Featured in STAR-CD Adapco *DYNAMICS* issue 23

Can still be downloaded here:

http://www.kr.cd-adapco.com/press_events/Dynamics/23/bma.html

Coupled simulations of industrial boilers

P.Tye, L. Gravel, F. McKenty, Brais Malouin & Associates Inc., Montréal, Canada

Many consider steam-generating installations to be technological dinosaurs, a legacy from the industrial revolution. However, contrary to popular belief, the steam boiler era did not end with coming of the age of aerospace. Even today over 50% of the US's energy needs are still being met using coal fired boilers. In the process industry, gas and oil fired boilers find use from oil refineries to breweries. Boiler design has until recently, however, ceased to evolve and no fundamental changes have been made to the basic boiler design for over 40 years. This position is changing rapidly, as CFD simulation provides new insight into the complex physics of an industrial boiler and allows radical new boiler concepts to be tested at relatively low cost.

This article shows how CFD can be used as a boiler design tool. An operating boiler involves the interaction of some of the most complex physical phenomena currently being handled in CFD simulations, namely turbulent combustion aerodynamics and boiling two-phase flow. The combustion aerodynamics solution is obtained using STAR-CD, which is coupled, to BMA's in house boiler waterside circulation code BOILER-II. The results of the coupled simulations provide boiler designers unprecedented insight into boiler operation.

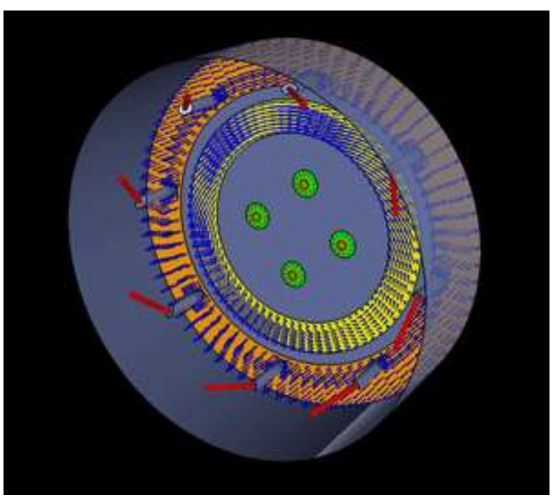

Fig 1 : Burner

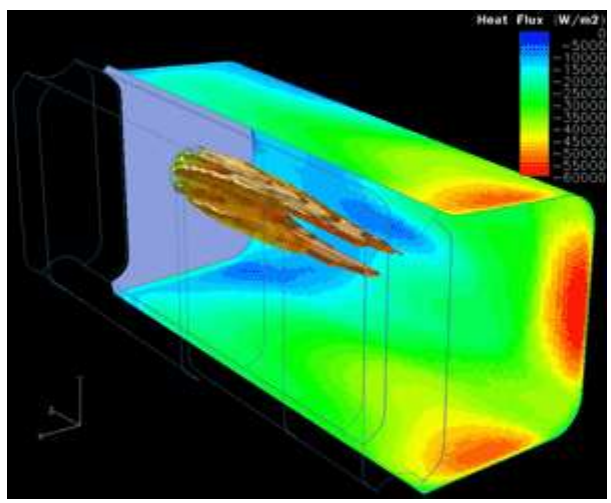

Fig 2 : Flame and furnace heat flux

The waterside flow in a boiler occurs due to natural circulation, that is to say no pump is involved. The walls of the boiler absorb the heat from the flame and produce steam, the difference in the density of the steam/water mixture in the heated tubes and the water in non-heated tubes known as downcomers drives the flow. Since the waterside circulation depends on the local heat absorption, the accuracy of its prediction is directly dependent on the quality of the combustion and aerodynamics prediction.

The boiler chosen for this study is a relatively small, 80,000 lb/hr, 12 foot drum-to-drum D-type boiler, with one 100 MMBTU/hr natural gas burner. The burner shown in Figure 1 has four central gas spuds surrounded by a 14 bladed annular air swirler, which is itself surrounded by an annular axial air injector in which 8 counter-swirling gas injectors are located. The flame resulting from the firing of this burner in this particular boiler is shown in figure 2. Its shape is the result of complex internal aerodynamics occurring in the furnace. Also shown in figure 2 is the heat flux on the water-cooled furnace surfaces. The heat flux is lowest at the front of the furnace and increases towards the target end wall where it reaches its maximum. This is further illustrated in figure 3, which shows the furnace wall temperatures, it can be clearly seen that the target wall receives the greatest amount of heat. The standard practice in the boiler industry has been to use the average furnace heat flux over an entire wall to carry out the circulation calculations. Figures 2 and 3 clearly demonstrate that with this approach potential operational problems could be overlooked.

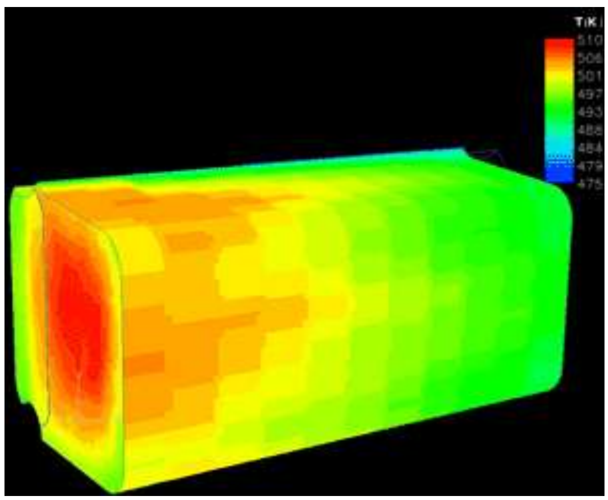

Fig 3 Furnace wall temperatures

The wall heat fluxes calculated by STAR-CD are passed to BOILER-II, which calculates the flow rates, the quality (vapor mass fraction), the void fraction (vapor volumetric fraction) and many other parameters important to proper boiler operation including the margins to dryout. Figure 4 shows the water flow rates in the furnace walls, which increase with the increase in heat flux from the front to the back of the boiler. This reflects the fact that for natural circulation the flow is a result of the heat absorption. This has the advantage of providing the greatest amount of cooling to the regions receiving the greatest amount of heat and providing a larger margin of safety to avoid overheating. The quality profile in the furnace is shown in figure 5, the quality is highest in the regions receiving the greatest amount of heat. Figure 6 shows the void fraction distribution in the furnace and in the convection bank. In the most heated regions the void fraction reach values of up to 80%. This is still perfectly safe, but doesn't tell the designer how much of a margin still exists to the operational limits. Information like this has until now been unavailable to boiler designers who for a lack of detailed information simply apply wide safety margins to their designs at large costs in boiler efficiency.

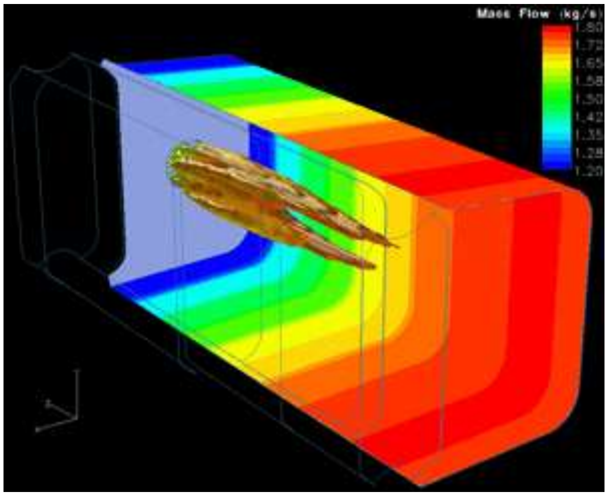

Fig 4 Water mass flow rate in furnace wall tubes

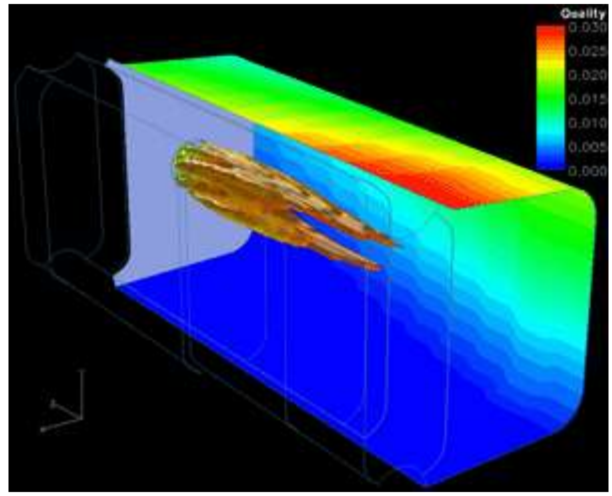

Fig 5 Quality profile in furnace wall tubes

In summary, fully coupled boiler combustion-aerodynamics and waterside circulation simulations carried out using STAR-CD and BOILER-II allow boiler designers to obtain detailed information about the operation of their boilers that was previously unavailable to them. These simulations allow new concepts to be rapidly tested and will lead to new more compact and efficient boiler designs. Boiler design engineers can now have a boiler operating in their computer the way it actually operates in the field permitting rapid testing of new design concepts.

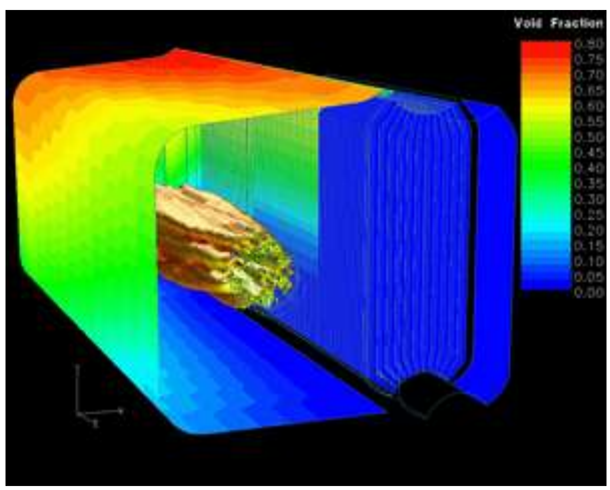

Fig 6 Void fraction distribution in furnace and convection bank