*Third International Conference on CFD in the Minerals and Process Industries*
*CSIRO, Melbourne, Australia*
*10-12 December 2003*

# HIGH ASPECT RATIO ORIFICE JET LEAKS WITHIN A PRODUCTION AREA OF AN OFFSHORE SUPERSTRUCTURE

**Sarah J. WAKES**[1]

[1] Department of Design Studies, University of Otago, PO Box 56, Dunedin 9001, NEW ZEALAND

## ABSTRACT

Safety is of paramount importance upon an offshore oil or gas superstructure used for drilling and production of hydrocarbons. Optimum positioning and numbers of detectors for escaped hydrocarbons can prevent disasters occurring such as fires or explosions, (Cullen, 1990). Sophisticated numerical techniques are often used now to model such scenarios before construction of a superstructure. With all such simulations there is a reliance on appropriate approximations for difficult physical and numerical situations such as the size, position and type of gas leak (Wakes, et al., 1997). It has been shown that approximating the leak as an axisymmetric jet, (Wakes, et al., 2002) is not acceptable within a larger CFD simulation of hydrocarbon gas dispersion for safety purposes as a large number of leaks result from failed gaskets (Papadakis, 2000). This paper investigates aspects of modelling the jet exit orifice and inlet conditions with validation against experimental results (Meares, 1998) with a view to improving safety.

## NOMENCLATURE

$C_\mu$ constant
$k$ turbulent kinetic energy
$p$ pressure
$\mathbf{u}$ velocity
$u^+$ non-dimensional velocity
y distance from wall
$y^+$ non-dimensional distance from wall
$\delta_{ij}$ Krounecker delta
$\varepsilon$ turbulent dissipation
$\rho$ density
$\mu$ dynamic viscosity
$\nu$ kinematic viscosity
$\nu_T$ turbulent viscosity

## INTRODUCTION

Safety is paramount on offshore production facilities with high pressure and temperature hydrocarbons being transported through networks of pipes joined by flanges and gaskets. It is impossible to predict where a gas leak might occur or from what although it has been concluded (Papadakis, 2000) that leaks from failed flanges or blown gaskets are the most likely source. It is important therefore when conducting a numerical study investigating gas dispersion for the whole of a production area that the worst case scenarios are modelled for the leaks. This means that the behaviour of the most common form of leaks needs to be known in order to assess what the worst case scenario will be. This will be most likely to be leaks emanating from high aspect ratio orifices.

Previous experimental work has been done (Meares, 1998) on some idealised high aspect ratio cross-sectional jets showing a wide range of behaviour in terms of spreading angles, centreline velocity profiles and velocity profiles across the jet. This work successfully proved that high aspect ratio cross-sectional jets do not approximate within the area of interest in behaviour to axisymmetric jets in terms of centreline velocity profiles, spreading angles and other measures of a jet behaviour. It belies the common practice of modelling a leak within a production area safety simulation as an axisymmetric jet, (Wakes, et al., 1997) as being a good enough approximation. This could have serious safety implications ranging from: conservative estimates of the time it would take for flammable fluids from the leak to reach hot surfaces, the estimated size of the explosive cloud of hydrocarbon/air mixture being too small and growing at an incorrect rate, and the incorrect positioning of gas detectors. For all safety scenario simulations it is vital that the worst case situations are modelled so as to test the safety procedures to their utmost. An axisymmetric jet in most cases is not the worst case scenario. With more affordable better computing power it is possible to model vital elements such as this to improve safety.

This work concentrates upon the fluid leaving an high aspect ratio orifice and emerging as a jet into the air. For an effective safety study the most appropriate way to model this sort of jet is needed. Some work has already been undertaken, (Wakes, 2003a, b) that establishes with validation against experimental work as well as initial investigation into the modelling of the inlet of the jet. The turbulence model selected was the realisable k-ε two-equation model as it is proven to be a better choice for axisymmetric and planar jet simulations, (Shih, et al., 1995).

For this work the jet will exit from the curved high aspect ratio cross-sectional orifice. The inlet velocity is determined by simulation of the flow within the pipe and gasket, also described in this paper.

## NUMERICAL METHOD

### Geometry

This work is based on experimental work, (Meares, 1998), that examined a set of gaskets with cut outs to represent idealised leaks with four pipe pressures. Figure 1 shows the experimental set-up with the pipe flange and blank end between which the gaskets are sandwiched. Comparing with the real-life situation of hydrocarbons at high temperatures the Grashof number indicates that the buoyancy forces does not dominate the flow therefore so the assumption of using air with no thermal forces is a reasonable one for the experimental and numerical experiments. It is suggested that this will give a worst case scenario. The experimental work looked at nine separate gasket shapes, seeded the air jet with smoke and used a high quality laser sheet to illuminate the jet and enable photographs to be taken. For four of these gaskets additional pressure measurements were taken giving centreline velocity details as well as the velocity across the jet.

**Figure 1**: Photograph showing the experimental set-up of the pipe, flange and blank end.

For the purpose of this numerical work two of the four gaskets with velocity data available are modelled numerically and can be seen in figure 2. The 00_90 gasket is a parallel sided leak with channel type nozzle geometry whilst 70_45 is a rapidly contracting then diverging exit geometry. The aspect ratios for the outer orifice are 119.7:1 and 26.6:1 respectively. The gasket width is 2mm for all cases. The equivalent hydraulic radius is approximately 0.004m for each case.

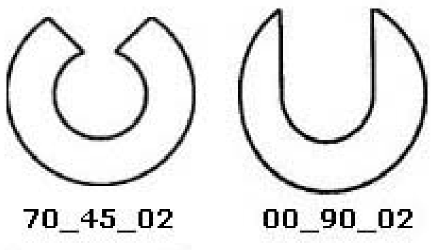
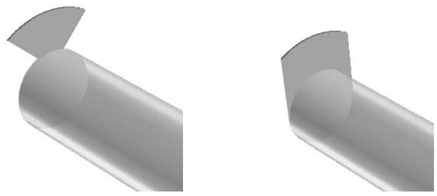



**Figure 2**: Schematic showing the 00_90 and 70_45 gaskets.

Only one pipe pressure is used, 68.9 KPa and two separate numerical simulations were run, an internal one to determine the correct velocity profile exiting the orifice and the jet exiting from this high aspect ratio cross-sectional orifice into the air. The Reynolds numbers for both cases are approximately 40,000. Figure 3 shows the geometries for the two gaskets.

Previous work, (Wakes, 2003b), has determined that the length of pipe for the internal pipe simulations to be 6.56D and the turbulence model to be the two equation k-ε, (Wakes, 2003a, b) for both numerical simulations.



(a) (b)

**Figure 3:**(a) 70_45 gasket geometry & (b) 00_90 gasket internal geometry

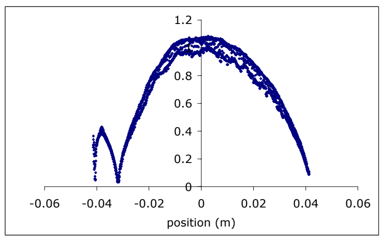
**Numerical Model**

For the internal pipe flow with a pressure of 68.9 KPa. For the external jet flow the number of computational cells is 824205 for the 00_90 gasket and 855486 for the 70_45 gasket each with a box of 0.1m x 0.5m x 0.5m surrounding the orifice. The inlet velocity profile is determined by the results from the internal flow. The commercial code Fluent was used and the simulations run for a steady state case.

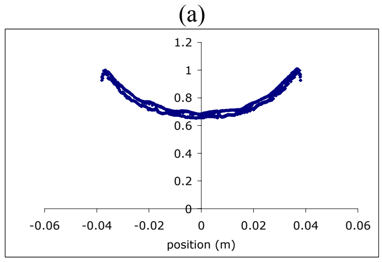
**Internal Pipe Flows**

Figure 4 shows the inlet profile used for the jet simulations. It can be seen for the 70_45 gasket the profile is not symmetrical across the inlet, possibly due to the recirculation within the pipe due to the blank end, and the peak velocity occurs within the middle of the gasket. The flow within the 70_45 gasket has recirculation areas that concentrate most of the flow within the central section of the gasket. The recirculation areas are caused by the narrow entrance to the gasket and wider exit.

For the 00_90 gasket the reverse is true with a saddleback type profile being displaced. The peak velocity occurs at the ends of the gaskets. The internal flow within this gasket is channelled by the parallel sides which leads to the velocity profile shown in figure 4(b). These will undoubtable have some influence on the behaviour of the jets emerging into air. To test this constant inlet velocity profiles are also evaluated.




(a)




(b)

**Figure 4**: inlet velocity profile for the external jet flow for (a) 70_45 & (b) 00_90

## RESULTS

### Spreading Angles

The spreading angle of the jet leak is important to offshore safety as it can ascertain the extent of the cloud of explosive gas occurring from the hydrocarbons entering the air from the gas leak. If predictions of the spreading angle of the jet are too small (too narrow a jet) then the inbuilt safety margins of the superstructure will not be sufficient and gas detectors maybe positioned wrongly. The larger the spreading angle then potentially the greater the air entrainment and hence the larger the cloud of explosive hydrocarbon/air mixture will be and will be reached faster. If the prediction exceeds the experimental spreading angle then there is a built-in safety margin for the detection of escaping hydrocarbons.

Table 1 shows the spreading angle compared for the numerical cases against the experimental data. It can be seen that in the plane parallel to the flange the predictions of the spreading angles are very good in the 00_90 gasket case but for the 70_45 gasket the constant inlet velocity case there is under prediction. In the perpendicular plane the predictions are not good, table 2. There is a vast overprediction by both numerical cases. For a safety case this is not too much of a problem as there will be a safety margin built in. The varying inlet case is a better choice as the over prediction is not so large.

| Gasket | Experimental case | Varying inlet velocity case | Constant inlet velocity case |
|---|---|---|---|
| 00_90 | 11.9° | 10° | 13° |
| 70_45 | 16° | 15.5° | 8° |

**Table 1**: Spreading angles in the parallel plane.

| Gasket | Experimental case | Varying inlet velocity case | Constant inlet velocity case |
|---|---|---|---|
| 00_90 | 8.3° | 17.5° | 25° |
| 70_45 | 13.8° | 18° | 26° |

**Table 2**: Spreading angles for the perpendicular plane.

This confirms the thought that in the parallel plane there is little viscous resistance. Within the perpendicular plane there is most of the viscous resistance therefore the predictions will be worse. The predictions for the 70_45 gasket are slightly better than for the 90_00 gasket. The constant velocity cases give worse predictions that leads to a conclusion that the varying velocity profile could be better to use.

### Centreline Velocity Decay

The centreline velocity decay profile is one of the indicators of how a jet behaves. It is a simple process to identify areas of interest within the jet such as the potential core region, characteristic decay region and the axisymmetric decay region. In terms of safety the centreline velocity decay profile indicates the size of the potential core, most likely to be of highly concentrated hydrocarbons outside of the explosive limits. There is also vital information on how the jets velocity decays with distance away from the jet that could have implications in terms of how far the jet will travel whilst mixing with entrained air and becoming an explosive hazard. There will be some help with the positioning of gas detectors that can be gleaned from this information.

For the 00_90 gasket figure 5 shows that the potential core is far shorter than found experimentally. For both numerical simulations there is little difference in the profiles except far away from the orifice. The characteristic decay region has a similar slope to the experimental results but within the axisymmetric decay region there are some differences. Figure 6 is the centreline velocity decay profile for 70_45 gasket and it can be seen that there is more difference between the two numerical solutions. For both of these cases the numerical solutions are predicting entrainment of the escaping gas with surrounding air early than the experimental work indicates. This could lead to the cloud of explosive hydrocarbon/air mixture being physically lower than occurs in reality. If gas detectors were positioned to detect the cloud in the numerically predicted position rather than for the experimentally predicted optimum position then detection would be earlier and again a safety margin has been built into the detection process.

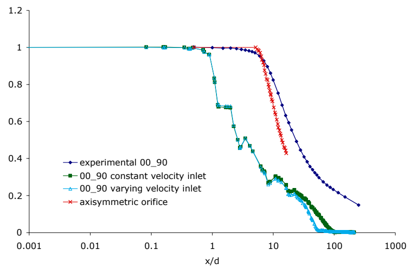



**Figure 5**: Centreline velocity decay profile for 00_90 gasket

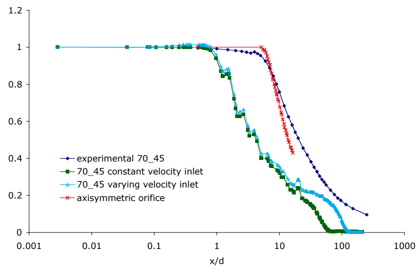



**Figure 6**: Centreline velocity decay profile for 70_45 gasket

### Velocity Profile Across the Jet

It is important to look at the velocity profile across the jet and not just centreline velocity decay. This can indicate important phenomena associated with the behaviour of a jet. Figure 7 shows that for the 00_90 gasket experimentally a saddleback velocity profile is found. This is where the velocity at the outer edges of the profile is higher than that of the centreline. As the inlet velocity profile for the jet imported from the internal simulations also displays these saddleback characteristics it is perpetuates the jet flow and therefore this case has a stronger saddleback profile downstream than the constant inlet velocity case. For the 70_45 gasket the experimental results do not give a saddleback profile and only the varying velocity profile gives this numerically. This could be due to the inlet profile for the 70_45 gasket having a velocity peak in the middle and therefore influencing the flow further downstream. This does then favour the use of a correct inlet profile in order to obtain a correct type of velocity profile further downstream. For both cases as the distance from the gasket increases it can be seen that the numerical predictions of the velocity profile are lower than predicted in experiments. This gives a conservative estimate of the jet behaviour as it implies that the jet will disperse less quickly.

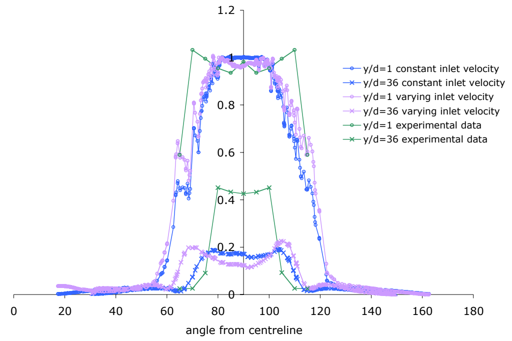



**Figure 7:** Velocity profile across 00_90 gasket

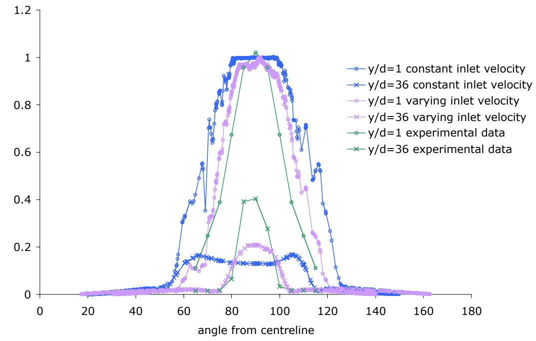
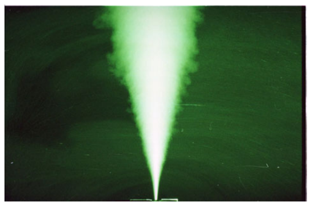



**Figure 8 :** Velocity profile across 70_45 gasket

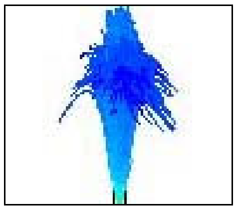
### Visual Comparison

Visual comparison is important as it gives an overall impression of the air entrainment into the jet and behaviour of the jet.

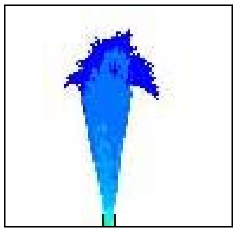
There are two planes of view that are significant within this flow. Parallel to the flange, the narrow plane of the jet and perpendicular to the flange the broad plane of the jet. As the jet is a three-dimensional flow the entrainment of surrounding air into the jet is important in both cases. A comparison is given here of the photographs from the experimental work and plane views of non-reacting particles being released into the flow to simulate the smoke particles seeding the flow experimentally. Experimentally zinc chloride particles of a diameter of 0.5µm were used, (Meares, et al., 1997) that were deemed to have minimal influence on the flow and within the simulation mass-less particles were used. Figures 9 & 10 show the parallel to the flange view for the 00_90 & 70_45 gaskets respectively. It can be seen that there is little difference between the two numerical cases, the varying and constant inlet velocity profiles and that they agree well with the photographs. The varying inlet profile cases for both gaskets can be seen to be a better visual match for the photographs. In the perpendicular plane there can be seen more of a difference. This is confirmed by the similarities of the spreading angles in the parallel plane and disparities in the perpendicular plane.



(a)

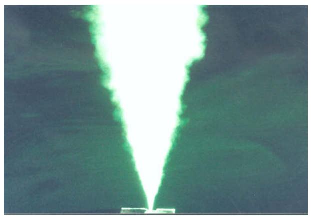


(b)

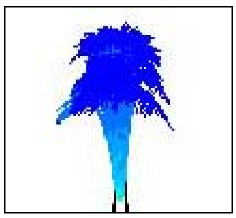


(c)

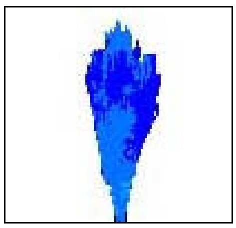
**Figure 9 :** 00_90 gasket in the view parallel plane (a) experimental (b) constant velocity inlet profile & (c) varying inlet profile



(a)



(b)

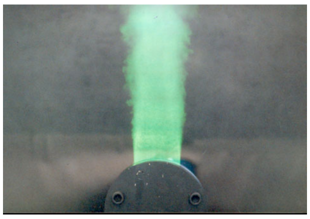


(c)

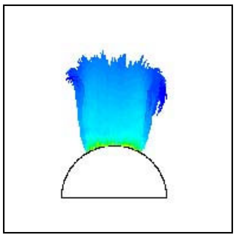
**Figure 10:** 70_45 gasket in the view parallel plane (a) experimental (b) constant velocity inlet profile & (c) varying inlet profile

In the plane perpendicular to the flange, figures 11 & 12, it can be seen that there is much more of a visible difference in the flow fields. Figure 11 shows that for the varying inlet profile case the saddleback velocity profile is in evidence but that there is more spreading than in the experimental case.

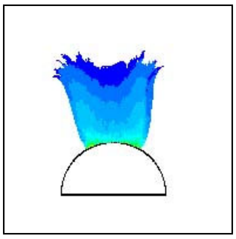
Figure 12 shows that for the 70_45 gasket the varying inlet velocity profile case the flow is not symmetrical. This is from the non-symmetrical nature of the inlet velocity profile that has a small peak near the left hand side of the gasket shown in figure 4(a). It can be seen though that visually the spreading of the constant inlet case is greater than the experimental and varying inlet case.



(a)

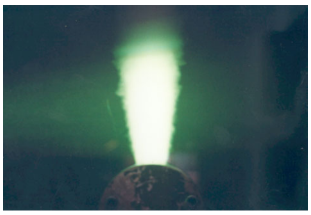


(b)

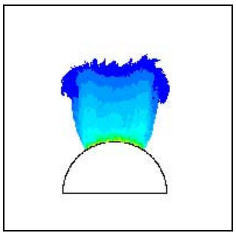


(c)

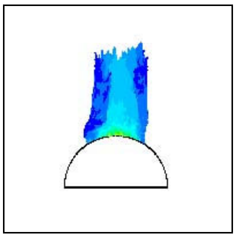
**Figure 11:** 00_90 gasket in the view perpendicular plane (a) experimental (b) constant velocity inlet profile & (c) varying inlet profile



(a)



(b)



(c)

**Figure 12:** 70_45 gasket in the view perpendicular plane (a) experimental (b) constant velocity inlet profile & (c) varying inlet profile

## DISCUSSION

It can be seen that having the varying velocity inlet has some influence on the flow from the jet out of the gasket. It reinforces the velocity profile of the jet downstream of the orifice and it becomes obvious why the two gaskets have difference velocities across the jet flow.

The blank end of the flange causes recirculation within the flange end of the pipe and this causes some asymmetry in the flow, particularly for the 70_45 gasket into the gasket and subsequently out of the orifice into jet flow. Further work is needed on the other gasket shapes. Using the varying velocity inlet is more realistic as it did produce more of the features seen in the experimental work. There are still significant shortfalls in the accuracy of the centreline velocity predictions and spreading angle predictions in the perpendicular plane. This indicates that the realisable k-ε two-equation turbulence model can be improved upon for the numerical predictions of such high aspect ratio orifice jets. This could be due to the high viscous resistance thought to be in the perpendicular plane and low resistance in the parallel plane. As each shape of gasket produces a different inlet velocity profile it could prove difficult to use effectively and a blanket constant velocity profile maybe more practical for larger safety simulations. It is never really known what the leak will look like until after the event so the worst case scenario needs to be decided to use for the larger safety simulations.

In terms of safety in offshore production areas where leaks could occur, the over prediction of the spreading angles, particularly in the perpendicular plane will give a built in safety margin to any safety analysis simulations done. Ideally it would be better to have closer numerical results to the experimental work but as long as the safety margins are on the conservative side then they can be used. The other crucial observation reinforces other papers, (Wakes, et al., 2002), is that such jets should never be modelled as axisymmetric as it is likely that all of the action will be within quite a close range to the orifice of the jet leak.

Within a larger safety simulation of an offshore superstructure, using a high aspect ratio cross-sectional orifice jet should improve safety predictions and with the greater computing power available now should be possible to model this level of detail even in a larger safety simulation to improve safety predictions.

## CONCLUSION

The following conclusions can be drawn from this preliminary work:

1. The use of axisymmetric jets to simulate leaks on an offshore structure is not recommended for safety simulations.
2. Further investigation into the most suitable turbulence model is needed, particularly looking at using Large Eddy Simulation to capture the time dependent nature of the flow and better predictions of the indicators of jet behaviour such as centreline velocity decay profiles.
3. There is very different behaviour of the flow at the orifice of the jet with different gasket shapes so further investigative work is needed to establish patterns.
4. In a simulation of such a leak the correct inlet velocity profile at the jet orifice should always be used if possible.

## ACKNOWLEDGEMENTS

The author would like to acknowledge the University of Otago Research Grant Number 200200533 used for this work and Edgar Rodriguez for his help in preparing the data.

## REFERENCES

CULLEN, T.H.L., (1990), "The public enquiry into the piper alpha disaster", *HMSO*.

MEARES, A.J., HOLDØ, A.E. and WAKES, S.J., (1997), "A novel seeding technique for the flow visualisation of pressurised air flows", *Meas. Sci. Tech,* **8**, 1183-1185.

PAPADAKIS, G.A., (2000), "Assessment requirements on safety management systems in EU regulations for the control of major hazard pipelines", *Journal of Hazardous Materials,* **78**, 63-89.

SHIH, T.H., LIOU, W.W., SHABBIR, A., YANG, Z. and ZHU, J., (1995), "A new k-e eddy viscosity model for high Reynolds number turbulent flows", *Computers & Fluids,* **24**, 227-238.

WAKES, S., (2003a), "Computational fluid dynamic modelling of high aspect ratio cross-sectional jets I: The effect of orifice shape on jet behaviour", *ASME PVP*, Cleveland, Ohio, USA, July 20-24.

WAKES, S., (2003b), "Computational fluid dynamic modelling of high aspect ratio cross-sectional jets ii: The flow within the pipe and flange", *ASME PVP*, Cleveland, Ohio, USA, July 20-24.

WAKES, S., HOLDØ, A.E. and MEARES, A.J., (2002), "Experimental investigation of the effect orifice shape and fluid pressure has on high aspect ratio cross-sectional jet behaviour", *Journal of Hazardous Materials,* **A89**, 1-27.

WAKES, S.J., TOFTEN, T.H. and INDRESTRAND, T., (1997), "The modelling of diffuse gas leaks on a superstructure using computational fluid dynamics and the safety implications", *Application of Fluid Dynamics in the Safe Design of Topsides and Superstructures*, I. Mar. E., London.