

DISPERSION OF FUEL IN OFFSHORE MODULES: Comparison of Predictions Using FLACS and Full Scale Experiments

**Chris Savvides & Vincent Tam, BP Exploration, Sunbury on Thames
Jon Erik Os, Olav Hansen, Kees van Wingerden & Jerome Renault, GexCon, Norway**

AUTHOR BIOGRAPHICAL NOTES

Olav Roald Hansen is R&D Director responsible for research and development within GexCon. This includes the development and validation of the FLACS-simulator. When he joined GexCon end of 2000, he had 8 years of experience in the field of explosion safety at Christian Michelsen Research. He is currently project manager for two research programmes within explosion safety for onshore/offshore installations and in hydroelectric power plants.

Jerome Renault has an M.Sc. degree from the French Institute of Mechanical Advance and has been with GexCon since 1998. Over the last three years he has been working on consultancy projects and research programmes.

Dr Kees van Wingerden is the general manager of GexCon, a consultancy in the field of explosion safety. He has been involved in safety work for the last 20 years. He was director of FLACS development from 1991 to 1999. He is a member of many international working groups and author and co-author of more than 50 articles on gas and dust explosions.

Dr Chris Savvides is a Senior Safety Engineer, within the Upstream Technology Group of BP Exploration. He has been involved in the application of CFD software in Gas Dispersion and Explosions Offshore for the last 15 years.

Dr Vincent Tam has been working in safety technology and consequence modelling since 1981. Over the past 10 years, has directed and carried out gas explosion research projects and has been working closely with BP design projects in advising on gas explosion control and mitigation

ABSTRACT

There is a need for operators of offshore installations to make realistic estimates of the risk associated with the accidental release of gas at high pressure. Accurate predictions allow operators to quantify the risks and thereby manage them. Predictions also allow the benefits of mitigation measures to be quantified. Over the years a variety of models had been used to predict gas dispersion and accumulation on offshore installations, varying from simple empirical correlations to computational fluid dynamics (CFD) models. It is essential that any model used in these predictions, have been fully validated. This paper presents a comparison of the predictions of the CFD software FLACS with the large-scale gas build-up experiments carried out in 1997-1998.

1. INTRODUCTION

There a number of methods used over recent years to provide estimates of flammable gas buildup due to an accidental release. The results of these models are then used to quantify the risk from a gas explosion. Over the last 20 years BP has been sponsoring (within the Gas Safety Programme consortium) the development of FLACS, Computational Fluid Dynamic (CFD) software, initially for gas explosion application and later enhanced to include gas dispersion.

In this paper we review the general approach used for gas dispersion simulations with FLACS. The application of this approach to the naturally ventilated offshore module used in the experimental measurement of gas buildup as part of the Joint Industry Project “Gas Buildup From High Pressure Natural Gas Releases in Naturally Ventilated Offshore Modules”^{1, 2} is discussed. The results of the FLACS simulations are compared with the experimental data.

2. OVERVIEW OF FLACS

FLACS has been in continuous development since 1980 supported by various major oil and gas companies at the Christian Michelsen Research. Recently this activity has been transferred to GexCon.

FLACS uses a 3D Cartesian implicit Navier-Stokes solver based on the Simple method by Patankar with a compressible extension. Implicit/explicit 2nd order terms are implemented for increased accuracy. Flow equations are closed with a k- ϵ model for turbulence. Models for wall function and turbulence generation from sub-grid geometry are required and implemented. A distributed porosity concept is used for mapping the complicated geometry (either defined in pre-processor or imported from CAD-system) onto the simulation mesh. Details of the numerical methods and physical models employed in FLACS can be found in reference 3.

The pressurised jet is modelled as an almost sonic (typically 250 m/s) release at ambient pressure, with a given mass flow (or velocity).

During a transient dispersion calculation, several fuel specific parameters relevant for an explosion risk assessment analysis are reported, for instance the amount of fuel between flammability limits, the estimated equivalent stoichiometric gas cloud, as well as volume of fuel that is flammable in each time interval. The two latter parameters are very useful for transient ignition modelling.

Typical of commercial CFD codes, the approach is to construct geometry and mesh to represent the flow domain of interest, the flow boundary conditions are then specified and a solution to the discretized equations obtained using a numerical solution method appropriate to the problem. The solution data is then analysed to obtain the results of importance including flow rates, gas volumes and visualisation of the flow patterns and other solution variables.

3. APPLICATION OF FLACS TO GAS DISPERSION ASSESSMENT

In applying a general CFD code to a specific analysis it is important to ensure that the most appropriate methods and approaches are used to achieve the desired consistency and accuracy of results. In this respect, the simulations performed for analysis of BP assets are undertaken based upon guidelines developed over recent years (reference 4)

The approach used in FLACS simulations is based upon explicitly incorporating as much geometry as is practical, generally into representative cylinders and boxes. In the case of the JIP module all the equipment has been incorporated explicitly with the exception of the grated deck, which is modeled as a 3.5cm thick plate with an 82% vertical porosity. Figure 1 shows the geometry of the CFD module used in the FLACS simulations.

When specifying the geometry it is also necessary to ensure that the extents of the domain size and shape surrounding the module of interest do not influence the predicted flow patterns. It is important to leave sufficient distance between the domain boundaries (inlet, outlet, sides and top) and the obstruction, or region of interest, within the flow.

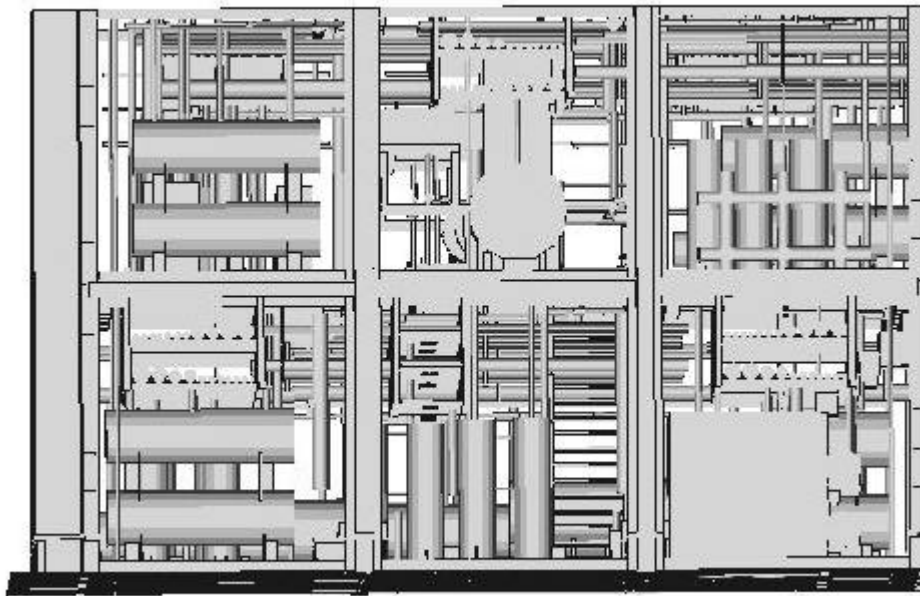


FIGURE 1: FLACS Geometry of the JIP Module

4. JIP MODULE SIMULATIONS: TYPICAL SIMULATION

4.1. Overview of JIP Module Experiments

The JIP gas dispersion experiments consisted of 66 scenarios with variations to the perimeter confinement, the release flow rate, location and direction and a variety of wind speeds and directions as dictated by the ambient conditions. The geometric configurations are illustrated in Figure 2. A series has open ends with parallel walls, B series is similar but with louvre's located on the ends to provide a partial blockage and C series uses an L shaped wall.

Large, medium and small gas releases were simulated from 3 locations and directions (North, South, East, West and Up and Down). Depending on the wind direction some of these releases were aligned or opposed to the natural ventilation flow in the module and some of the gas jets were impinging on adjacent equipment or the module walls.

Measurements were made over a reasonably long time scale and averaged results are reported in reference 1. Wind speed and direction were made using a sonic anemometer located upwind of the module. The module was instrumented to measure the gas concentration at

192 locations on a 6 by 8 by 4 array. Other measurements included the gas release flow rate and the pre-release and during-release module ventilation rates. Further details of the experiments and scenario's investigated can be found in reference 1 and 2.

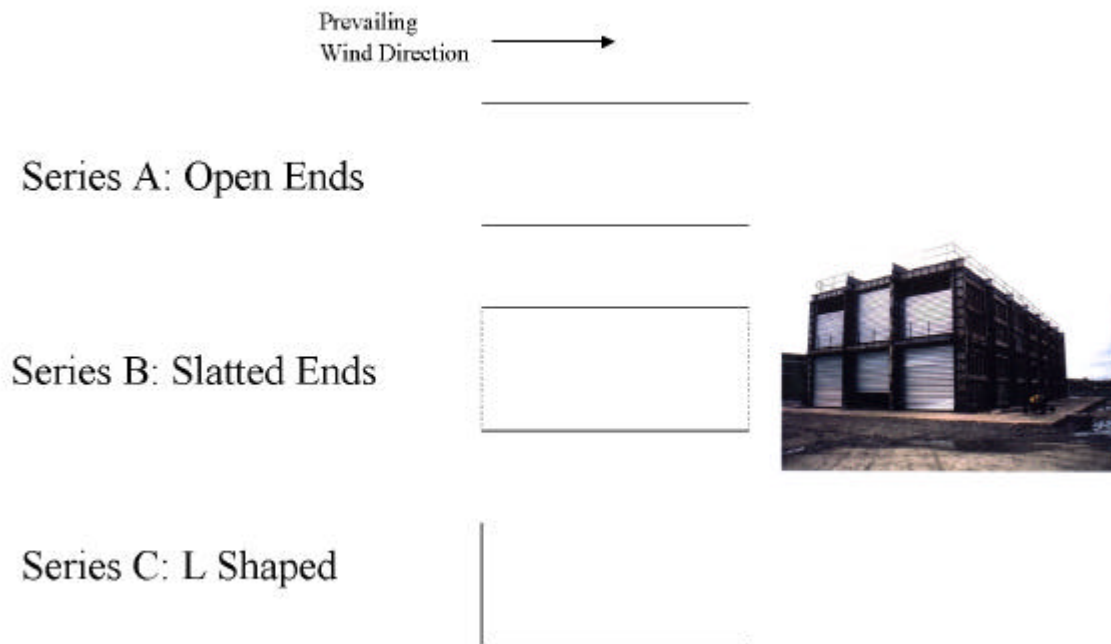


FIGURE 2: Definition of Configuration for A, B and C series tests

4.2. Overview of Typical Simulation Results

An individual FLACS simulation can provide a vast amount of very detailed information regarding the flow and other variables. Only a brief analysis of the results has been undertaken, as the prime purpose of this work is to compare the volumes of the predicted gas cloud with the measured data. Figure 3 show some flow vectors within and around the module and the predicted gas cloud is shown in Figure 4. Numerical data is also derived from the simulation results. Specifically the gas cloud volumes at half LFL, LFL, and UFL are obtained by integrating the volumes of those cells within the relevant concentration ranges. The ventilation rates for the module were also obtained as well as other data used to check the setup of the module and convergence.

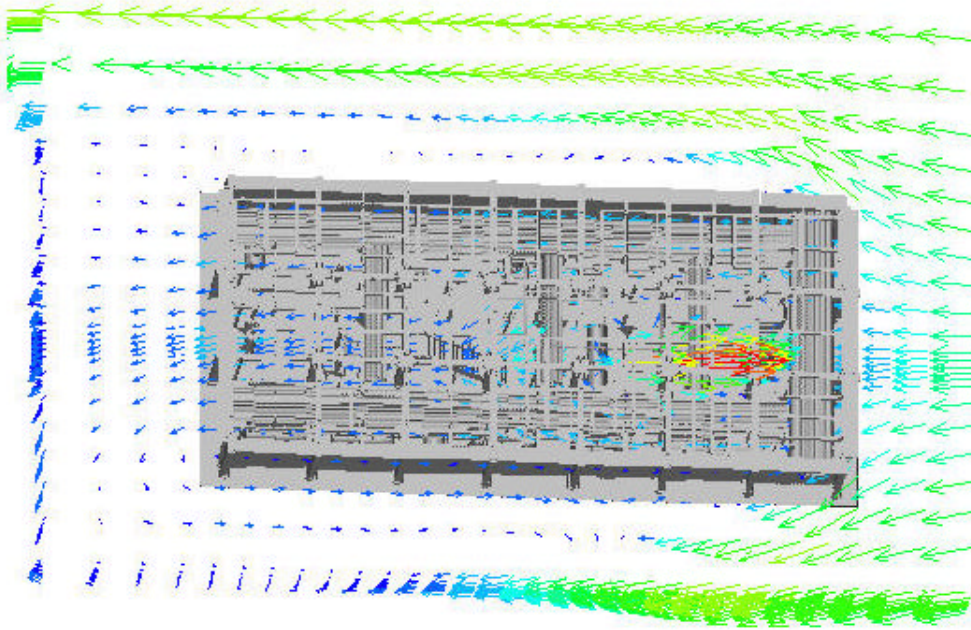


FIGURE 3: Simulated Velocity Vectors at a height of 5m for A19

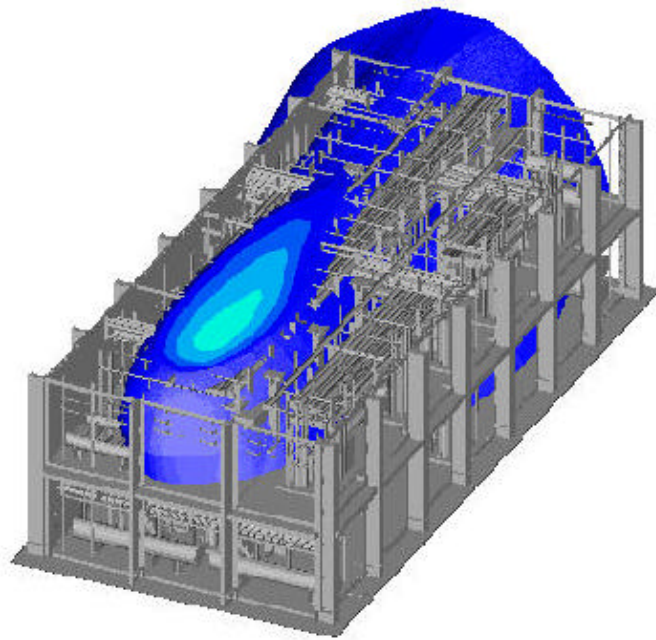


FIGURE 4: Simulated Gas Cloud for experiment A19

5. JIP MODULE SIMULATIONS: COMPARISON OF FLACS RESULTS AND MEASURED DATA

Only comparisons of the predicted and observed volumes above a predefined concentration will be presented here. The concentration chosen is 5% (Natural Gas lower flammable limit). Also examined in this section is the effect of the environmental conditions (wind speed and direction) that would have affected the experimental data.

5.1. Gas Cloud Volume Comparison

All the results are summarized in Figure 5. This shows the experimental gas cloud volumes (concentration >5%) plotted against the FLACS predictions. It can be seen that almost all predictions fall within 30% of the observed data.

Another way of measuring the performance of the model is to plot the geometric mean bias (MG) with the geometric mean variance (VG). Figure 6 shows such a plot. Overall the predictions are quite good with a Geometric mean of 1.25 and a Geometric variance of 1.16 (>5%). For concentrations between 5-15% the Geometric mean improves to 1.02 but the Geometric variance increases to 1.32. In these results no account was taken of the variability of the experimental data due to the environmental conditions.

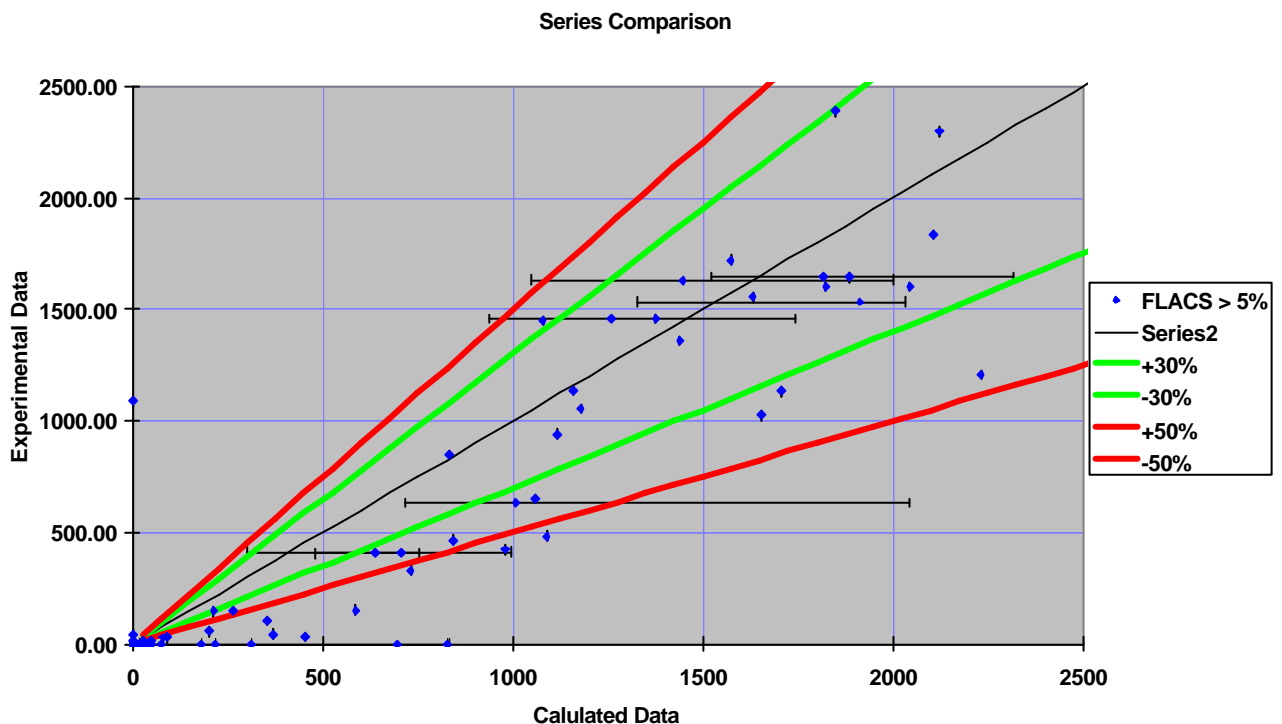


FIGURE 5: Predicted Vs Observed Gas Cloud Volume

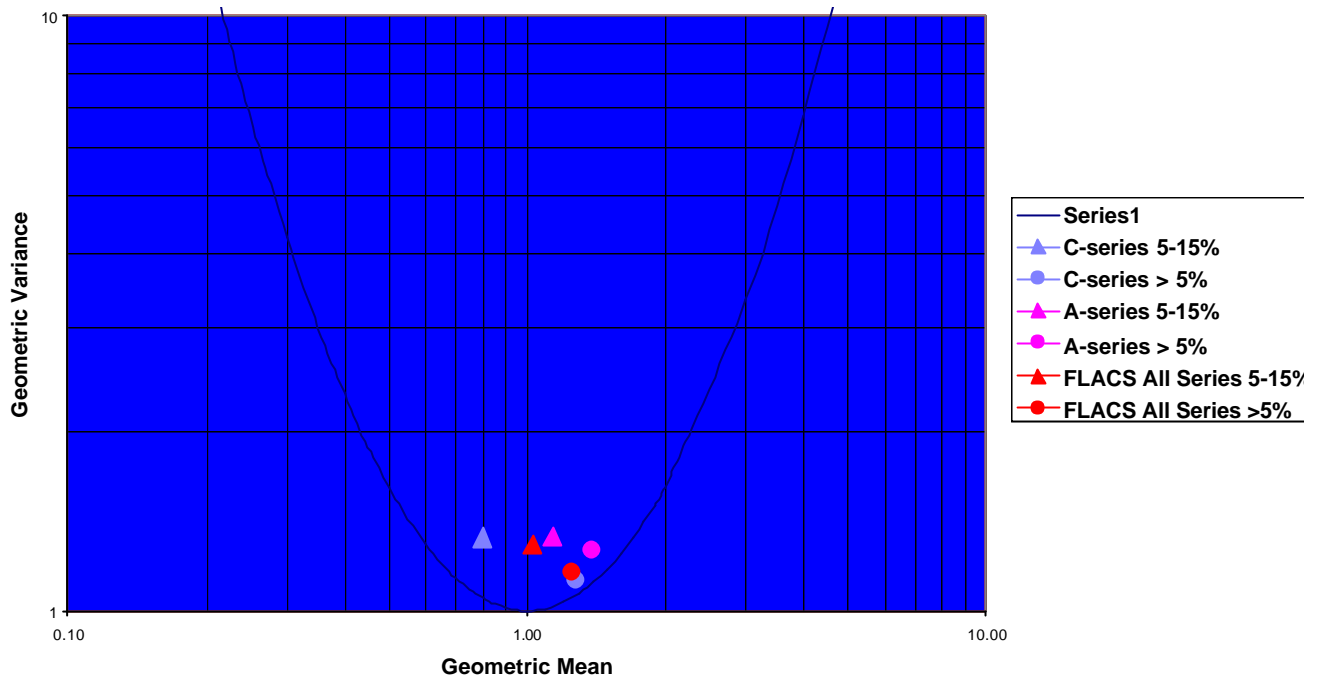


FIGURE 6: Model Performance – Gas Cloud Volume

5.2. Sensitivity of FLACS predictions to Wind Speed and Direction

The predictions take a specific constant wind direction and speed whereas in reality these parameters are variable due to the ambient conditions. To assess the sensitivity of the predictions to these variables additional simulations have been undertaken for extreme values of wind speed and direction. The spread of some of the results are shown in Figure 5

It can be seen that changes in the environmental conditions would have a marked effect on the gas cloud built-up. The wind speed would have a greater effect on the A and B series results (through flow), while the wind direction would have a greater effect on the C series results (L-shape). The wind speed and direction effect for each experiment varies but typically would be between 10-30%.

6. CONCLUSIONS

A large number of experiments in different configurations and with varying wind and leakage conditions have been model successfully using the CFD software FLACS. The current validation exercise has shown that FLACS can predict with good accuracy the dispersion and gas accumulation from a release in an offshore module. In particular the predicted gas volume as defined by the LFL concentration agrees well with the measured data.

7. REFERENCES

- 1 *Gas Buildup from High Pressure Natural Gas Releases in Naturally Ventilated Offshore Modules*, Joint Industry Project Large Scale Experiments, April 1999, BG Technology.
- 2 *Gas Buildup from High Pressure Natural Gas Releases in Naturally Ventilated Offshore Modules*, Joint Industry Project Technical Report, May 1999, BP Amoco, BG Technology, Shell Global Solutions.
- 3 FLACS User Manuals, GexCon, Norway, 1999
- 4 FLACS Gas Dispersion and Explosion Modelling Operations Manual, BP Internal Document, 2001.

Copyright ERA, BP and GexCon