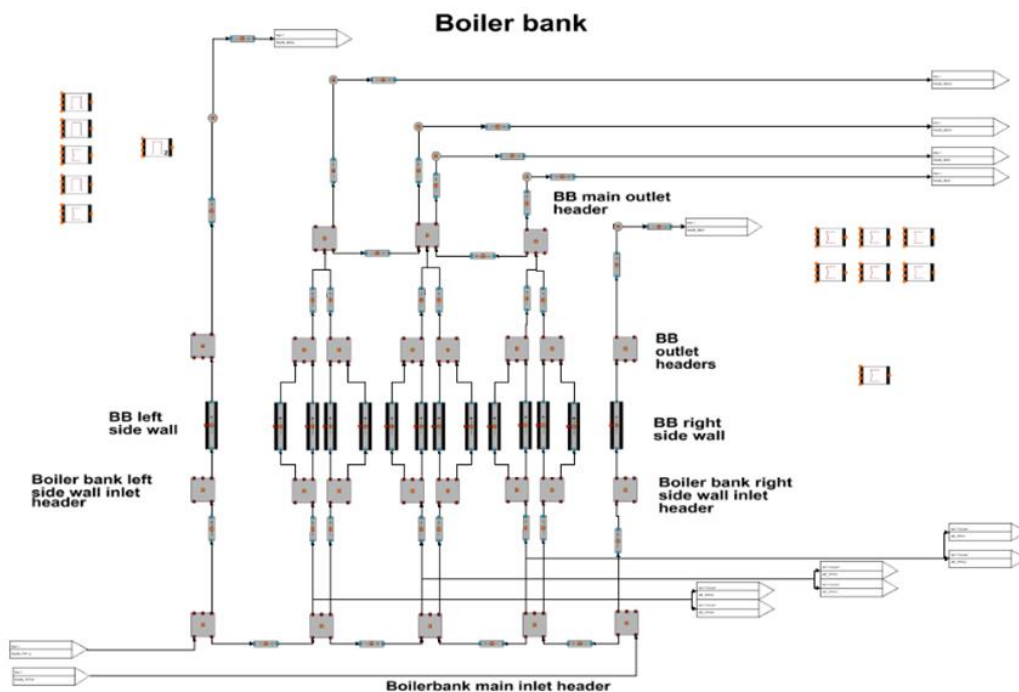


Figure 3. The flow diagram of the boiler bank

RESULTS AND DISCUSSION

The new heat flux distribution and the complete new procedure have been tested with several recovery boiler sizes and capacities. The new circulation model has been compared with the previous approach. With this comparison we were able to get information of how the heat flux distribution affects the water circulation results.

When the heat flux values are based on a modern CFD model, the heat flux map is much more detailed spatially than before. The main differences compared to previous method are in the furnace. Thermal power to furnace walls is very high and the flue gas flow inside the furnace is very turbulent. CFD simulation results from a furnace region are shown in Figure 4. Above the bull nose the heat fluxes are significantly smaller compared to the furnace region, and the differences between the old and new method are minor in this section.

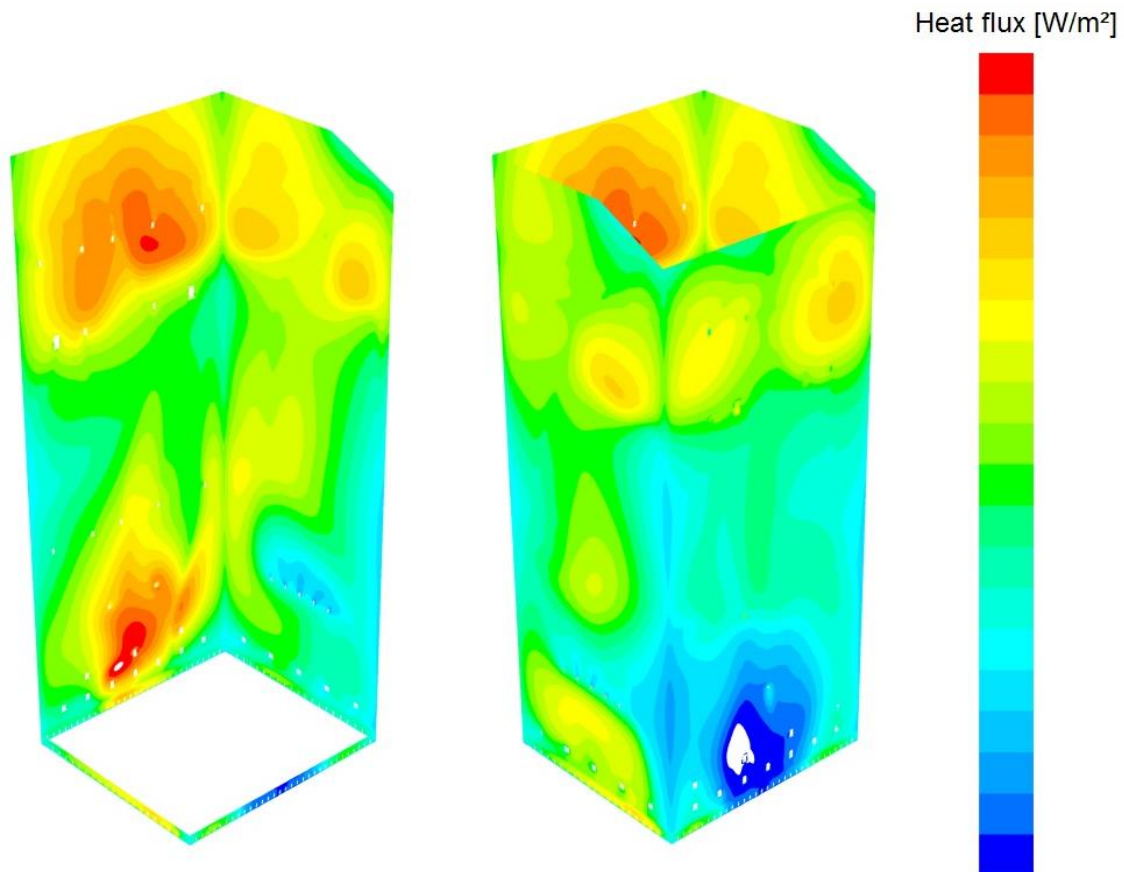


Figure 4. CFD simulation of the recovery boiler furnace. The char bed has been hidden from the figure, since heat flux to the char bed is not solved. The heat flux intensity is shown with a color map (red: high values, blue: low values)

To improve the accuracy of the simulation, the number of pipe groups was increased by 50% compared to previous simulations. This is reasonable to do with increased resolution of heat flux values. This enables to use smaller pipe groups, and therefore it is possible to get more specific data from the circulation. Improvements in the corner areas are a major achievement of this model.

The shape of the heat flux distributions vary significantly, between the old and new models. In the older version, the heat fluxes on the walls are vertically divided symmetrically into two parts and the maximum heat flux values are located near by the highest air level. In the new model, the highest heat flux values are located in the lower and upper part of the furnace. The main reasons for the different heat flux distributions are caused by improved calculation methods. Compared to the previous methods, the modern CFD model has more turbulent gas flows, different air systems and slightly different furnace geometry.

Heat flux differences cause clear changes in the water circulation results, between the old and new simulations. The total thermal power of the furnace region is identical in both cases, but in the new model, the cooler part in the lower rear wall increases the heat flux values in other furnace walls. In the previous models the thermal powers of all the furnace walls were more identical to each other, compared to this CFD based model, where the difference between the front and rear walls is notably. This is why the flow velocities at the inlet of the tubes increase in the other furnace walls, except in the rear wall. Large heat flux distribution changes occur also in the furnace screen and boiler bank tubes, causing differences in flow velocities compared to the old model. The distribution of the more accurate heat flux values is more preferable for the adequate water circulation.

Figure 5 shows that the flow velocities at the tube inlets increase in the furnace walls, except in the rear wall. Higher heat flux values are much more focused on the lower parts of the heat transfer surfaces, which causes smaller average fluid densities (ρ_G). Smaller density in the riser tubes improves the driving force and that of course increases the flow velocities. Also the height difference increases between the steam drum and the level where boiling starts. This leads to a greater driving force, according to Equation 1

$$\Delta p_{st} = g \Delta H (\rho_w - \rho_G) \quad (1)$$

where Δp_{st} is the driving force, g is the standard gravity, ΔH is the difference between the drum level and the point in the risers where the vaporizing starts, ρ_w is the density of saturated water in the downcomer and ρ_G is the average density of water-steam mixture in the risers [4].

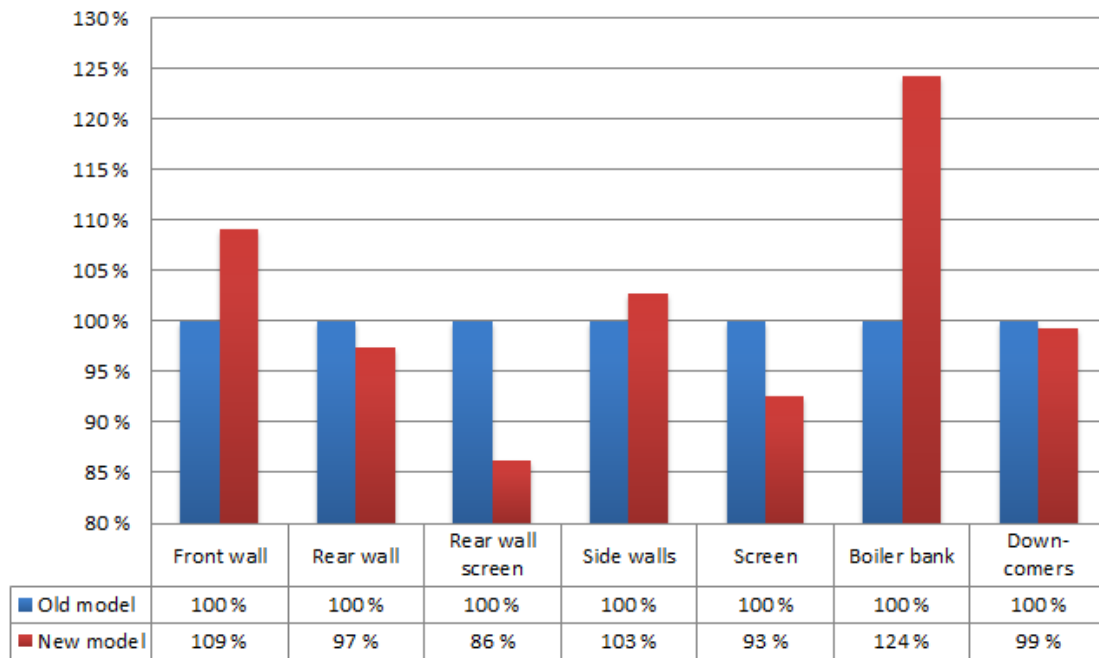


Figure 5. Comparison of average inlet flow velocities between the old and new modelling approach

Changes in the heat flux distributions can be noticed in the circulation ratio values of the recovery boiler. Circulation ratio describes theoretically how many times water have to circulate the circuit before it is vaporized totally in theory. Higher heat flux values increase the steam generation and flow velocities inside the tubes. The pressure losses in the tubes are typically higher when flow velocities and/or the steam fractions are higher and therefore the circulation ratios decrease, because the mass flows in the tubes do not increase in the same proportion with bigger steam amounts. This phenomenon can be seen in circulation ratio results in Figure 6. When the heat transfer areas with inadequate circulations are located, it is possible to implement actions to improve the circulations.

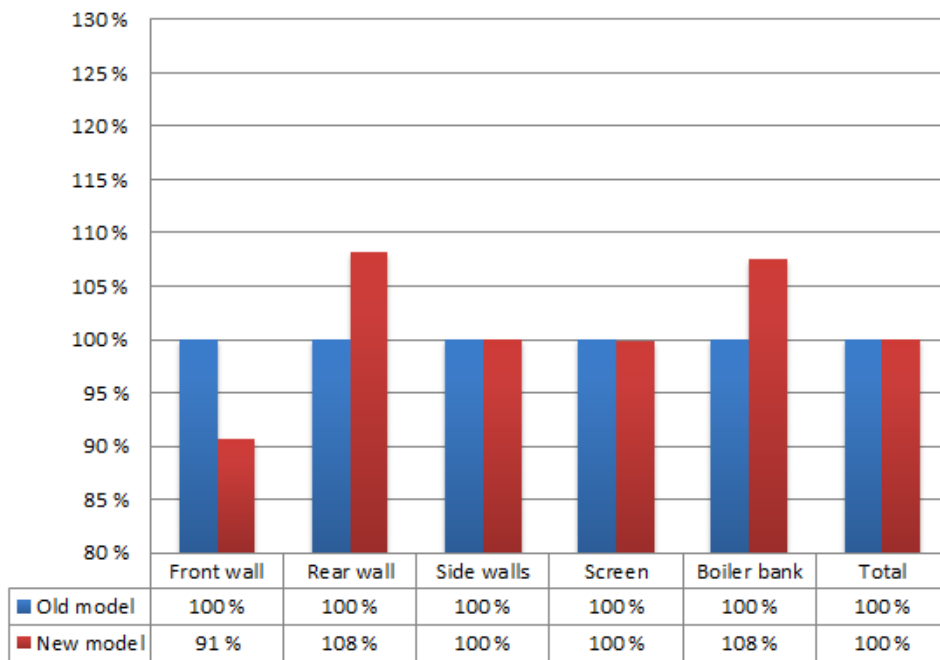


Figure 6. Comparison of circulation ratios between the old and new modelling approach

CONCLUSIONS

The aim of this work was to develop an enhanced method- for combining heat flux data from CFD calculations with water circulation model. With the new approach we succeeded to improve circulation calculation accuracy significantly. In the new model, heat flux data is based on heat flux maps that are made with state-of-the-art CFD models and the latest knowledge. Therefore, the results are much more accurate than before.

The flow in the water-steam circulation increased, which was mainly caused by the larger heat flux values on lower parts of the heat transfer surfaces. This change increases the driving force in the circuits, which increases the flow velocity and mass flow. In the new model, the resolution of the heat fluxes were increased in the corner areas which in turn increased the average heat fluxes in that area too. Particularly in the upper areas where the thermal power is commonly quite modest, small changes in the heat flux can cause relatively big shifts in the flow. All in all this change improved the circulation, especially in the boiler bank tubes. Generally, the differences were not large in thermal powers per individual heat surface, but changes in heat flux distribution were large enough to make a difference.

Inaccuracies in the results are partly due to different generalizations made in the component models and property values in the model, as well as inaccuracy in average flow results. The model components of pipes and tanks do not always meet the characteristics of specific real design and dimensions. For example, pipes with heat structures allow lumping of parallel pipes together, which greatly reduces the modelling effort of a water wall, but on the other hand, restricts the heat flux resolution for the tubes. These kinds of pipe groups are gathered together using average parameter values, which may cause small errors. Number of tubes in these groups varies depending on the position at the wall, but because of easier way to handle a lot of data, flow results are assumed as equals. These discrepancies are small and do not cause any large overall errors, but it is important to be aware of these simplifications.

Computational modeling of recovery boilers has improved remarkably in last years, but completely accurate models are impossible to implement. At the moment, the CFD models are the best tool to solve the heat flux distribution, but even the best water circulation model can only be as accurate as the CFD model. Differences in the heat flux distributions have large effects to the results of water circulation models, as can be seen in the previous simulation results. The development of better recovery boiler models continues and even better methods to simulate water circulations will be generated in the future.

REFERENCES

- [1] Stultz, S. & Kitto, J. Steam its generation and use, 40th edition, Babcock & Wilcox. 5-3 – 5-6, 1992
- [2] Maakala, V. Järvinen, M. Vuorinen, V., Experimental Validation of a Recovery Boiler Superheater Region CFD Model, PEERS Conference, Jacksonville, Florida, 2016, 921-944
- [3] Hänninen M, Phenomenological extensions to APROS six-equation model. Non-condensable gas, supercritical pressure, improved CCFL and reduced numerical diffusion for scalar transport calculation. VTT Publications 2009, 60-62
- [4] Gullichsen, J. & Fogerholm, C. Chemical Pulping – Papermaking science and technology. Helsinki: Fapet Oy. 263-265, 1999