13th Australasian Fluid Mechanics Conference
Monash University, Melbourne, Australia
13-18 December 1998

AN INVESTIGATION INTO MECHANISMS OF MINE BURIAL ON THE SEA BED

Paul R. KLEPETKO and David F. FLETCHER

Department of Chemical Engineering
The University of Sydney, New South Wales, AUSTRALIA

ABSTRACT

The work presented in this paper investigates the application of a numerical method (CFD) to understanding the burial process of objects on the seabed. Vortex formation and end flow patterns about submerged objects were predicted in the presence of bulk fluid oscillatory flow patterns. Observations were also made on the ability of numerical methods to predict higher order wave flow patterns and profiles.

INTRODUCTION

Wave theory itself is well-known, with early work begun by Laplus, Airy and Stokes. However the extension of wave theory to sediment transport is not as developed. Sediment transport caused by wave motion adds many complications to simple wave motion. The results are often critically dependent on the sea floor boundary layer, where viscous forces, negligible elsewhere in the wave, impact significantly on initial particle entrainment.

The application of wave sediment transport theory to scour and burial of an object has only begun recently, with Herbich, Schiller, Watanabe and Dunlap (1984) providing the first notable literature on the topic. They well describe the topic by the statement, "The mechanics of sediment transport and scour in the ocean environment are very complicated". Their work does not deal with small objects on the seabed, but larger structures, such as sea walls and platform piles.

Experimental work conducted by Mulhearn (1993) and (1995) explicitly examined two scenarios for mine burial. In both cases a mine was placed perpendicular to the prevailing swell direction.

The first experiment (Mulhearn 1993) was conducted at a water depth of 25 metres. In this experiment, very little burial occurred, even after periods of very severe weather.

Mulhearn's second experiment (Mulhearn 1995) was conducted in water of depth 11 metres. In contrast to the first experiment, complete burial was achieved after a period of very heavy weather.

The work in this paper simulated seabed flow conditions numerically using the commercial CFD package CFX 4 (CFX, 1997). The simulation was developed to replicate the time dependant conditions observed in Mulhearn's second experiment that led to burial of the mine. Flow patterns, and shear stresses acting upon the sea bed and mine, were analysed at various levels of burial. This allowed the mechanisms of sediment scour and mine motion to be identified that result in burial of the mine.

DESCRIPTION OF THE MODEL

A solution space was generated as shown in figure 1 below. The dimensions of this box were set to capture the conditions observed by Mulhearn. Defined dimensions were a length (l) of approximately 60 meters and height (h) of 20 meters. For analysis of the flow effects about the bulk cylinder, a 2 dimensional section was analysed. To determine flow disturbances caused by the ends of the mine, a full 3 dimensional model was defined. In this case the cylinder length was 2 meters and the solution domain length was 4 meters.

The cylinder was also analysed under various situations. With a defined diameter (d) of 0.3 meters, numerical flow solutions were computed when the cylinder was 91% exposed, 50% exposed and 9% exposed. These results allowed the examination of fluid disturbances about the object at various stages of the burial process.

A structured, non orthogonal grid was constructed within the solution space concentrating small volumes close to the bottom boundary. Gridding was also concentrated towards the centre of the solution space, where the mine was located.

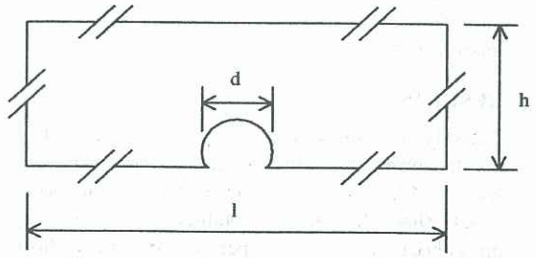


Figure 1 : Model solution space.

Fluid Parameters

A multiphase system was required to achieve realistic flow patterns within the solution space. Water and air were used as incompressible, Newtonian fluids, allowing the model to be built with a gas phase present above a liquid phase. This allowed the boundary conditions to be built so that the flow in and out of the solution space could be modelled.

Well-defined wave models (discussed later) were used to define the end boundaries with the top boundary set at a constant pressure. Turbulence was modelled using the standard k-ε model.

Boundary Conditions

The desired outcome of this model was to generate a realistic wave action for conditions similar to those observed by Mulhearn. To achieve this the model therefore had to be capable of generating a wave in 11 meters of water, with a period of 12 seconds, and velocity 0.9 m/s at a height of 1 meter above the seabed. (These values are consistent with measurements given in Mulhearn, 1993).

After reviewing wave theory literature a first order approximation (Airy wave) was used under the conditions described above. Work done by Longuet and Higgins (1953) was used to confirm that this wave model was sufficiently accurate to be applied to this model.

Velocity profiles for the wave were calculated to be:

$$u(x,y,t) = 0.8822\cosh(0.0512y)\cos(0.0512x - 0.5236t)$$
$$v(x,y,t) = 0.8822\sinh(0.0512y)\sin(0.0512x - 0.5236t)$$

The surface profile was generated via:

$$\eta(x,t) = \cos(0.0512x - 05236t)$$

Initial conditions were generated setting the "t" term to zero seconds. Boundary conditions were assigned by setting the "x" term to 0 meters and 60 meters.

Solver Parameters

The model was run using fixed time stepping. 120 steps of 0.1 seconds were made, reflecting in total an entire wavelength of the wave defined by the boundary conditions.

For each finite volume the Navier Stokes equations have been solved and k-ε model used to simulate turbulent flow close to the walls. A homogeneous multiphase model with a surface sharpening algorithm, was used to ensure that the air/water interface remained distinct.

Solution convergence was obtained by monitoring the mass flux residual until it was under 0.1% of the total mass flow rate.

RESULTS

Velocity and shear stresses were recorded at various points during the transient calculations. Analysing the wave profile generated in the entire solution space it was noted that the wave obtained a realistic profile throughout the transient period. Maximum horizontal velocities were generated under the wave crest and trough, and maximum vertical velocities were obtained as the wave passed through the mean water level.

Figure 2 shows that the wave motion generated slightly lags the theoretical conditions imposed on the boundary. This error can also be seen to grow throughout the transient run and is consistent with the difference in assumptions between the first order boundary conditions and CFX solution conditions. Two particular assumptions of linear wave theory were considered to have contributed significantly to these errors.

CFX includes the modelling of viscous and turbulence effects, and hence the assumption of an inlet profile based on inviscid conditions is only approximate. The lags generated through the additional forces were observed to oppose the wave motion as the boundary conditions moved ahead of the bulk wave.

CFX also generated waves with flatter troughs and steeper crests, which is markedly different from the sinusoidal profile assumed in Airy wave theory. Linear wave theory makes the assumption that the wave profile can be accurately truncated to a first order solution, generating a sinusoidal wave approximation. Higher order solutions generated by Stokes have shown that a solution with flatter troughs and steeper crests can actually be obtained. These solutions retain the first order term as generated by Airy, and also contain a second order term that oscillates at twice the wave frequency. This second term generates a wave profile similar to that modelled by CFX.

The use of the second order solution has many advantages over Airy wave theory. Prediction of an open orbital for the fluid particles in the wave cycle, and a net mass drift in the wave direction, are just two of the more realistic features achieved by higher order solutions. The use of Linear wave theory in this model was shown to be able to produce reasonable results, but further work needs to be conducted to determine more accurately the errors that the various wave theories generate in the model.

Vortex Ejection

The flow solutions obtained from a 2 dimensional run (depicted in figure 3) showed that the disturbances caused by the cylinder generated vortices that grew and dissipated throughout a half wave cycle. The vortices were observed to have formed by the time the maximum bulk fluid velocities were obtained (times 0s, 6s and 12s) and, as the bulk fluid decelerated, these vortices were noted to grow to reach their maximum intensity at the smallest bulk fluid velocities (times 3s and 9s). During the acceleration of the bulk fluid velocities the vortices proceeded to move out into the bulk fluid (ejecting), rapidly dissipating and beginning the formation process in the opposite direction.

Vortices were observed to be generated on both sides of the cylinder simultaneously. The downstream vortex (or primary vortex) was observed to be significantly larger than the upstream vortex (or secondary vortex). As the wave cycle proceeded it was observed that the primary vortex, at its maximum intensity, was recirculating water from more that five diameters downstream of the cylinder. At this time, the vortex was also generating seabed velocities towards the cylinder of approximately half the maximum bulk fluid velocity achieved throughout the wave cycle.

The secondary vortex was observed to be significantly smaller than the primary vortex, and at its maximum intensity, it was observed to be generating a low velocity back flow away from the cylinder, projecting about two diameters from the cylinder. This vortex was noted to dissipate earlier than the primary vortex.

It should be noted that the two vortices were generated by very different mechanisms.

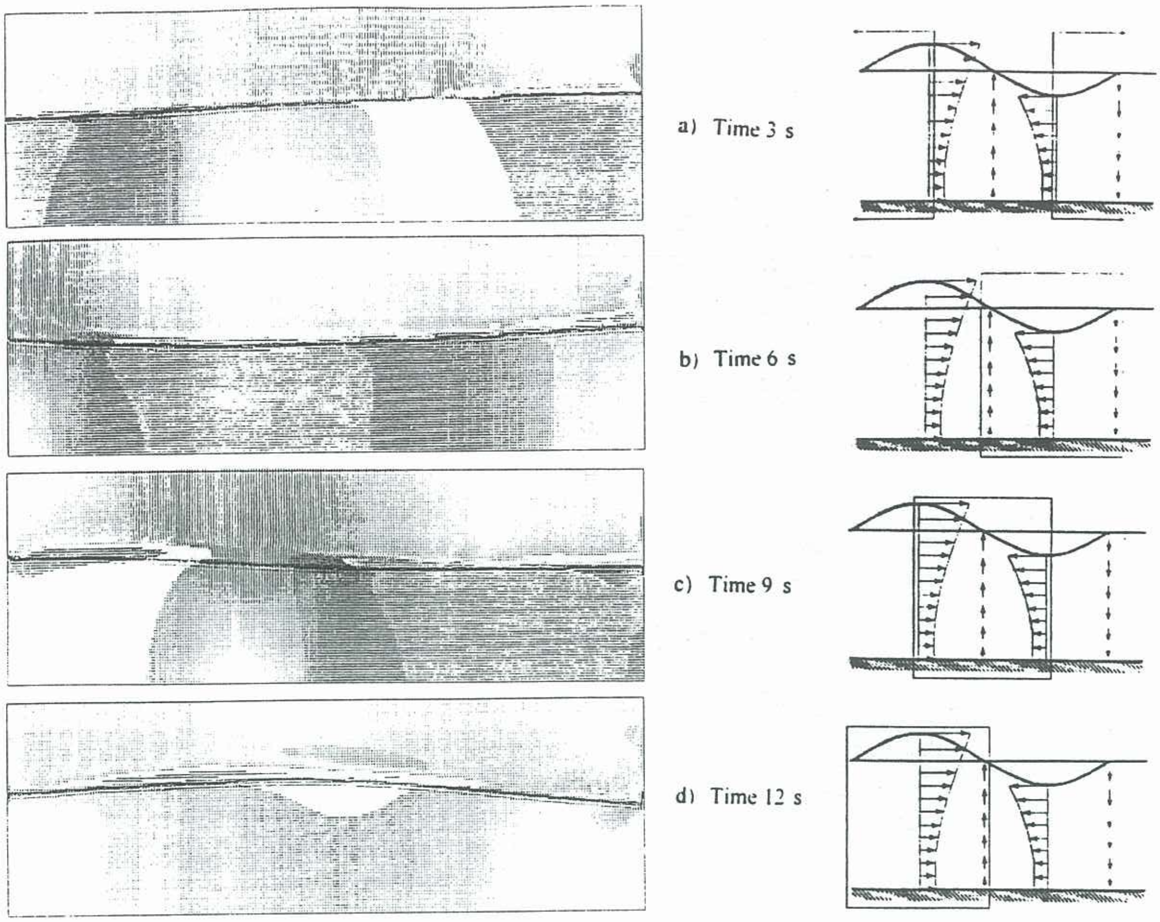


Figure 2: Wave Profile Generated in Full Solution Space

The primary vortex appeared to be generated as the fluid accelerated over the top of the cylinder, its momentum carrying it downstream of the cylinder and creating a low pressure region behind the cylinder. This generated a near bed recirculation stream flowing upstream towards the cylinder from which the vortex grew.

The secondary vortex was generated from the high pressure region in front of the cylinder. A small volume of fluid in this region was observed to flow towards the cylinder, and to deflect under the cylinder as it neared it. The area close to the bed generated a recirculation current flowing upstream from the local high pressure region under the cylinder. This stream however, was observed to dissipate quickly into the stronger bulk fluid flow.

From these mechanisms of formation of the vortices, an understanding of the relative strengths of the vortices formed and their directions of circulation was achieved.

The effect of the vortices on sediment transport was analysed by determining bulk fluid velocity profiles required to entrain critically sized particles. Komar's (1976) correlations were used to determine if initial particle motion could be obtained. To determine if the particle could be retained in suspension its fall velocity was compared with the maximum vertical velocity at the edge of the boundary layer.

Comparative analysis was also made for correlations for rippled beds made by Nielsen (1979) and sediment transport rates by bedload as documented by Sleath (1982).

This analysis showed that the conditions generated by the primary vortex are able to cause scour about the cylinder.

End Effects

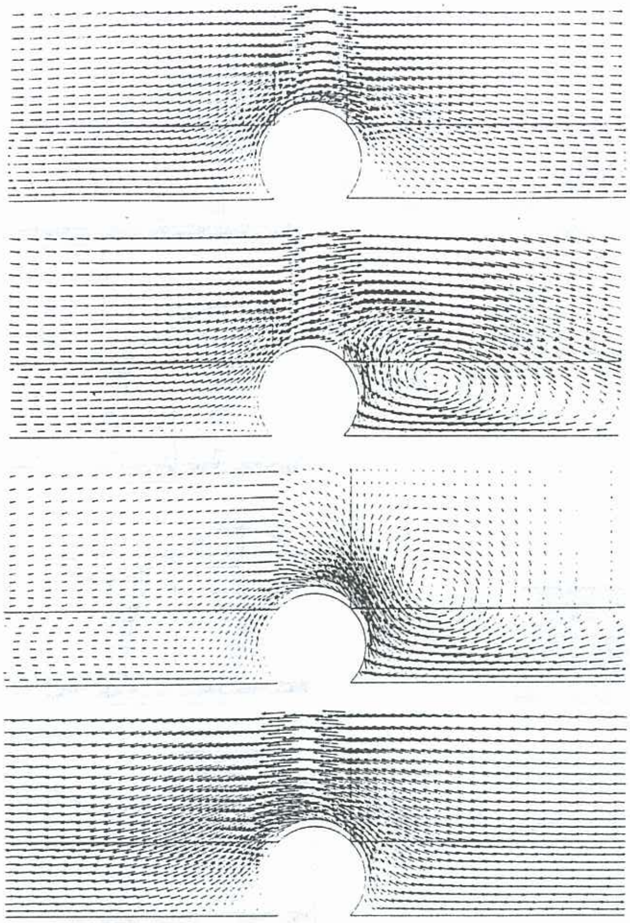
The full 3 dimensional analysis showed that flow patterns about the end of the cylinder were significantly different from those generated either in the bulk fluid, or along the bulk cylinder. The transition regions were found to extend approximately 2 diameters out from the cylinder into the bulk fluid, and 2 diameters back up the cylinder before bulk conditions were again encountered.

Investigation focused on analysing results when the vortices were at their largest. At this point the vortices were ejecting into low bulk fluid flow conditions and their impact on sediment transport was most pronounced.

At the peak of vortex ejection, local fluid velocities exceeded those of the bulk fluid, in both the 2 and 3 dimensional cases. The momentum of the vortex fluid flowing towards the bulk of the cylinder created a region of high pressure close to the cylinder. In the 2 dimensional case the fluid current flowing towards the bulk cylinder was observed to turn upwards and back into the bulk flow above the cylinder.

Two key results were observed in the transition regions of the 3 dimensional case:

1. Penetration of the vortex recirculation upstream of the cylinder, and
2. Flow generation parallel to the cylinder. (i.e. perpendicular to the bulk fluid flow).



a) Time 0 s — Vortices formed under maximum bulk fluid velocity. Fluid velocities in vortex regions slower than bulk fluid velocity.

b) Time 3 s — Vortices reach maximum intensity as bulk fluid slows. Both vortices are observed to rotate in the same direction, and the primary vortex is significantly larger than the secondary vortex. The primary vortex also generates bed velocities greater than the velocity of the bulk fluid.

c) Time 3.7 s — The vortices shed into the bulk fluid as the bulk fluid flow reverses direction. The secondary vortex is destroyed before the primary vortex.

d) Time 6 s — As the flow reaches a maximum in the reverse direction, the process of vortex generation begins again on the opposite side of cylinder.

Figure 3: Vortex Ejection about Bulk Cylinder

At this point in the wave cycle the vortex current dissipation into the bulk fluid assisted the development of a vortical flow past the end of the cylinder. A helical path for entrained sediment was developed by these currents, providing a mechanism for the removal of sediment from the end of the cylinder.

During periods of high bulk fluid flow, conditions for scour on the upstream face of the cylinder end were present. Currents parallel to the axis of the cylinder formed at the front of the cylinder channelling sediment from the cylinder ends.

The results discussed in this section show that significant scour occurs at the ends of the cylinder and that the sediment removal from this area is permanent. It has been suggested that scour about the ends of similar objects on the ocean bed results in their burial by a "rocking" mechanism. This occurs as the object ends alternatively sink into the sand following successive periods of end scour. The 3 dimensional models generated and analysed in this work supports this type of burial process.

CONCLUSION

An investigation into the process of mine burial on the sea bed was presented. By using CFD, the Navier-Sokes equations in turbulent flow were solved, and flow patterns and disturbances about objects on the seabed predicted. Vortices were found to exist either side of the mine during the wave cycle while strong fluid currents formed about the end of the mine. An accurate wave profile was generated over the entire solution space from first order boundary conditions.

ACKNOWLEDGEMENT

This work was part funded by the Defence, Science and Technology Organisation. The support of Phil Mulhearn is also gratefully acknowledged.

REFERENCES

CFX, Solver Manual, CFX, AEA Technology, Didcot, Oxon., OX11 0RA, UK.

HERBLICH, J.B., SCHILLER, R.E. (Jr), WATANABE, R.K., DUNLAP, W.A., "Seafloor Scour. Design Guidelines for Ocean-Founded Structures." *Marcel Dekker, Inc.*, (1984).

KOMAR, P.D., "Beach Processes and Sedimentation." *Prentice Hall, Inc.*, (1976)

LONGUET-HIGGINS, M.S., "Mass Transport in Water Waves". *Philos. Trans. Royal Society*, Series A 245. pp535 - 581, (1953)

MULHEARN P.J., "Experiment on Mine Burial By Scour", *Research Laboratory Technical Note MRL-TN-632*, 1993.

MULHEARN P.J., "An Experiment on Mine Burial by Seabed Scouring Under Wave Action", *Defence Science and Technology Organisation Technical Note DSTO-TN-0006*, March 1995.

NEILSEN, P., "Some Basic Concepts of Wave Sediment Transport.", Institute of Hydrodynamics Hydraul. Eng. Tech., University of Denmark, Series Paper 20.

SLEATH, J.F.A., ELLIS, A.C., "Ripple Geometry In Oscillatory Flow." University of Cambridge, Department of Engineering Report A/Hydrol. /TR2, (1978)