Inter Conf on CFD in Mineral & Metal Processing and Power Generation
CSIRO 1997

Experience with CFD in the Power Industry

M. Somogyi and T.E. Davidson

HRL Technology Pty Ltd, Mulgrave, Victoria

ABSTRACT

HRL (including previous work as the R&D Department of the State Electricity Commission of Victoria) has been active in Computational Fluid Dynamics (CFD) for more than 20 years. During this time many 2D and 3D models have been developed and applied to flow problems in power stations.

In 1994 HRL's modelling capabilities were extended by the lease of a mainstream CFD package which has capabilities for body-fitted and locally refinable grids and also includes additional process models, e.g. combustion.

In this paper several recent examples of successful applications of CFD modelling are described to demonstrate the benefits that can be realised when this technique is used to optimise processes of fluid flow, heat transfer, combustion and particle tracking. Examples selected are:

- velocity distribution in inlet ducts of precipitators
- mixing of hot and cold gas streams
- flow and temperature transients in a cooling pond
- simulation of gas combustors

1. INTRODUCTION

Mathematical modelling of fluid mechanics has a long tradition at HRL (including previous work as the R&D Department of the State Electricity Commission of Victoria). In the last twenty years or so, many models have been developed and applied to solve specific problems in power stations.

With mostly in-house developed programs we have solved many problems regarding 2D flows (e.g. though finite cascades), startup transients in long ducts, and isothermal turbulent 3D flows in complex ducts and furnaces which lent themselves to approximation by brick-like grid elements.

In 1994 we licensed a mainstream CFD package, Tascflow, from Advanced Scientific Computing Ltd in Waterloo, Canada. The body-fitted and locally refinable grids and the combustion capabilities of the ASC program greatly extended the area of applications.

This paper describes a number of applications of CFD undertaken at HRL.

2. ELECTROSTATIC PRECIPITATORS

One very successful application was the combined mathematical/physical modelling of the electrostatic precipitators of Hazelwood Power Station Units 1 and 2. Each unit has three precipitator boxes operating in parallel. Figure 1 shows the inlet ducting system from the airheater outlets to the precipitator inlets. Due to high dust emission levels the Units could not be operated at rated power output.

Detailed cold air velocity surveys were conducted in all three precipitator boxes of both units. The measurements showed very uneven velocity distributions at the inlet of the precipitator plates and the flow was found to be very unsteady.

Hazelwood Power had commissioned a 1/8 scale physical model study with the aim of improving the velocity distributions and, as a consequence, reducing the dust emissions through improved collection efficiency. The model measurements with the original flow correction devices showed the same problems as were noted in the site measurements.

It was important to know the velocity distributions in the complex inlet duct but due to large areas with recirculating flows detailed measurements would have been very time consuming, so we turned to numerical

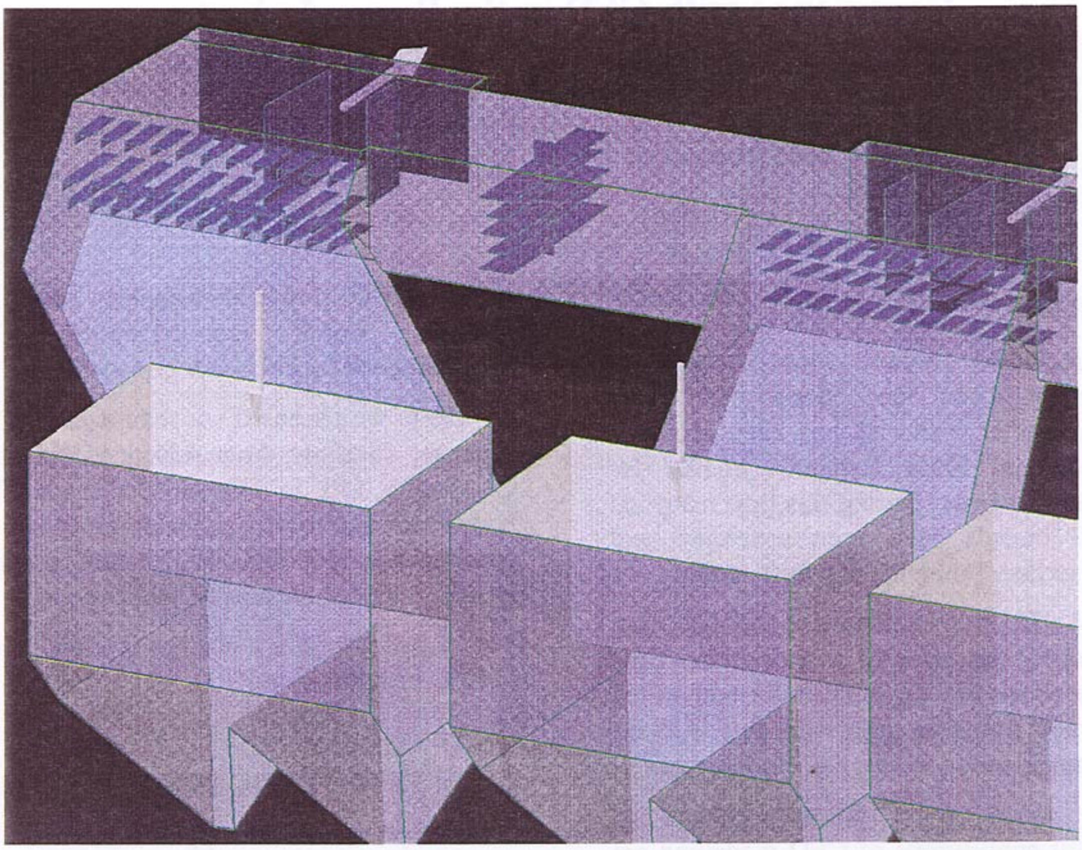

Figure 1: The inlet ducting system of the Hazelwood Unit 2 precipitator (original baffles)

modelling. We modelled the flow in one third of the manifold duct, albeit without the numerous baffles, relatively easily.

Figure 2 is a snapshot of an animation sequence (using tracks produced by Tascflow) showing the trajectories of small solid particles in this leg of the precipitator. The picture shows a large recirculation zone at the left and also indicates a maldistribution of dust at the exit. Figures 3 and 4 also show velocity vectors and streaklines within this duct.

These figures show that problems start far upstream of the original baffles. The compound effect is that the baffles are completely ineffective in providing a uniform velocity profile to the precipitator boxes.

New flow control devices were designed and the high solidity inlet screen before the precipitator plates was changed for two low solidity screens in each box. Model tests showed a vast improvement in the aerodynamic performance of the precipitators. Flow uniformity improved by a factor of 2.4, unsteadiness decreased by a factor of 3.6[1] and the pressure loss coefficient across the model was reduced by a factor of 1.5.

Hazelwood Power implemented the suggested modifications and detailed cold-air velocity surveys were conducted again in the real precipitators. These measurements showed much reduced Coefficients of Variation throughout the precipitator boxes and correlated very well with the physical model results.

---

[1] Velocity surveys in the physical model were taken using multipoint thermistor probes, capable of recording ten velocity values per second.

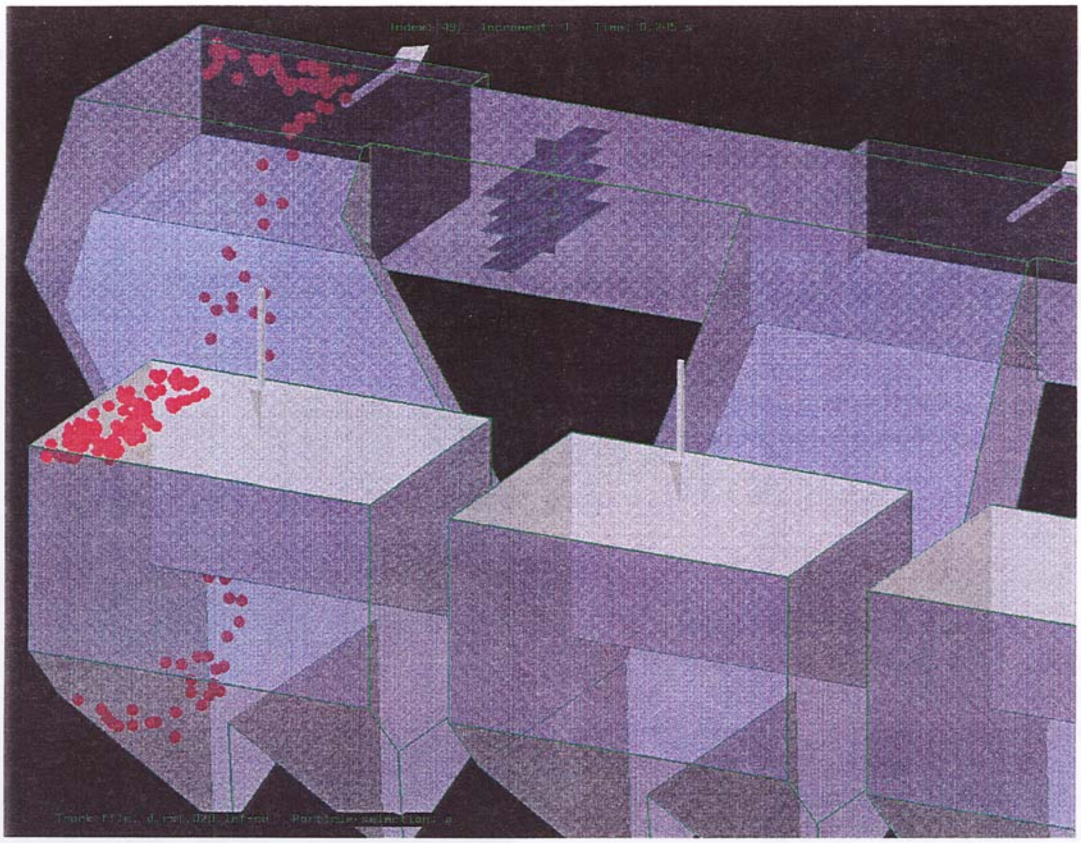

Figure 2: A snapshot of particles at a given time and at time = 0

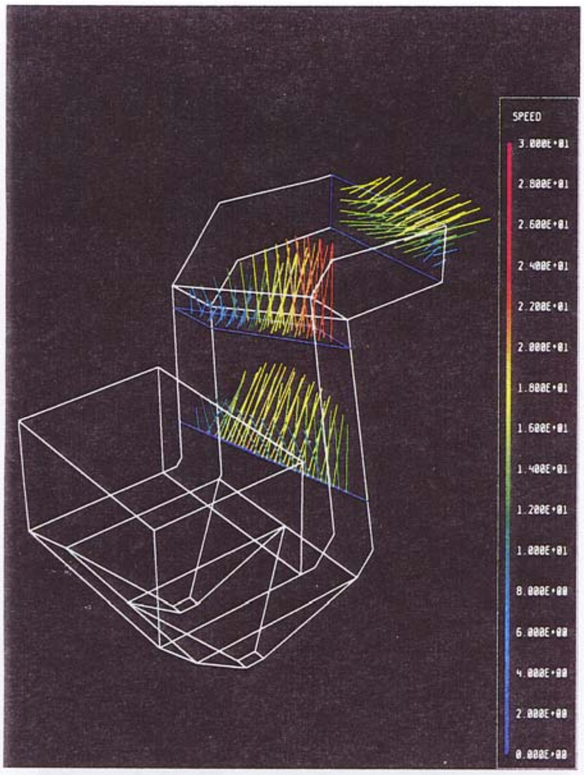

Figure 3: Velocity vectors at the beginning and at the end of the "riser" duct

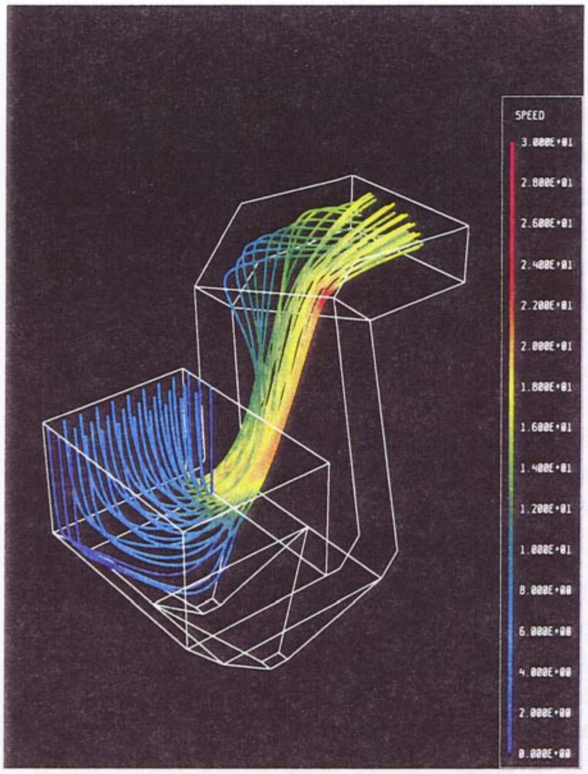

Figure 4: Streaklines starting at the left side of the inlet

When the modified precipitators went into service, the Units could be operated at full load without exceeding the dust emission limits. The distribution of dust within the ductwork and precipitators was also substantially improved and when opened for cleaning the outlet evase now had only 150 mm of dust layer instead of up to 1500 mm as it used to be. Another beneficial effect of breaking-up the very large and unsteady eddies in the inlet duct is the damping-down of a low frequency vibration through the furnace and the precipitator.

3. GAS OFFTAKE

Another application was the re-arrangement of attemperating air ducts on the gas offtakes of a power station furnace (Figure 5). Ash particles may stick to the walls of these offtakes and a thick coating may form in a short period of time. The Station employs air lances to remove these coatings whilst the boiler is in operation. However, it is difficult to remove deposits from the top end of the vertical part of the duct. The purpose of the study was to find ways of lowering the wall temperature there below the fusion temperature so that ash particles would not adhere to the surface.

A numerical model was set up which modelled the flow in the offtake in detail. In order to provide truly representative inlet conditions to it, a much simplified furnace was "attached" to the duct (Figure 6). The furnace inlet conditions were set to give gas offtake inlet temperatures that matched the actual conditions being modelled. Only one offtake was modelled in detail, the others were treated as simple rectangular ducts. The whole model was less than 100,000 nodes, easily manageable on a modern workstation.

The calculations showed where the large particles impacted at the back of the vertical duct wall and thus where the temperature reduction was required. The massflow of the attemperating air jets was sufficient to provide the average mixing temperature required but the jet velocities were too high, and the jets merged mostly at the centre of the duct thus reducing temperatures at the wrong place (Figure 7). With more pipes and lower attemperating air velocity the surface temperatures could easily be controlled to more than 100°C below the ash fusion temperature.

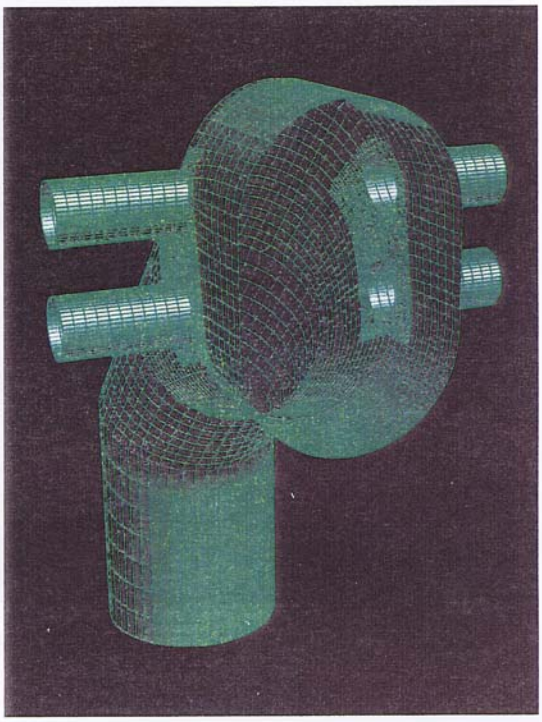

Figure 5: Gas offtake with original attemperating air pipes

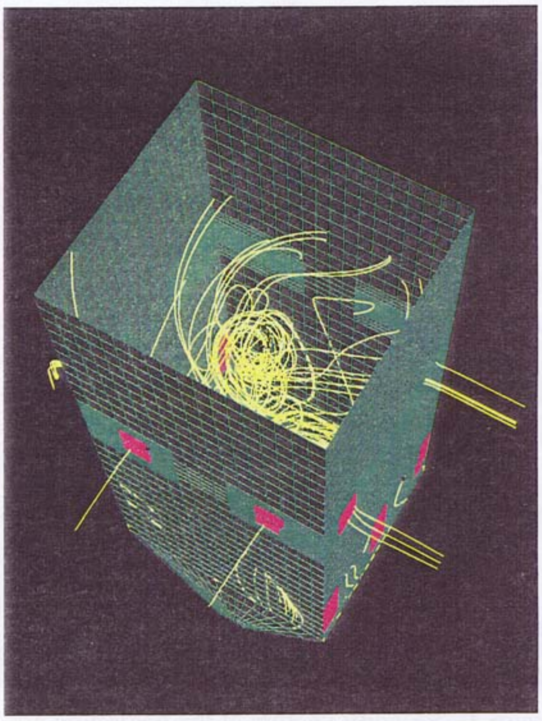

Figure 6: Flow in a simplified furnace "attached to the gas offtake" in order to provide realistic boundary conditions

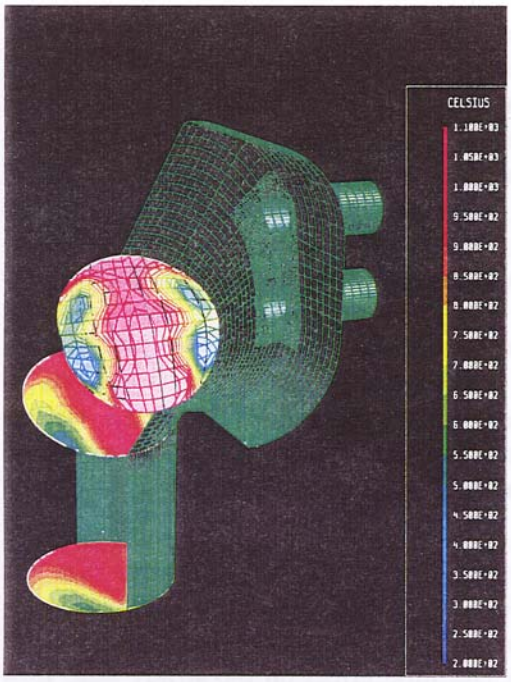

Figure 7: Temperatures on cross-sections of the offtake duct with the original attemperating air pipes

4. IDGCC

A number of investigations were carried out for a major research project into the Integrated Drying Gasification Combined Cycle (IDGCC) process.

We computed combustion with the Eddy Dissipation Model (EDM) of Tascflow in several combustors and above the solid bed of a gasifier. The results are in good agreement with observations and measurements and give us confidence in the prediction of flow and combustion phenomena in cases when physical experiments would be too costly or difficult to conduct.

Figure 8 also shows some of the results of modelling flow in the free-board of HRL's pilot-scale gasifier. Oxygen volume fractions in a pilot-scale rich-lean combustor for low calorific value gas are also shown in Figure 9. Due to symmetry considerations, only a segment of the combustor needed to be simulated.

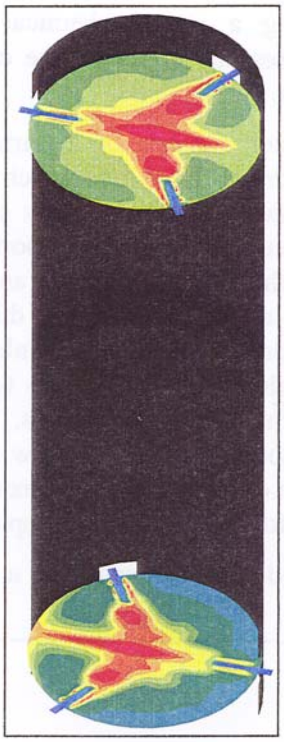

Figure 8: Temperature distribution in cross sections of the HRL pilot-scale gasifier

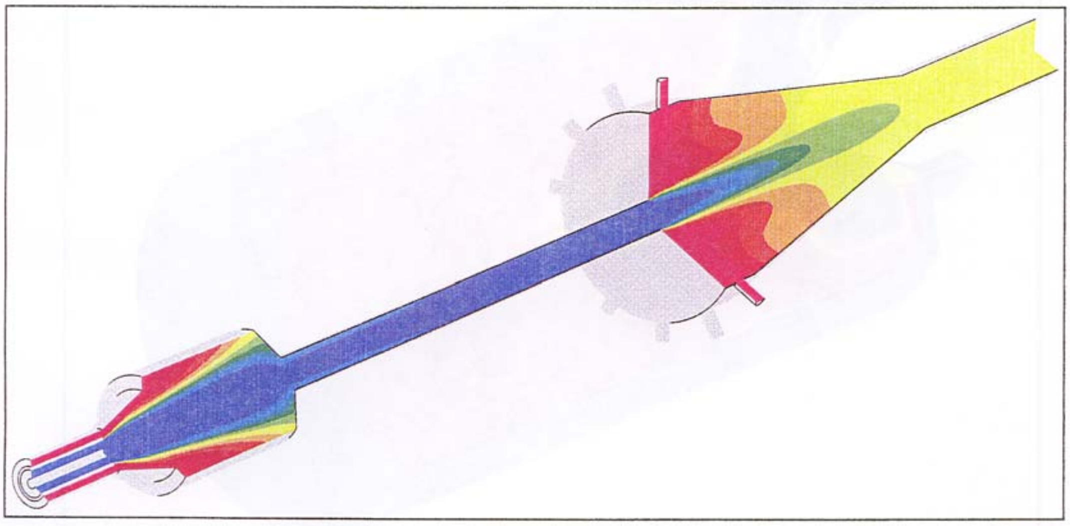

Figure 9: O_2 concentrations in HRL's pilot-scale rich-lean combustor for low calorific value gas

HRL's Coal Gasification Development Facility (CGDF) at Morwell includes a 5 MW gas turbine which has 6 metal can combustors. During normal operation, the combustors burn LCV (low calorific value) coal gas but at startup, oil is burned. Tascflow was used to find the steady state flow patterns and species concentrations existing in the combustors for a number of air and oil inlet conditions. The effects of modifying the burner geometry were also investigated. In addition, one run was repeated using a staged chemical reaction scheme to assess its effect on the combustor characteristics.

Each combustor consists of a burner and a circular cylindrical flame tube which contracts to an outlet duct. The burner has concentric swirlers and numerous other inlet ports. On the periphery of the flame tube there are a series of secondary air inlet holes and dilution air holes. There are also annular air inlet regions at locations along the flame tube to provide cooling for the flame tube walls. The grid generation capabilities of Tascflow made the representation of the complex geometry within these combustor cans relatively simple.

Oil is sprayed into the cans via a series of nozzles near the burner axis. In order to model its combustion, the oil was assumed to consist of a single hydrocarbon. The liquid oil was assumed to be vaporised on entry to the combustor and for the initial runs, oxidation was modelled by a single reaction producing CO_2 and H_2O. Combustion was modelled using the EDM which specifies the reaction rate in terms of fuel and oxidant concentrations and the turbulence parameters k and ε. Figure 10 shows the variations in steady state temperatures predicted on the centre and periodic planes of a simplified combustor model.

The model was also run using a more complex description of the chemistry. Combustion was represented by a four reaction mechanism for which kinetic reaction rates were specified. This had little effect on the steady state flow patterns but resulted in a slight increase in maximum temperature.

The modelling gave an insight into temperature and flow patterns existing in the combustor for various inlet conditions.

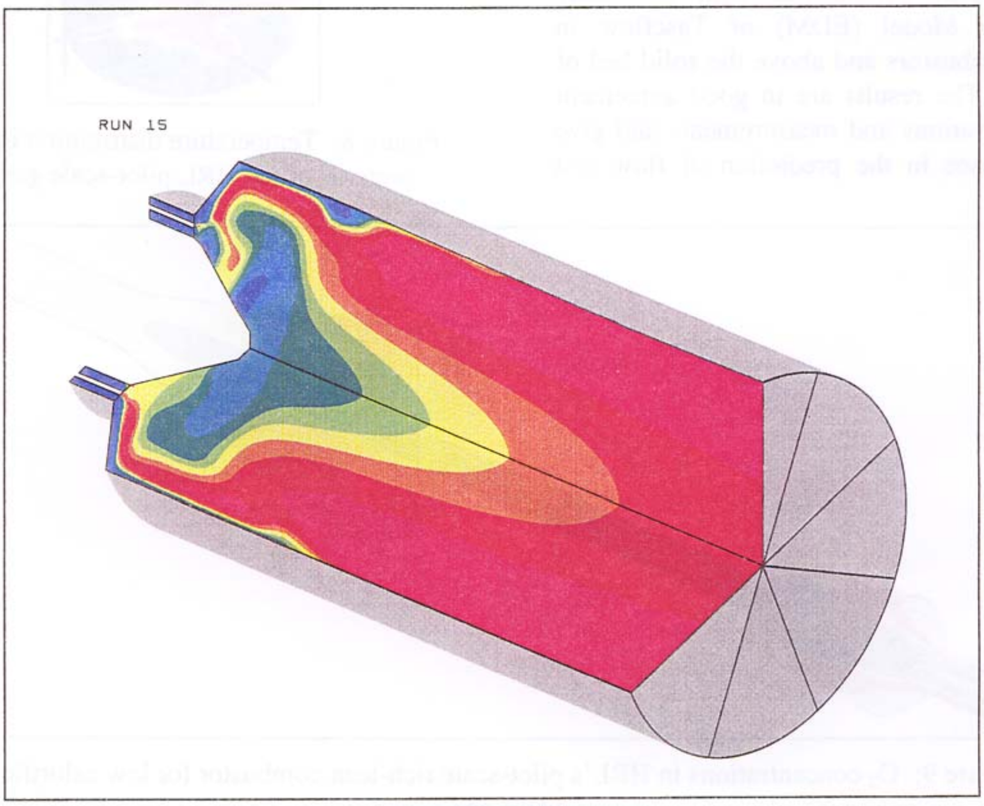

Figure 10: Temperature on the centre and periodic planes of a simplified gas combustor model.

5. COOLING POND

A model was developed using Tascflow to predict the flow and temperature fields in a power station cooling pond under different operating and meteorological conditions. The model was used to simulate the effect on pond temperature of the operation of different numbers of power station units during hot summer weather.

Since Tascflow cannot model the motion of a free surface, the water surface was assumed to be flat and a symmetry boundary condition was applied, essentially treating the surface as a slip-wall. The level of the pond was assumed to be constant.

Buoyancy forces were modelled via Tascflow's treatment of natural convection based on the Boussinesq approximation. Density variation was assumed to be proportional to temperature variation with compressibility set to a constant value.

Because of the potentially large vertical temperature gradients existing in the pond, it is desirable to have a reasonably fine grid resolution in the vertical. However, the pond's horizontal dimensions are more than 100 times greater than its depth and large grid cell aspect ratios can cause rounding errors in the model. Therefore a compromise had to be made between model accuracy, which requires high vertical resolution and low aspect ratios, and the number of grid points which is limited by available computer hardware. A vertical grid spacing of no more than 1m was used.

At the pond surface, heat is transferred between the water and air due to processes such as radiation and evaporation. This heat flux depends on water surface temperature, solar radiation and ambient air conditions and therefore varies with time and location. Since Tascflow is unable to simulate such a boundary condition directly, a source term was included in the energy equation for each surface grid cell.

The stress acting on the pond surface due to wind is also time dependent and cannot be specified directly as a boundary condition in Tascflow. Therefore, wind stress was incorporated into the model by applying source terms to the two horizontal momentum

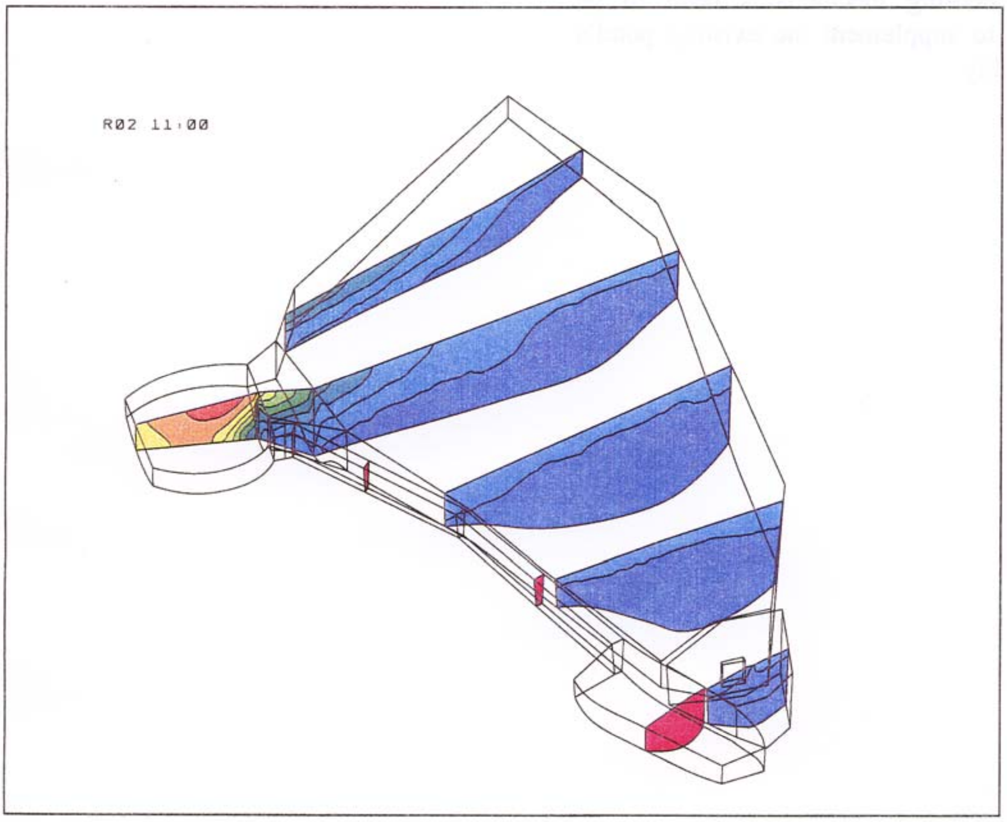

Figure 11: Predicted temperature profiles on the cooling pond (vertical scale increased by a factor of 50 for clarity)

equations in the layer of cells at the pond surface.

Lack of detailed historical temperature and flow measurements limited the extent to which the model could be validated and made specification of initial conditions difficult. Power station outlet flow rate information was also limited so it was assumed that the flow rate was proportional to the number of units in operation. The number of units in operation was estimated from the daily average power output figures. It was also assumed that there was a constant temperature rise across the power station condensers between the pond outlet and inlet.

The model was applied to two different power station operating regimes in hot summer weather. Temperature distributions on a series of vertical planes in the pond during one of the simulations are shown in Figure 11. Temperature stratification can clearly be seen, with a warm layer forming near the surface.

Results of the simulations gave an indication of the pond's capacity to dissipate heat under the most extreme conditions likely to exist. This information can be used to assess whether additional cooling mechanisms need to be introduced to supplement the existing pond's cooling ability.

6. CONCLUSIONS

The above examples demonstrate that CFD can be used to great advantage in many areas in a Power Station environment either in "stand-alone" mode or in conjunction with field tests and physical model studies.

Typical problems are

(i) the difficulty in obtaining reliable data; and

(ii) the complexity of systems where no cross section with a more-or-less uniform velocity distribution can be found to serve as a starting point for the model.

Costs of CFD models are much lower than physical model studies and CFD can provide more detailed information on distributions of velocities etc. In some instances, due to the complexity of Power Station systems it may also be preferable to validate the CFD predictions with a physical model study.

ACKNOWLEDGEMENTS

HRL would like to acknowledge the permission of its clients to include these studies in this presentation.