Harpur Hill, Buxton
Derbyshire, SK17 9JN
T: +44 (0)1298 218000
F: +44 (0)1298 218590
W: www.hsl.gov.uk

HEALTH & SAFETY LABORATORY

Review of CEBAM Explosion Model

HSL/2006/112

Project Leader: **H. S. Ledin**

Author(s): **H. S. Ledin PhD DIC MSc**

Science Group: **Health Improvement Group**

ACKNOWLEDGEMENTS

The input from Dr J. K. Clutter, Analytical & Computational Engineering Inc. and University of Texas at San Antonio in USA, and Mr M. V. Hasson, MMI Engineering Ltd. in the UK, is gratefully acknowledged.

CONTENTS

EXECUTIVE SUMMARY

A number of different gas explosion models are used by duty holders, or by the duty holder employed contractor, in the preparation of safety cases. The CEBAM model, developed by Dr Clutter at University of Texas at San Antonio, USA, is the focus of attention in the present report.

Objectives

- To review and assess the physical sub-models and numerics in the CEBAM code;
- To assess the strengths and limitations of the CEBAM code; and
- To review the validation of the physical sub-models in the CEBAM code.

Main Findings

- CEBAM is based on a gas-phase, multi-step, finite-rate combustion CFD code for aero-propulsion applications;
- CEBAM assumed a controlled, known combustion rate, which is not influenced by random fluctuations;
- The CEBAM code is intended to be used for evaluating various designs during the design concept phase of a project;
- The CEBAM code can solve the governing set of partial differential equations in either Euler or Navier-Stokes form. Typically, the Euler form of the equations is solved when modelling gas explosion scenarios;
- The combustion reaction is assumed to be a one-step, irreversible chemical reaction involving three species, fuel, air and products;
- The computational grid typically consists of cubic cells, with 1 m long sides. The user can specify smaller cells if required. It is not clear if the constants in the sub-grid scale modelling of obstacles are still valid if smaller than 1 m cube cells are specified. The mesh does not explicitly represent smaller obstacles, since these will frequently be of much smaller dimension than 1 m;
- The user has to specify an appropriate flame Mach number that will take into account the effects of the turbulence generated as the flow passes around obstacles, etc. The flame Mach number is the most important factor influencing the predicted explosion-generated over-pressure. Essentially guessing the flame speed is analogous to guessing the over-pressure;
- The choice of appropriate flame Mach number is based on: the experience of the user, tables generated by Baker Engineering and by consulting the developers of the CEBAM code and not on real time, accurate experiments;
- Choosing the flame Mach number is *difficult*, particularly as it is not even constant within a single explosion scenario. It is *also difficult* to ensure that the chosen value leads to a *conservative* over-pressure prediction. This can lead to highly user dependent results; and

- The CEBAM code cannot handle external explosions, which make a large contribution to the overall pressure, without specifying a flame Mach number in the region outside the process module where the gas explosion was initiated.

Recommendations

- The choice of flame Mach number that would lead to a conservative prediction of explosion-generated over-pressure is crucial. There is a certain amount of arbitrariness in the way the flame Mach number is chosen, which makes it very difficult to audit the process, i.e. how to decide whether an appropriate flame Mach number has been chosen – this can really only be answered by comparison of the predictions with experimental data. The flame Mach number could be arrived at through the use of another model;

- The CEBAM code is intended to be used during the design concept phase of a project and not as a tool for calculating over-pressures on existing platforms. This should be borne in mind when considering use of the code;

- Explosion modelling using CEBAM is probably more dependent on the expertise and judgement of the user than with other CFD codes. This increases the importance of training and support for users, the desirability of guidelines for users of the code, and the information that must be sought by the regulator in assessing CEBAM-based safety submissions; and

- CEBAM is being actively marketed with the UK. It has already being used in support of offshore safety cases. It is recommended that HSE obtains in-house experience in the use of CEBAM to better understand its behaviour and to inform the assessment of safety submissions based on the code.

1 INTRODUCTION

Assessment of the risks posed by gas explosions requires predictive models. The physics of the processes involved in an explosion, i.e. coupled fluid dynamics and combustion, are very complex. The understanding of these processes is not complete, something which is reflected in the models.

There are a number of models of differing levels of complexity, which can be used to predict the consequences of explosions. These range from simple empirical correlations based on sets of experiments, via phenomenological models - in which more physics has been retained, to Computational Fluid Dynamics (CFD) models. Each type of model has its strengths and weaknesses, see Ledin (2002) for a more detailed explanation. Briefly, the empirical models are really only applicable within the range of conditions used in the experiments. It is not advisable to use models outside this range. Phenomenological models involve solving ordinary differential equations for various zones. The models are tuned to sets of experiments, in a similar fashion to the empirical models, and should not be used outside this range of conditions against which the models have been calibrated.

The most sophisticated type of model is the CFD model. CFD involves solving the Navier-Stokes equations, which govern all fluid flows. The limitations of our understanding of the physical processes, coupled with highly complex geometries and the requirement to account for all length and time scales of importance make CFD models more expensive, in terms of computer run times and memory requirement. There are also problems, which are not directly related to the physics, e.g. numerical issues such as discretisation of the partial differential equations, etc., which need to be considered. Nevertheless, CFD models are the only type of model which offer the prospect of generality, although the current models tend to fall some way short of meeting this requirement.

The CEBAM model, which is the topic of the present report, falls into the category of CFD models. However it embodies some simpler models to represent combustion. Information about the model has been gathered from articles published in the open literature (see reference list in section 4) and from a meeting on 5/12/02 with one of the code developers – Dr Keith Clutter, of University of Texas at San Antonio, and Analytical and Computational Engineering. It would appear that the roots of CEBAM are in the modelling of combustion problems for aero-propulsion. It has been extended to the modelling of gas explosions.

The main objective of the present work has been to gain greater understanding of what the CEBAM code does when modelling gas explosions and how it does it, i.e. its physical and numerical basis, and validation. However it would appear that the code is also used to model dispersion of a gas release, as part of a risk assessment, but this is has only been done in-house and no thorough validation against dispersion experiments has been carried out. The applicability of the CEBAM code for dispersion modelling is not discussed further.

This report describes the physical sub-models and the numerics implemented in the CEBAM code as used for modelling gas explosions; code validation is also covered.

2 DESCRIPTION OF PHYSICAL SUB-MODELS AND NUMERICS

2.1 GOVERNING EQUATIONS

Fluid flow can be represented by a number of conservation equations, which take the form of a set of partial differential equations.

The equations below are written in their Navier-Stokes Cartesian tensor form. The Euler form of the equations is used in most, if not all, dispersion and explosion calculations, according to Dr Clutter (2002). This means that the effects of turbulence and viscosity are not included in the governing equations. It is unusual to see the Euler equations used as the starting point for modelling the near-field effects of gas explosions, since the effects of turbulence in accelerating the flame speed is a key means for the generation of over-pressure. More commonly the Euler equations are applicable for inviscid flows, applied where the effects of turbulence and viscosity are negligible, such as blast wave propagation.

The governing equations solved in CEBAM are given below. It is useful to reproduce these here since it is not always entirely clear as to the exact form of equations solved.

2.1.1 Mass conservation equation

The mass conservation equation can be written as:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho U_j}{\partial x_j} = 0 \qquad (1)$$

where ρ is the fluid density, x_j is the spatial location in direction 'j' and U_j is the velocity component in spatial direction 'j'.

2.1.2 Momentum conservation equation

The equation for the conservation of momentum can be written as:

$$\frac{\partial \rho U_i}{\partial t} + \frac{\partial \rho U_i U_j}{\partial x_j} = \frac{\partial}{\partial x_j}\tau_{ij} - \frac{\partial P}{\partial x_i} + \rho g_i \qquad (2)$$

where P is the pressure, g_i is the gravitational acceleration in spatial direction 'i', and U_i and U_j are the velocity components in spatial directions 'i' and 'j', respectively. The first term on the right hand side of Eq. 2, is the viscous stress tensor, τ_{ij} – the stress acting in the 'i' direction on a surface normal to the 'j' direction. The stress tensor is set to zero in the Euler equations that are typically solved for gas explosion scenarios in CEBAM, e.g. all viscous effects are neglected and thus no account is taken of turbulence generated by flow around obstacles, etc. However, the flow generated by a gas explosion is viscous. The turbulent effects are contained in the flame speed, which has to be somehow assigned an appropriate value.

2.1.3 Energy conservation equation

It is necessary to solve an equation for the conservation of energy, or enthalpy, when modelling compressible flows. The energy conservation equation can be written as:

$$\frac{\partial \rho E}{\partial t} + \frac{\partial \rho U_j E}{\partial x_j} = -\frac{\partial}{\partial x_j} J_{h,j,} + S_h \qquad (3)$$

Where the total energy is given by

$$E = e + \frac{1}{2}\vec{u}^2,$$

and $J_{h,j}$ is the diffusion of enthalpy in the jth direction, e is the internal energy and $1/2\,\vec{u}^2$ represents the kinetic energy. The internal energy for each of the chemical species can be calculated from;

$$e = E - \tfrac{1}{2}\vec{u}^2 = \sum_{l=1}^{n_{species}} \alpha_l h_l - \frac{P}{\rho}$$

where

$$h_l = h_{f,l}^o + \int_{T_{ref}}^{T} C_{p,l}\, dT$$

where $h_{f,l}^o$ is the heat of formation and C_p is the specific heat capacity at constant pressure for species 'l', respectively.

The enthalpy flux term $J_{h,j}$ is set to zero in the Euler form of the equation. The effects of diffusion on the enthalpy is neglected, e.g. the flow is inviscid.

2.1.4 Species conservation equation

It is necessary to solve a species conservation equation for each of the species involved in the combustion reactions. The species conservation equation can be written as:

$$\frac{\partial \rho \alpha_l}{\partial t} + \frac{\partial \rho U_j \alpha_l}{\partial x_j} = -\frac{\partial}{\partial x_j} J_{l,j} + \dot{\omega}_l \qquad (4)$$

where α_l is the mass fraction of species 'l', $J_{l,j}$ is the mass diffusion of species 'l' in the jth direction, and $\dot{\omega}_l$ is the chemical reaction source term for species 'l'. The mass diffusion term is set to zero in the Euler equations.

2.2 TURBULENCE MODELLING

Viscous and turbulence effects on flame behaviour and acceleration, a key component in over-pressure generation, are not calculated explicitly in CEBAM. Instead they are taken into account through a reduced model based on user-prescribed flame speeds, Clutter (2001). Indeed the Euler equations, which do not contain any turbulence or viscous effects, are used when calculating gas explosions. It is not clear from the literature how well the over-pressure is predicted; i.e. the accuracy of the model predictions for complex, real geometries has not been shown. Turbulence models have apparently been implemented in the CEBAM code, Clutter

(2002), but it is not known which models are currently available in the code and it appears that they are not used for explosion calculations.

2.3 COMBUSTION MODELLING

The combustion processes are modelled as an irreversible one-step chemical reaction involving three species, namely Reactant + Air → Product. The specific heat for each species is assumed to be constant and is averaged over the range of temperatures characteristic of a gas explosion. The values used in the CEBAM code were chosen to reproduce the specific heat ratio, γ, used in Luckritz's model, Luckritz (1977).

The combustion model in the CEBAM code is flame speed based, Clutter (2001). Indeed the user must specify a flow Mach number, M_w, which is equivalent to the flame speed. Note that the flame speed is not known a-priori for all situations. Nevertheless, tables of flame speeds have been compiled based on empirical information by Baker, Tang, Scheier and Silva (1996) and Baker, Doolittle, Fitzgerald and Tang (1998). The flame speed is dependent on the dimensionality of the flame expansion, fuel reactivity and obstacle density, Tang and Baker (1998).

The table of flame speeds referred to above was compiled for a different model, the Baker-Strehlow model. Nevertheless, these empirical flame speed correlations, along with engineering judgment, are used to specify the combustion in CEBAM explosion calculations, Clutter and Luckritz (1999). Table 1 shows the flame Mach numbers of Baker *et al*. (1998). A series of articles have been published in which it is described how these curves were generated, e.g. Tang and Baker (1998), Tang and Baker (1999) and Tang and Baker (2000). The flame Mach number is not assigned a value solely based on the tabulated data from the papers by Baker *et al.* (1998). The assignment of the flame Mach number also draws upon the experience of the user of the code. This makes it difficult to audit the process. There is also an increased risk that the results could become highly user dependent.

Table 1 Flame Mach number as a function of flame dimensionality, fuel reactivity and level of congestion, Baker, Doolittle, Fitzgerald, and Tang (1998)

Flame Expansion	***Fuel Reactivity***	***Obstacle Density***		
		High	***Medium***	***Low***
1D	High	5.2	5.2	5.2
	Medium	2.27	1.77	1.03
	Low	2.27	1.03	0.294
2D	High	1.77	1.03	0.588
	Medium	1.24	0.662	0.118
	Low	0.662	0.471	0.079
3D	High	0.588	0.153	0.071
	Medium	0.206	0.100	0.037
	Low	0.147	0.100	0.037

The user can either let the CEBAM code choose the flame speed in the congested regions, using the empirical correlations or use his/her own judgment when assigning a flame speed. It was suggested in discussions, Clutter (2002), that one would possibly not look at each individual congested region, but rather consider the process module in its entirety. It is not immediately obvious how a suitable characteristic flame speed can be assigned for an entire process module, which will always give a conservative answer, unless a very high flame speed is prescribed.

Two different flame speed models appear to have been implemented in the CEBAM code – a constant flame speed model and a decelerating flame speed model.

The constant flame speed model, Clutter and Luckritz (1999), was apparently implemented only for the purpose of testing whether the flame speed model would yield results in line with Luckritz's model, Luckritz (1977). The constant flame speed model has an effective reaction rate, which is defined as $k_{eff} = 1/\tau_{C,}$ where τ_C is the chemical time scale related to the flame speed in a congested region. The chemical timescale is very simply calculated as the cell size divided by the assigned flame speed, e.g. it is not related to the reaction kinetics.

The Decelerating Flame Speed Model (DFSM) allows the flame to slow down in a linear fashion when the flame leaves a congested area. The local reaction rate, in a congested region, is based on the assigned flame speed in CEBAM. In CEBAM, the assigned flame speed is a truly local parameter, e.g. the flow retains no memory of the flame speed in a neighbouring congested region, i.e. a flame speed attained in a congested region is not used in its congested region neighbours. Figure 1 shows a scenario with two congested regions. In Figure 1, the hatching of the regions is used for illustrative purposes only in order to represent the level of congestion, with region 2 being more congested than region 1 and thus $M_{flame,2} > M_{flame,1}$.

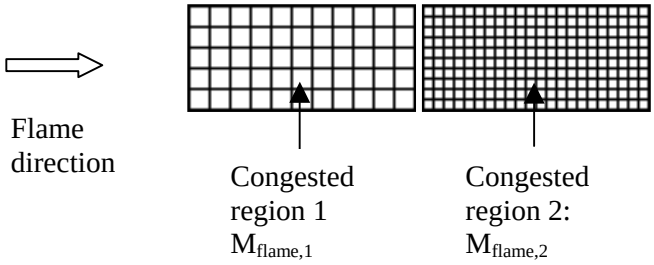

Figure 1 Schematic diagram showing two regions with different levels of congestion.

In a real scenario the flame will be accelerated in the first of the congested regions. The flame speed is then further increased as the flame moves through the second congested region. Thus the flame speed in a real scenario can be much higher than if only local flame speeds are prescribed as in CEBAM methodology. Higher flame speeds will lead to higher explosion generated over-pressures. The CEBAM approach could therefore lead to a non-conservative estimation of the over-pressure, unless one selects a very high flame speed – it is not obvious how high the flame speed needs to be for the resultant predicted over-pressure to be conservative. In CEBAM the flame will decelerate if the separation distance between the congested regions is large – unless an imaginary congested region is defined between the two actual congested regions.

2.4 NUMERICAL AND MESHING FEATURES

2.4.1 Computational grid

It appears that the CEBAM model works with structured, Cartesian meshes. The cell size is of the order of 1 m. The appropriate cell size to represent, say, 2" pipes, would be three order of magnitude small than the cell size used in CEBAM. The user can improve the grid resolution in order to resolve large objects, such as process vessels. However, it is not practically possible with any CFD model to resolve all obstacles that can be found in an offshore process module. A technique for modelling sub-grid scale structures has therefore been implemented, Clutter (2001).

The effect of sub-grid scale obstacles on flame behaviour is lumped into the *user-prescribed* flame speed described in Section 2.3. The sub-grid scale modelling for momentum/pressure loss

is based on a technique proposed by Ingram, Batten, Causon and Saunders (1997): The pressure loss calculated from the sub-grid scale model is due to two mechanisms, a) the force exerted by the fluid on the obstacles and b) friction.

The size of the cells nearest solid walls is not that critical in CEBAM, since boundary layers and near-wall turbulence are ignored in the code when modelling gas explosions.

No grid dependency study appears to have been carried out as part of the code validation. Clutter (2003) provided information about the results of a grid dependency exercise that had been undertaken for a project; the evidence was only verbal due to the classified nature of the project.

2.4.2 Discretisation and solution schemes

Clutter and Shyy (1998) describe the procedure by which the equations are solved. The inviscid numerical fluxes are modelled using the Steger-Warming vector splitting technique. A second-order accurate upwind discretisation scheme, based on the MUSCL approach, has been implemented in the code. This Total Variation Diminishing (TVD) scheme introduces limiters to ensure that non-physical extrema are not introduced. TVD approaches are often used to model flows in which there are shock waves and other steep gradients. Further details of TVD schemes can be found in Roe (1981), van Leer (1974, 1982). Fletcher (1988a, 1988b), Hirsch (1989a, 1989b) and Versteeg and Malalasekera (1995) provide more general details on equation discretisation.

The source terms are treated in an implicit fashion while the other terms in the equations are treated explicitly. A predictor-corrector solution method ensures that the scheme remains second-order accurate in space. Clutter and Shyy (1998) compared the accuracy of the explicit-implicit method to a fully implicit approach and found that the former technique was as accurate as the latter.

The numerical approach appears to stem from the original application area for the code, namely shock wave propagation. It should also be suitable for computing gas explosions.

2.5 VALIDATION

Published validation of the CEBAM code is predominantly from small-scale and medium-scale experiments carried out in the CEC sponsored EMERGE project, Mercx, Popat and Linga (1997), and the experiments in the large-scale offshore module used in Phase 2 of the Blast and Fire Engineering for Topside Structures Joint Industry Project, Selby and Burgan (1998).

2.5.1 EMERGE experiments

The EMERGE project was sponsored by EU through a Joint Industry Project as a follow-up to MERGE. The partners involved in the project included large European oil and gas companies and European universities.

The objectives of the MERGE project were to study the effects of scaling, confinement and initial turbulence on the explosion generated over-pressure. The EMERGE project involved experiments in small-scale to medium-scale geometries that were congested and confined. The congestion in the rigs consisted of a large number of pipes, which were placed in an ordered grid-like arrangement with rows of pipes orientated in both the vertical and horizontal direction.

Clutter (2001) presented results of calculations of some of the EMERGE experiments.

2.5.1.1 Methane

Two different flame speeds were used in the calculations of a small-scale rig containing a stoichiometric mixture of methane and air. A flame Mach number of 0.32 was used for the higher velocity cases, i.e. for velocities of around 100 m s^{-1} or higher, while a flame Mach number of 0.17 was used in the lower velocity cases. The predicted radial variation in the non-dimensionalised over-pressure is in reasonably good agreement with the experiments. Though it is interesting that one has to choose two different flame Mach numbers. Clutter (2003) explained that two different flame mach numbers were used to reflect the differences in the congestion between the two experimental set-ups; the calculations were carried out to see that the flame Mach number had been set correctly and that the correct pressure field was produced.

Calculations of the explosion of a stoichiometric mixture of methane and air in the medium-scale EMERGE rig required either a flame Mach number of 0.265 for the lower velocity cases and 0.72 for the high velocity cases, e.g. when the velocity is of the order of 250 m s^{-1}.

The CEBAM results were in reasonable agreement with the experiments. It would appear that the code gives conservative over-pressures for some of the cases, while under-predicting the over-pressure in others. It is difficult to say by exactly how much the predicted over-pressures differ from the measurements due to the scaling of the graph. It would appear that the predictions agree with the measurements to within a factor of two, at worst, but in many cases the agreement is better than that. The agreement between the CEBAM calculations and the experiments would thus be on a par with the other CFD codes for explosion calculations, e.g. AutoReaGas, EXSIM and FLACS.

2.5.1.2 Propane

Two different flame Mach numbers were used to calculate the over-pressure in the small-scale EMERGE rig. The lower flame Mach number of 0.18 was used for the lower velocity cases, e.g. when the velocity was less than 100 m s^{-1}, while a flame Mach number of 0.6 was used for the higher velocity cases.

Calculations of the explosion of a stoichiometric propane/air mixture in the medium-scale EMERGE rig were also carried out. As in the small-scale rig calculations, two different flame Mach numbers were used depending on the flow velocity. A flame Mach number of 0.34 gave a reasonable agreement between the predictions and the measurements. A considerably higher flame Mach number of 1.14 was used to get reasonable agreement between the calculated and measured over-pressures for the high velocity cases, where the velocity was of the order of 400 m s^{-1}. Though it is again interesting that one has to choose two different flame Mach numbers. It would again appear that this might be a calibration exercise; Clutter (2001) does not clearly state how these flame Mach numbers were chosen – it is possible that the empirical correlations found in Baker *et al.* (1996) are used, but this is not clear from the text.

The same trend was observed in the propane calculations as in the methane calculations – the CEBAM results were in reasonable agreement with the experiments. Sometimes the code would give conservative over-pressures and at other times the over-pressure would be under-predicted compared to the pressure that was actually measured in the experiments. It is difficult to say by exactly how much the predicted over-pressures differ from the measurements due to the scaling of the graph. It would again appear that the predictions agree with the measurements to within a factor of two, at worst, but in many cases the agreement is better than that. The agreement between the CEBAM calculations and the experiments would thus again be on a par with those of other CFD codes for explosion calculations, e.g. AutoReaGas, EXSIM and FLACS.

2.5.1.3 BFETS – Phase 2 experiments

A large, multi-million Joint Industry Project, Blast and Fire Engineering for Topsides Structures (BFETS), was initiated in the aftermath of the Piper Alpha disaster on 7th July 1988, where 167 workers lost their lives. The BFETS project involved the HSE and all the major oil and gas companies operating on the UK Continental Shelf. The project consisted of two phases, which involved both conducting large-scale experiments and CFD simulations, but was later extended with a third phase:

- Phase 1
  - Assessment of the technical understanding of gas explosions
  - Assessment of the state of the computer models
  - Identification of experiments involving gas explosions
- Phase 2
  - Large-scale experiments in a mock-up of an offshore process module, with a typical congestion and confinement, using natural gas and propane
  - Modelling of tests where the experimental data was already available to the modellers
  - Modelling of tests prior to the experimental data being available, so-called blind predictions
  - Refinement of the physical sub-models in the CFD codes, i.e. calibration of model constants to the large-scale experiments
  - Modelling of the scenarios, for which blind calculations had been performed, with the recalibrated models
- Phase 3
  - Experiments where mitigation measures were implemented
  - Experiments with Partial fills of process modules

The locations at which the pressure sensors were located during the tests appear not to have been presented in the final summary report, Selby and Burgan (1998), of Phase 2 of the BFETS project – upon which the CEBAM validation is based. This has had the unfortunate effect that the comparisons between over-pressures measured in the experiments and those predicted by the code cannot be easily compared, since the location of a pressure sensor can clearly be crucial to the pressure it registers.

The CEBAM code could not correctly predict the explosion-generated over-pressure for one particular large-scale experiment from the BFETS experimental campaign. This was in part due to the fact that the model did not pick up the external explosion, which took place in the experiment and contributed greatly to the overall over-pressure. It was therefore necessary to artificially prescribe a congested region with its associated flame speed outside the module. The choice of flame speed in this region, which is not even congested, seems rather arbitrary. It seems reasonable if one already knows the answer, but there may be situations where the choice of flame speed does not lead to a conservative estimation.

Clutter and Mathis (2002) present a number of comparisons between calculated and measured pressure traces for the BFETS Phase 2 experiments. The code sometimes under-predicts the over-pressure, by a factor of two, while at other times the calculated over-pressure is in excess of the measurements.

The code is not able to explicitly model external explosions, e.g. the explosion occurs outside the process module itself and is due to the fact that combustible gases gets pushed out of the module. The external explosion can contribute significantly to the total over-pressure. The CEBAM way around this is to specify an imaginary congested region, with its associated flame

speed, outside the module, Clutter and Mathis (2002). The CEBAM calculations gave reasonable agreement with the measurements in terms of time of pressure rise for some of the test cases.

2.6 OUTSTANDING AREAS OF CONCERN

A meeting with Dr Clutter was arranged to be able to hear about and ask questions about the code and the models implemented therein.

The choice of flame speed within a module is user-dependent. It is therefore quite possible that different users of the code could calculate the same test case and come up with significantly different answers. It appears that Analytical and Computational Engineering, who develop the CEBAM code, could be initiating an exercise where a number of different consultants get to calculate a problem, to ascertain how dependent the solution is on the skill and knowledge of the user. It is not clear whether such an exercise has been undertaken and what the results were. The potential variability in calculated over-pressures due to the seemingly arbitrary way in which the flame speed is decided is of concern. If too low a flame speed is chosen, then over-pressure could be significantly under-predicted.

The code developers train and guide the current users of the CEBAM code according to Dr Clutter, for example advising on the choice of flame speed. The code developers therefore act essentially as consultants to those carrying out the modelling work. It is clear that Dr Clutter has a thorough understanding of the code and what it does. Dr Clutter also has experience in assessing the level of congestion in a region/module and assigning a suitable flame speed. Dr Clutter recommends that if there is uncertainty on this parameter then a conservative approach should be taken – with selection of high flame speeds. There can be no certainty that the predicted over-pressures are conservative for all scenarios.

However as CEBAM becomes used more extensively – and it is being strongly marketed in the UK, it could become possible that inexperienced users may apply the code inappropriately: by choosing too low a flame speed; by assigning a very low flame speed in uncongested regions even though in reality the flame will travel rapidly through such regions due to its prior history; omitting to assign imaginary congested regions beyond a module to account for the external explosion. It is not clear whether best practice guidance is available and issued to new users. Dr Clutter said that the code is intended to be used by trained personnel at consultancy firms, who will seek advice from Analytical and Computational Engineering.

It is also clear that it is difficult for the regulator to assess the adequacy of results obtained using CEBAM, since knowledge of the flame speeds set by the user and an appreciation of the likely flame speeds, which are attainable, is required.

3 DISCUSSION AND CONCLUSIONS

The CEBAM code has its origin in the modelling of combustion problems in the field of aero-propulsion. A solid explosives version of CEBAM also exists. It is sufficient to solve only the Euler equations, when modelling solid explosives. The full Navier-Stokes equations and turbulence models have now been implemented in the code. However, it would appear that the gas explosions are modelled solving just the Euler equations, where viscous effects such as diffusion have been neglected.

However, the code developer recommends that the Euler version of the code be used in the calculations of gas explosions, even though the influence of turbulence on the burning velocity cannot therefore be taken into account explicitly.

For explosions, the effect of turbulence on the flame burning velocity is taken into account by the user assigning a flame Mach number, which is local to the region in which it has been specified. In practice the flame Mach number is dependent on the fuel type, physical dimension of the domain/region, level of congestion and level of confinement. There is therefore a risk that the calculated over-pressure may be user dependent, and potentially non-conservative if an inappropriate choice of flame speed is made.

The CEBAM code takes a different approach to explosion modelling than all other CFD codes in use. The CEBAM code solves only the Euler equations: the effect of turbulence on the burning velocity is incorporated in a flame Mach number which the user has to choose judiciously – taking into account type of fuel and levels of congestion and confinement. In other CFD explosion codes these effects are computed, and therefore do not rely as critically on the choice of the user.

This does not imply that these other CFD codes are without their limitations. Thus a combination of a rather coarse grid resolution and the relatively simple k-ε turbulence model means that it is most unlikely that these codes can resolve turbulence generation around large obstacles. Small obstacles are typically treated using sub-grid models, in which the problem of turbulence-combustion interactions is greatly simplified. What must be inherent shortcomings in these codes are however compensated to some extent by their having been calibrated against experiments at a range of scales.

The CEBAM code is not able to explicitly model external explosions, e.g. the explosion occurs outside the process module itself and is due to the fact that combustible gases get pushed out of the module. The external explosion can contribute significantly to the total over-pressure. The CEBAM way around this is to specify an imaginary congested region, with its associated flame speed, outside the module. However there is a risk with this approach that the calculated over-pressures are not conservative.

Explosion modelling using CEBAM is thus probably more dependent on the expertise and judgement of the user than with other CFD codes. This increases the importance of training and support for users, the desirability of guidelines for users of the code, and the information that must be sought and assessed by the regulator in assessing CEBAM-based safety submissions. CEBAM is being actively marketed with the UK. It is already being used in support of offshore safety cases. However, the CEBAM code was intended to be used during the concept design phase of a project, when the layout is frequently changed and a more in-depth and computer intensive study is not feasible or practical. It is recommended that HSE maintains/develops its in-house experience in the use of CEBAM to better understand its behaviour and to inform the assessment of safety submissions based on the code.

4 REFERENCES

Baker, Q. A., Tang, M. J., Scheier, E. A., and Silva, G. J. (1996)
Vapour Cloud Explosion Analysis
Process Safety Progress **15**(2):106-109

Baker, Q. A., Doolittle, C. M., Fitzgerald, G. A., and Tang, M. J. (1998)
Recent Developments in the Baker-Strehlow VCE Analysis Methodology
Process Safety Progress **17**(4):297-301

Clutter, J. K. (1999)
Use of a reduced combustion model in industrial explosion simulations
AIAA paper no. **99-0463**, *37th AIAA Aerospace Sciences Meeting*

Clutter, J. K. (2001)
A reduced combustion model for vapor cloud explosions validated against full-scale data
Journal of Loss Prevention in the Process Industries **14**:181-192

Clutter, J. K. (2002)
Private communication

Clutter, J. K. (2003)
Private communication

Clutter, J. K., and Luckritz, R. T. (1999)
Comparison of blast wave curve and a reduced explosion model to experimental results
Journal of Hazardous Materials **79**(1/2):41-46

Clutter, J. K., and Mathis, J. (2002)
Computational modelling of vapor cloud explosions in off-shore rigs using a flame-speed based combustion model
Journal of Loss Prevention in the Process Industries **15**:391-401

Clutter, J. K., and Shyy, W. (1998)
Numerical Methods for Treating Disparate Scales in High-Speed Reacting Flows
Numerical Heat Transfer Part B **34**:165-189

Clutter, J. K., and Whitney, M. G. (2001)
Use of computational modelling to identify the cause of vapor cloud explosion incidents
Journal of Loss Prevention in the Process Industries **14**:337-347

Fletcher, C. A. (1988a)
Computational Techniques for Fluid Dynamics, vol I: Fundamental and General Techniques
Springer Verlag, Berlin, Germany.

Fletcher, C. A. (1988b)
Computational Techniques for Fluid Dynamics, vol II: Specific Techniques for Different Flow Categories
Springer Verlag, Berlin, Germany.

Hirsch, C. (1989a)
Numerical Computation of Internal and External Flows 1: *Fundamentals of Numerical Discretisation*
Wiley & Sons.

Hirsch, C. (1989b)
Numerical Computation of Internal and External Flows 2: *Computational Methods for Inviscid and Viscous Flows*
Wiley & Sons.

Ingram, D. M., Batten, P., Causon, D. M., and Saunders, R. (1997)
The Representation of Small Scale Obstructions in Hydrocode Calculations
Flow and Dispersion Through Groups of Obstacles, Eds: Perkins, R. J., and Belcher, S. E., Clarendon Press, Oxford, UK

Ledin, H. S. (2002)
A Review of State-of-the-Art in Gas Explosion Modelling
Health and Safety Laboratory Report No. **HSL/2002/02**

Leer, B. van (1974)
Towards the Ultimate Conservative Difference Scheme. II. Monotonicity and Conservation Combined in a Second-Order Scheme
Journal of Computational Physics **14**:361-370.

Leer, B. van (1982)
Flux Vector Splitting for the Euler Equations
Lecture Notes in Physics, Springer-Verlag, **170**:507-512.

Luckritz, R. T. (1977)
An investigation of blast waves generated by constant velocity flames
PhD Dissertation, University of Maryland

Mercx, W. P. M., Popat, N. R., and Linga, H. (1997)
Experiments to investigate the influence of an initial turbulence field on the explosion process
Final Summary report for EMERGE, Commission of the European Communities Contract EV5V-CT93-0274

Roe, P. L. (1981)
Approximate Riemann Solvers, Parameter Vectors, and Difference Schemes
Journal of Computational Physics **43**:357-372.

Selby, C. A., and Burgan, B. A. (1998)
Blast and fire Engineering for Topside Structures – Phase 2: Final Summary report
SCI Publication No. **253**, Steel Construction Institute, Ascot, UK

Tang, M. J., and Baker, Q. A. (1998)
Comparison of blast curves from vapor cloud explosions
Proceedings of the International Symposium on Hazards, Prevention and Mitigation of Industrial Explosions, Schaumburg, Illinois, USA

Tang, M. J., and Baker, Q. A. (1999)
A New Set of Blast Curves from Vapor Cloud Explosion
Process Safety Progress **18**(3):235-240

Tang, M. J., and Baker, Q. A. (2000)
Comparison of blast curves from vapor cloud explosions
Journal of Loss Prevention in the Process Industries **13**:433-438

Versteeg, H.K., and Malalasekera, W. M. G. (1995)
An Introduction to Computational Fluid Dynamics: The Finite Volume Method