

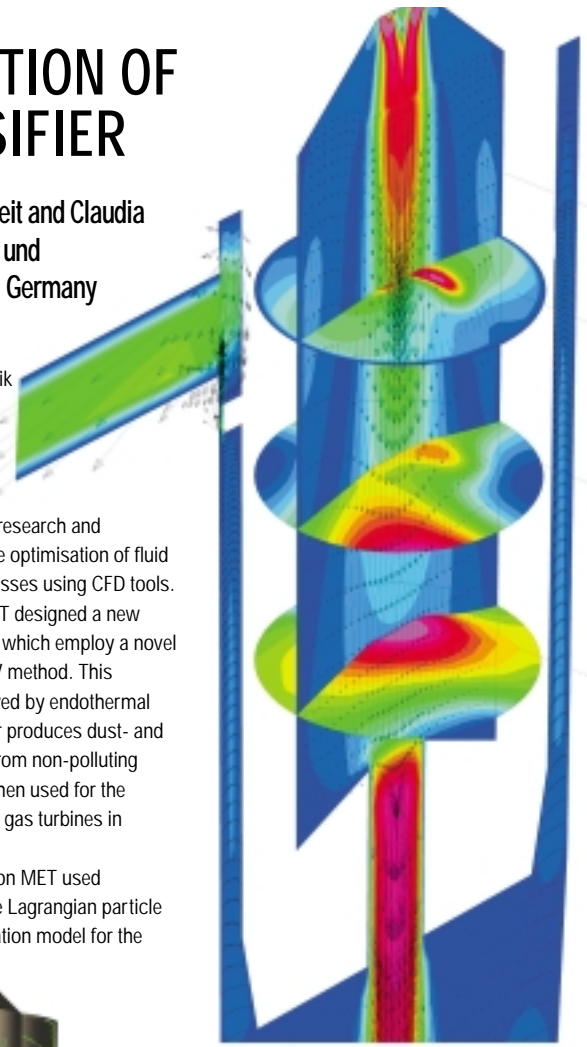
# OPTIMIZATION OF COAL GASIFIER

by Siegfried Bludszweit and Claudia Escher, MET Motoren- und Energietechnik GmbH, Germany

Founded in 1992, MET Motoren- und Energietechnik GmbH is an independent consulting company providing a wide variety of services in different areas of mechanical engineering research and development. These include optimisation of fluid flow and combustion processes using CFD tools.

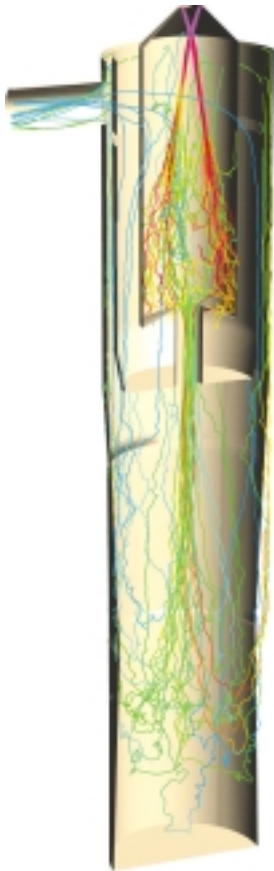
In a recent project, MET designed a new generation of coal gasifiers which employ a novel process, called the Carbo-V method. This consists of pyrolysis followed by endothermal gasification, which together produces dust- and sulphur-free gaseous fuel from non-polluting raw materials. This fuel is then used for the operation of gas engines or gas turbines in small power plants.

For the flow optimisation MET used CFX-TASCflow, coupling the Lagrangian particle tracking model, the gasification model for the



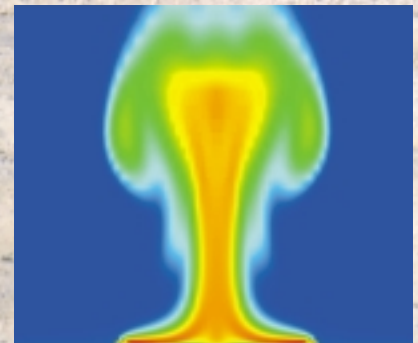
Above: Flow distribution in the gasifier.

Left: Coal particles in the gasifier.



combustion of coal dust and a three-step eddy dissipation model for combustion in the gas phase. MET calculated the flow processes, temperature and concentration of selected gases in the Carbo-V facility. In contrast to traditional experimentally-based empirical design methods, where the gasifier is a black box with defined inlet and outlet conditions, the CFD simulation gives information on the internal processes in detail. In particular, coal dust streak lines show where most of the reactions occur as the particle diameter decreases with combustion.

A more thorough understanding of the gasification process can be expected from further applications of this virtual gasifier concept. Better predictions of gas and particle thermodynamics, and their complex interacting chemical reactions will provide an optimal gasifier. Moreover, expensive series of experiments can be reduced to a minimum, and the production of gas of consistent quality can be guaranteed.



# Pool fire modelling

by Yehuda Sinai, AEA Technology

The physics of pool fires is extremely complex, as the supply of fuel through evaporation is driven by the radiative and convective heat exchange at the pool surface. The bulk flow in such fires exhibits the whole spectrum of conditions, from laminar to fully turbulent (and buoyant, of course), and the thermal radiation emission is fuel-dependent, principally through soot formation and the level of shielding afforded by the soot.

Good correlations are available for the gross characteristics of unconfined pool fires, but it behoves the CFD community to demonstrate that this scenario can be predicted. Most successful CFD predictions of this state, with or without wind, have assumed a source of turbulence at the foot of, or within the fire, generally on the basis of burner experiments. Unfortunately, these sources are not universal constants, and doubts arise in relation to extrapolation of burner behaviour to pool fires, as well as the influence of scale. However, AEA Technology reported in 1995 that with careful modelling, realistic

predictions of the with-wind case could be made using the standard buoyancy-modified  $k-\epsilon$  model, without the additional turbulence sources.

AEA Technology is currently investigating whether the more difficult scenario of a pool fire without wind can be simulated equally well with the standard models. The computations include combustion (one-step eddy break-up) and radiation, and have successfully predicted the character and frequency of the quasi-regular shedding of toroidal vortices known as 'puffing'. Generally, these vortices have a major effect on entrainment and radiative emission.

The work, on a 20m kerosene pool, has adopted a fixed evaporation rate, based on experimental data. The predicted average puffing period is about 3 seconds, which agrees well with the Cetegen-Ahmed correlation. The predictions indicate a bimodal behaviour, in which alternate vortices overtake preceding ones along the core.

Future work will consider alternative turbulence models as appropriate, as well as the fuel-fire coupling.

Background: A 6m kerosene pool fire experiment conducted at AEA Technology's Pool Fire Test Facility. Large eddies are revealed by the copious soot regions.

Right: A typical image from a 3-D simulation. In addition to the puffing and bursting phenomena, CFX predicts the plume 'buckling' which has been identified in the literature.

Below: A sequence of temperature fields from the axisymmetric simulation showing bimodal behaviour. The puffing phenomenon produces large slow vortices that alternate with bursts of hot gas along the fire core.

