

A comparison between big scale experiments and calculations on fire exposing an object.

A case of multi physics simulation.

Dr. Geir Berge, Petrell as
Dr. Arve Klavenes, Petrell as

Abstract

Multi physics simulation is a term introduced when multiple physical models are interacting simultaneously in a single simulation. New technology is introduced to treat this kind of problem. An object-oriented based simulation system, Brilliant, is used to build the multi physics case.

In this paper an example of a multi physics simulation is outlined, and a specific case is demonstrated. The example treats process equipment exposed to fire, and the case demonstrated is a flame exposing a box where both the flame and the temperature distribution of the box are simulated simultaneously. This specific case is chosen because well-documented experimental data are available.

The simulation and the experiments give results that agree reasonably well.

Multi physics simulation

The oil and gas, metallurgical and process industry is a typical area for multi physics problems. A typical example is process equipment exposed to fire. This is an important safety issue that has to be included in the design of the installation. An example is illustrated in Figure 1.

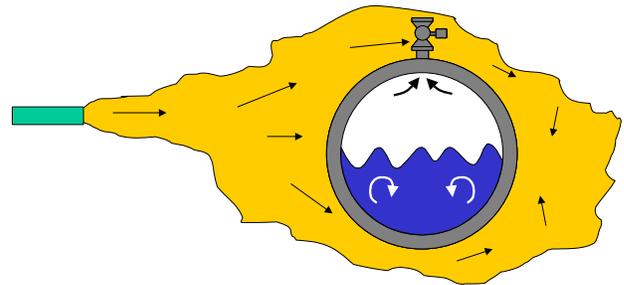


Figure 1 Principle illustration of a typical multi physics problem. A fire is exposing a vessel.

The scenario can be described as a leak from a process segment with early ignition resulting in a jet fire. The fire is hitting a vessel (or part of a segment) containing hydrocarbons and is exposing the vessel with thermal radiation and convective heat. The vessel shell is heated, and the temperature of the inventory is starting to increase. This means increased temperature in both the gas and the liquid phase of the inventory. The liquid will evaporate, and the pressure increases as a result of the evaporation and the increasing gas temperature. If the vessel is equipped with a blowdown valve, a blowdown system can be activated and gas can be released to reduce the pressure to a safe level.

The situation described is quite complex and includes several physical processes. A safe blowdown of such a fire scenario is dependent on a well-designed system. For that reason multi physics simulations are

necessary. Lately, this kind of problem has been treated with a system called VessFire that simulates the whole process except the fire. More details on this are described in [6], [7] and [8]. To also include the fire in the process, new technology has to be introduced.

Some features of the simulation system

Brilliant is a new code designed using object oriented technology. The system was designed to meet four main specifications:

- Full freedom to shape, size and locate control volumes in the calculation domain without using sub-grid or other traditional grid refinement techniques.
- Freedom to add and remove control volumes during simulation.
- The possibility to use multiple physical models simultaneously.
- The possibility to change models during simulation.

To meet these specifications a fundamentally different organisation of the code, compared to traditional codes, had to be developed. The structure known from the FORTRAN type of codes with the use of arrays and indexing is abandoned. Instead, every control volume is a self-contained object and linked together in lists. The control volumes can communicate with each other and are therefore able to treat complex connections in a relatively simple manner.

The administration of the system is kept at a minimum and is even less than traditionally organised multipurpose codes.

The system is based on a well-known finite volume discretization technique. A good review of the method is found in [1] and [2]. The system is independent of the solution scheme, and different solution schemes can be applied within the same frame. In the present version the SIMPLEC method is used. The method is implemented on a time dependent and compressible form.

Brilliant is using centrally located velocity components and Rhie & Chow interpolation [4] to calculate the convective terms at the cell boundaries.

Brilliant is not bound to use a special solution scheme.

The different models available today are:

- Conduction in solid materials. A single object can consist of a combination of different materials.
- Laminar and turbulent flow. Presently, the k- ϵ model is implemented.
- Gas dispersion. Sources with different locations and release times can be applied. The composition of each source can vary and so can the release profile.
- Radiation model [10]. Calculation of radiation between solids including effect of radiative gas.
- Combustion model [5]. The implementation of the combustion model allows for the use of real mixtures. As much as 20 species can be included in the calculation.
- Neutral model. A neutral model is just influencing the flow without interacting with the other models.
- A two-phase layer model. Allowing phase shift, evaporation and condensing of liquid and gas.

All these models can be applied simultaneously in a simulation, interacting with each other. Some variable will be solved all through the different models where it is included. Typically will the temperature be calculated both for the flow-model and for the conduction model. This technique gives a more precise picture of the interaction between the different models and therefore also the multi physics process.

Brilliant has two databases connected: A database for the calculation of hydrocarbon fluid properties (also water) including

flashing, and a database including properties for solid materials. The code is case independent in the manner that no programming is necessary to describe a case.

In the following a calculation case is presented to illustrate the possibilities that a multi physics simulation can offer.

Case description

The case chosen is a jet impinging on a steel structure. Actually, the set-up is the one used for jet fire testing. The reason for choosing this case was two-fold: Fire against a steel structure is also interesting with respect to process equipment exposed to fire, as described above. The second reason is that measurements open for reference exists that can be used as a comparison to the simulation results. The experimental results are described in [9].

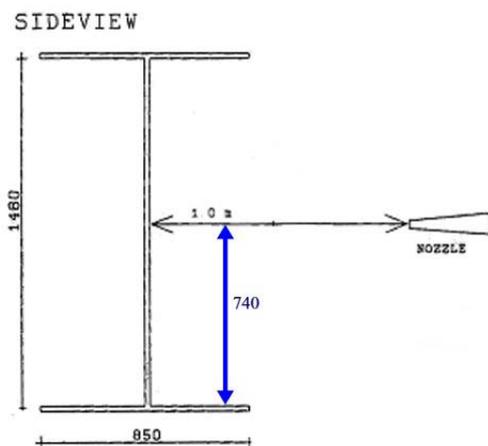


Figure 2 Schematic picture of the experimental set-up for the case.

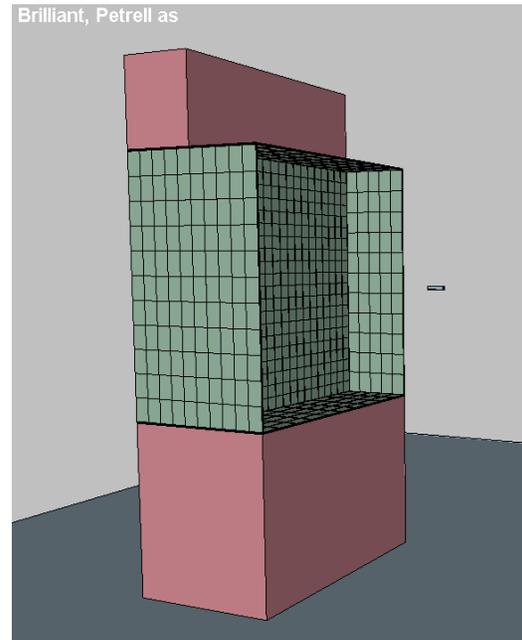


Figure 3 Calculation geometry. The small object in front of the box is the nozzle.

Figure 2 shows the experimental set-up. The nozzle is located 1 m from the centre of the box. The nozzle had an inner diameter at the outlet of 17.8 mm. The fuel was propane (C₃H₈) and during the test the flow was kept constant at 0.3 kg/s. The box was made of steel-plates, 10 mm thick and the inside dimensions were 1480 x 1480 mm. The flange length was 850 mm and the depth was 420 mm. Figure 3 shows the geometry used for the calculations.

The temperature was measured on the unexposed side of the box. Thermocouples were located vertically along the centre line. The lowest (no. 16) was located 110 mm from the inside flange and the rest were located 140 mm apart. The upper thermocouple (no. 25) was located 110 mm from the upper flange. The results are shown in Figure 4. The fire lasted for about 232 seconds. The gas flow was gradually increased over a period of about 30 seconds.

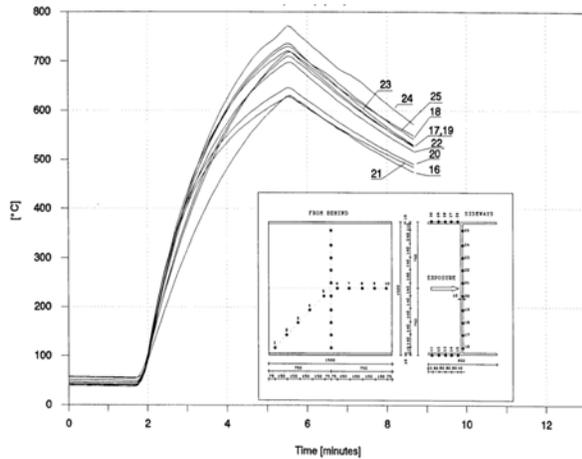


Figure 4 Experimental results showing the temperature of the steel surface at the opposite side of the flame exposure. The numbers are referring to the thermocouples located along a vertical line in the middle of the box. See the right, lower part of the figure. Thermocouple no. 16 is the lowest on the vertical line and no. 25 is the uppermost.

Figure 5 shows the measurements of velocity, measured along the vertical centre line in the centre of the box. Five probes were used, four placed at the surface of the centre plate and one at the upper edge of the flange. Probe 2 was located 100 mm from the centre plate measuring the velocity normal to the centre plate. The other probes were measuring the velocities parallel to the centre plate. The results are presented in Figure 5.

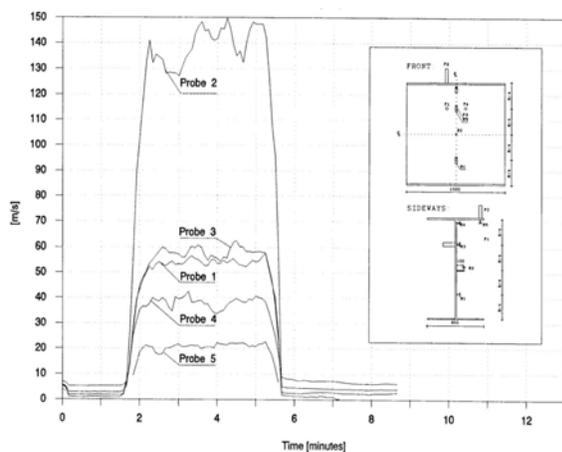


Figure 5 Gas velocities at the surface of the box, normal to the jet.

Calculation results

Results from the calculations are shown in the following figures. Figure 6 shows the calculation domain. The box is placed on bricks on the floor and some bricks are located on the top. The box itself, as well as the floor, uses the conduction model. The bricks and the nozzle are modelled with the neutral model, which means they do not interact except from being obstacles. The rest of the domain uses the combustion model, which in turn depends on the dispersion, flow and radiation models.

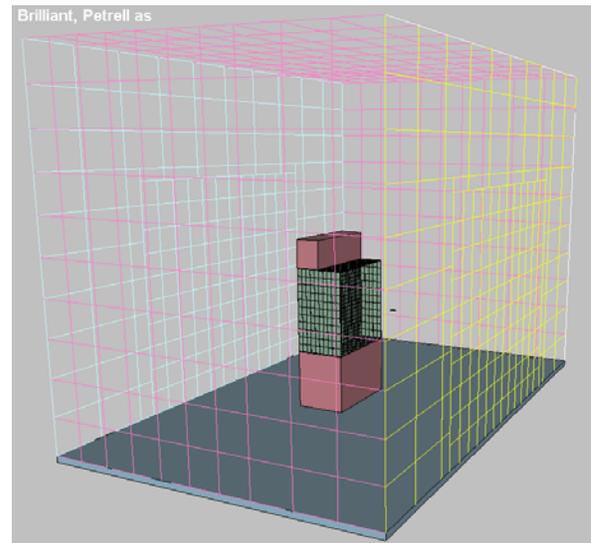


Figure 6 The box and nozzle inside the calculation domain.

The model includes the nozzle flow as well as the rest of the jet. This gives a more realistic turbulent structure. Figure 7 shows details from the nozzle exit. The gas is released inside the nozzle and the system is calculating the velocities at the outlet based on pressure gradients in the nozzle.

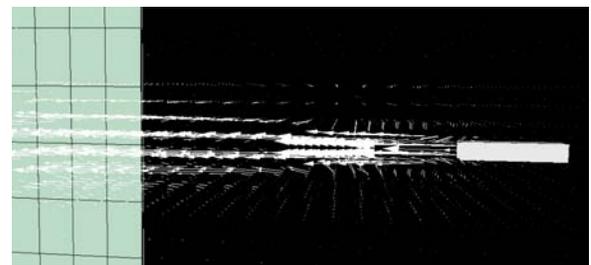


Figure 7 Velocities around the nozzle outlet.

The temperature distribution of the box is illustrated in Figure 8, Figure 9, Figure 10 and Figure 11.

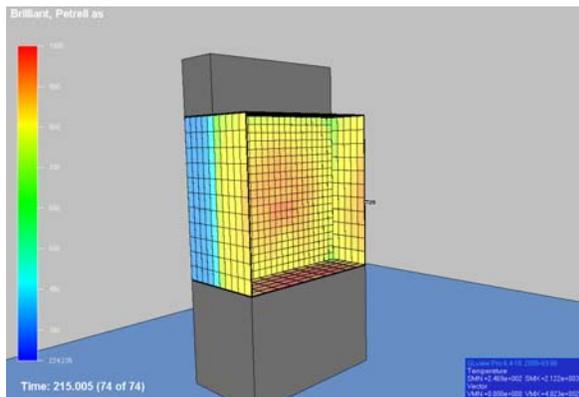


Figure 8 Temperature profile inside the box at the flame exposure side.

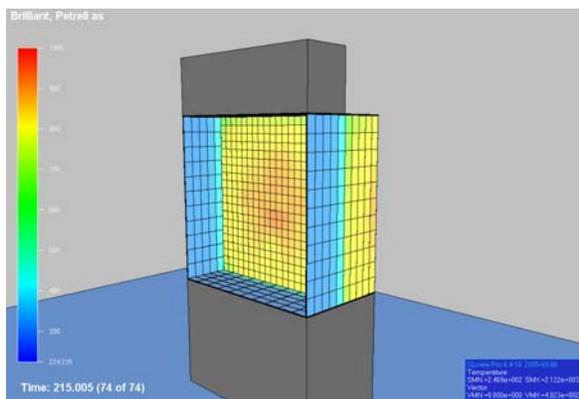


Figure 9 Box temperature after 3.6 minutes at the backside of the box.

Figure 11 shows the temperature variation with time for three locations on the backside of the box. The temperatures are plotted together with the measured temperatures for the same area.

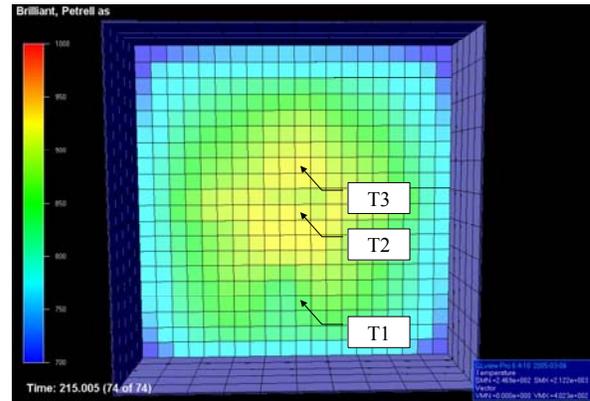


Figure 10 Locations where the calculated temperatures have been monitored on the opposite side of the flame exposure.

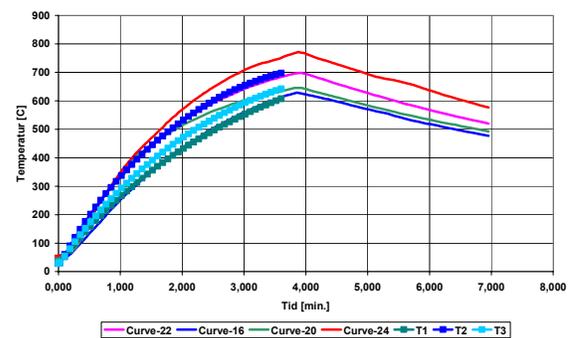


Figure 11 Temperatures on the backside of the box as functions of time. T1, T2 and T3 refer to Figure 10.

The temperature of the flame is illustrated in the next three figures. The flame is shown at different times during the early stage of the release. In Figure 14 the flame is fully developed. The effect of buoyancy is clearly seen in the figure.

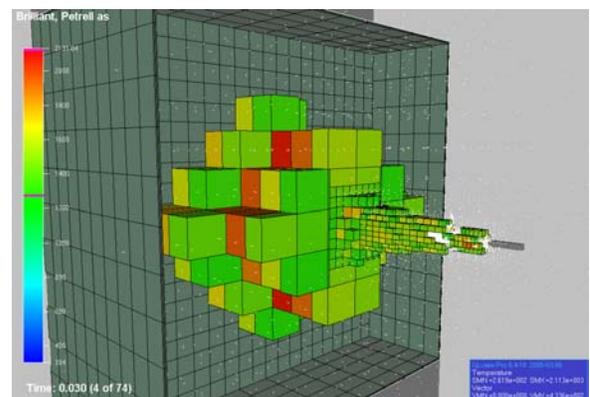


Figure 12 Flame temperatures right after ignition at 0.03 seconds.

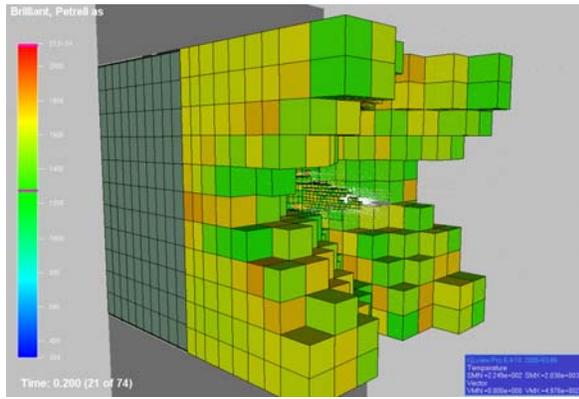


Figure 13 Flame temperatures at 0.2 seconds.

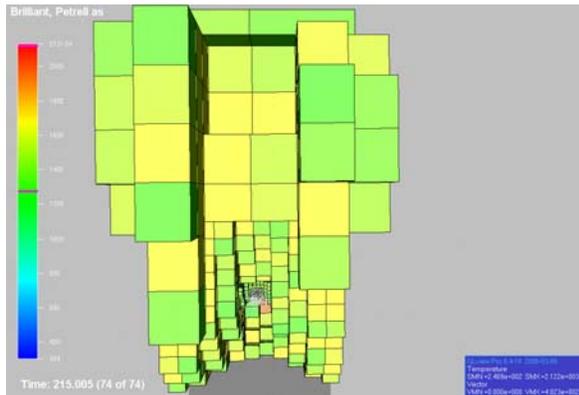


Figure 14 Flame temperatures in a developed flame. The effect of buoyancy is clearly demonstrated.

The next figures show radiated heat from the flame. It is the exposing heat that is illustrated.

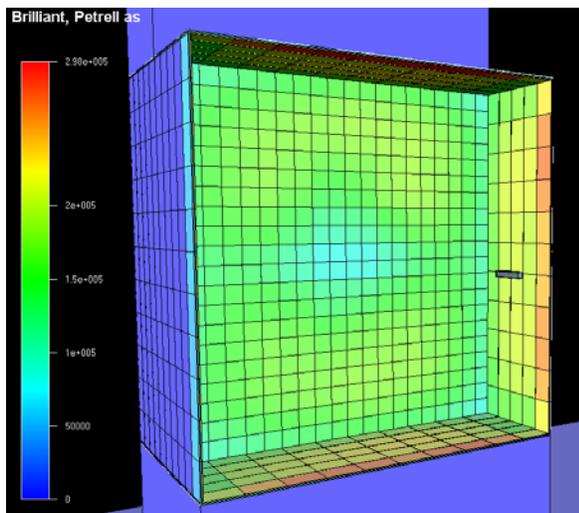


Figure 15 Radiation heat load from the flame in a developed flame.

From Figure 15 it can be seen that the radiative heat exposure varies only slightly

across the surface. In Figure 16 the heat variation is plotted against time for the most exposed location at the centre of the surface. A variation can be noticed in the initial phase, but it is soon stabilised.

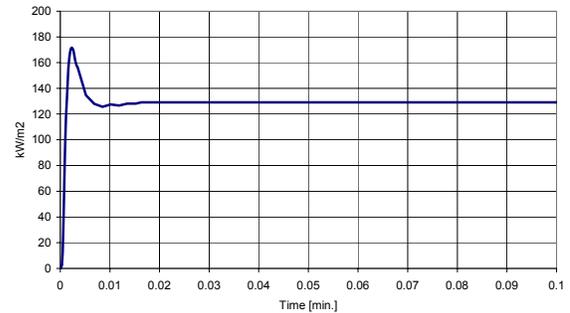


Figure 16 Radiative heat flux as a function of time. The curve shows typical values on the box surface.

The convective heat is illustrated in Figure 17 and Figure 18, and Figure 19 shows the sum of the radiative and convective heat loads.

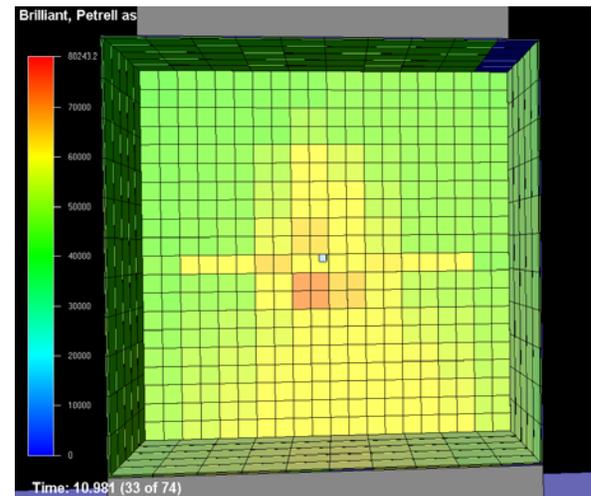


Figure 17 Convective heat transfer to the box surface.

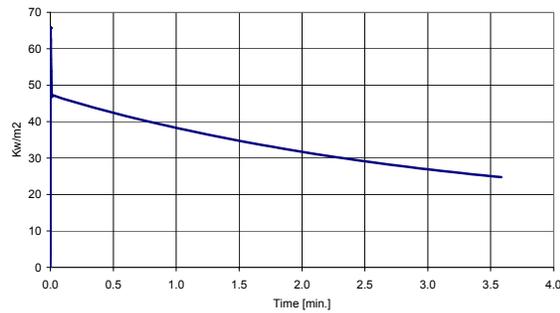


Figure 18 Convective heat transfer to the box surface.

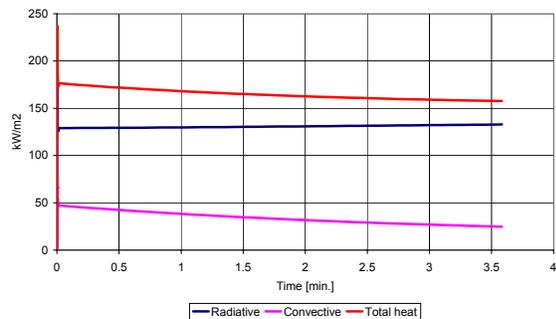


Figure 19 Total heat load exposing the box surface

The last two figures show the velocity where the jet is hitting the centre of the surface. The velocity component normal to the surface at three locations with different distances from the surface are plotted in Figure 21.

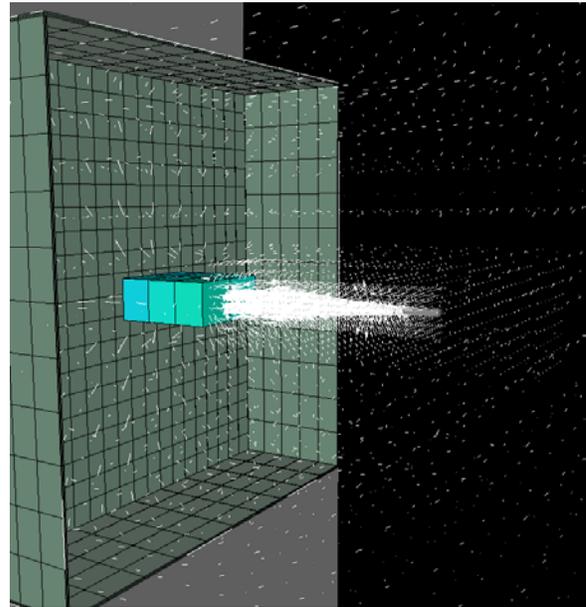


Figure 20 Velocity vectors at the surface of the box. The three coloured control volumes show where the velocities are monitored.

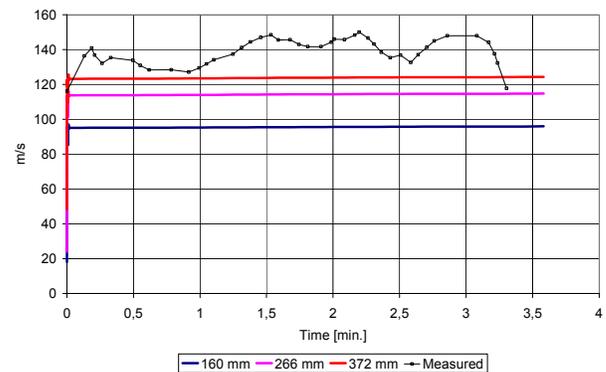


Figure 21 Monitored velocities at three different distances from the box surface. The related control volumes are shown in Figure 20.

Comparing Figure 5 (probe 2) and Figure 21 it can be seen that the velocities are of the same magnitude.

Conclusions

Multi physics simulation is a promising technique. It allows for the combination of models for different physical phenomena. This can give better predictions and save time when treating complex problems. The initial problem stated is a typical example where several physical models are necessary to describe the complete process.

A crucial model in such a problem is the heat transfer from the flame to the shell surface. This is the problem that has been investigated more closely in this paper. It can be concluded that the agreement between the measurements and the calculations are reasonably good. The calculated steel temperatures are in the same range as the measured values. There are always uncertainties involved in experimental measurements as well as in calculations. The most reliable measurements are without doubt the measured steel temperatures. The measured radiative and convective heat fluxes have greater uncertainties. Nevertheless, the results seem to be within reasonable limits.

Acknowledgements

This work is sponsored by the Petroleum Safety Authority Norway.



The authors want to thank the representatives

Harald Olstad and
Per Endresen

for their engagement and support during the work.

References

- [1] Versteeg, H.K. and Malalasekera, W. 1995. An introduction to Computational Fluid Dynamics. Longman Group Ltd.
- [2] Hirsch, C. 1994. Numerical Computation of Internal and External Flows, Vol. 1 and 2. John Willey & Sons.
- [3] Van Doormal, J.P. and Raithby, G.D. 1984. Enhancement of the SIMPLE Method for Predicting Incompressible Fluid Flows. Numerical Heat Transfer, Vol. 7, pp. 147-163.
- [4] Rhie, C.M. and Chow, W.L. 1983. Numerical Study of the Turbulent Flow Past an Airfoil with Trailing Edge Separation. AIAA Journal, Vol. 21, No. 11.
- [5] Magnussen, B.F., Hjertager, B.H. 1976. On the mathematical modelling of turbulent combustion with special emphasis on soot formation and combustion. 16th Symp.(Int) on combustion, pp 719-729.
- [6] Berge, G. and Medonos, S. 2000. Performance Of Materials Used On An Offshore Separator Affected By A Fire. 19th International OMAE Conference OMAE'2000 February 14 – 17, 2000, New Orleans.
- [7] Berge, G. and Wighus, R. 2002. On Modelling Of Heat Load To Objects Exposed To Fire. 21st International Conference on Offshore Mechanics and Arctic Engineering 23-28 June 2002, Oslo, Norway.
- [8] Berge, G. 2004. Process equipment exposed to fire. Why investigation of heat loads based on simulated experiments is important. Major Hazards Offshore, ERA Technology Ltd, London.
- [9] Wighus, R. and Drangsholt, G. 1993. Impinging jet fire experiments – propane 14 MW laboratory tests. SINTEF NBL, STF25 A92026, 1993-03-10.
- [10] F.C. Lockwood and N.G. Shah, A new radiation solution method for incorporation in general in general combustion prediction procedures, Eighteenth Symposium (International) on Combustion. The combustion institute, pp 1405 – 1409, 1981.