CFD Validation of Impact loading on a Rectangular Vertical Cylinder following a Free Surface Dam Breaking Event

P. Arnold, Minerva Dynamics Limited, Bath, UK

Abstract

A free surface flow dam breaking event is numerically simulated using the commercial CFD code Flow-3D and the results compared against experimental data to assess its accuracy and computational cost. The configuration simulated is representative of a large broken wave, or bore resulting from a tsunamis, interacting with a coastal structure. The results are likely to be of interest to coastal and offshore engineers and marine energy developers with an interest in CFD. Laminar simulations were carried out on a series of three progressively finer meshes and using a series of progressively smaller time steps to assess grid and time step dependence. The influence of the three momentum advection schemes and two of the turbulence models available were also tested. The results found to be in excellent agreement with experimental data after taking into account experimental uncertainty, whilst incurring a relatively low computational cost.

Keywords: CFD, dam break, Flow-3D, free surface, validation, wave loading, vertical cylinder

*Authors email address : peter.arnold@minervadynamics.com

Introduction

The 3D dam breaking event with rectangular vertical cylinder test case simulated here corresponds to an experiment described by Gomez-Gesteira and Dalrymple 2004, [1] known as a "bore in a box". The experiment was originally derived to validate Smoothed Particle Hydrodnamics (SPH) simulations and now forms part of the SPHERIC database of tests cases for validating SPH and CFD methods [2,3]. A similar dam break test case using a smaller rectangular object located in a larger tank has also been extensively used by the CFD and SPH communities for validation [4]. However as pointed out by Arnold 2013 [5] and Lobovsky 2014, [6] the experimental uncertainty and repeatability of both Dam Break problems has not been assessed and so the margins of error into which numerical simulations should fall are nonexistent. Lobovsky conducted a detailed analysis of the experimental repeatability for a generalised Dam Break problem, by placing pressure sensors in the end walls of a tank with no object in and conducted over 100 repeat experiments whilst varying factors which would have been expected to change the results such as the gate opening times. They found significant variation in the peak pressures measured but could not attribute this to any one particular physical cause. They did however find increased variability and breakdown of two dimensionality as the initial water height increased. Stansby et al 1998 [7] and Janosi et al 2004, [8] concluded that the complexity of the flow and the propagation of the bore down the tank is affected by the wetness of the tank bottom wall ahead of the gate due to low level leakage under the gate. A simple visual inspection of the flow confirms its complexity and consequent sensitivity to small changes in boundary and initial conditions. Also from a statistical viewpoint since peak values have larger degrees of variability than average or integrated quantities, we must take this into account when comparing our numerical simulation results to the experimental results.

The level of agreement between previous CFD simulation based on the Navier Stokes equations and experiment for the original Dam Break Problem [4] , has in general been very good despite the flow's complexity for example Arnold 2013 [5] used an earlier version of the Flow-3D code whilst Kleefsman et al 2005 [9] used the COMFLOW code and compared results across four surface elevation sensors and 8 pressure sensors located on the object. The areas of less than good agreement occurred where the experiment signal fluctuated very rapidly and could possibly be attributed to non unique vales for the surface elevation sensors or where the pressure peaked, which can be explained by the reasons discussed earlier. SPH simulations of the current test case have also had similar success for example Gomez et al 2004 [1] and Silvester and Cleary 2006 [10].

Experimental Setup

The experiment consist of a volume of water 40cm in length and 30cm high and 61cm wide, initially confined by a gate at one end of a tank which is 160cm long, 61cm wide and 75cm high. A square cylinder of width 12cm and height 75cm is placed 50cm downstream of the gate and 24cm from one side wall and 25cm from the other. Due to difficulties with the gate sealing the experimentalists reported a layer of water between 0.5cm and 1cm deep lining the remaining bottom of the tank. The apparatus is shown in Fig 1.

Measurements of time histories of the X direction (along centreline of the tank from water to object) force on the cylinder were made using a load cell. The X direction fluid velocity at a point on the tanks centreline 14.6cm upstream of the cylinder and 2.6cm above the tank floor was measured using Laser Doppler Velocimetry (LDV) . The experimental data is compared with simulations over the first 3 seconds. The Reynolds number of the case can be estimated based on a length scale is taken to be the distance between the gate and the end wall equal to 1.2m and a velocity based on $V = 2\,[gH]^{0.5}$ where g is gravitational acceleration an H is the initial water elevation equal to 0.3m, hence a Reynolds number of 4.2×10^6.

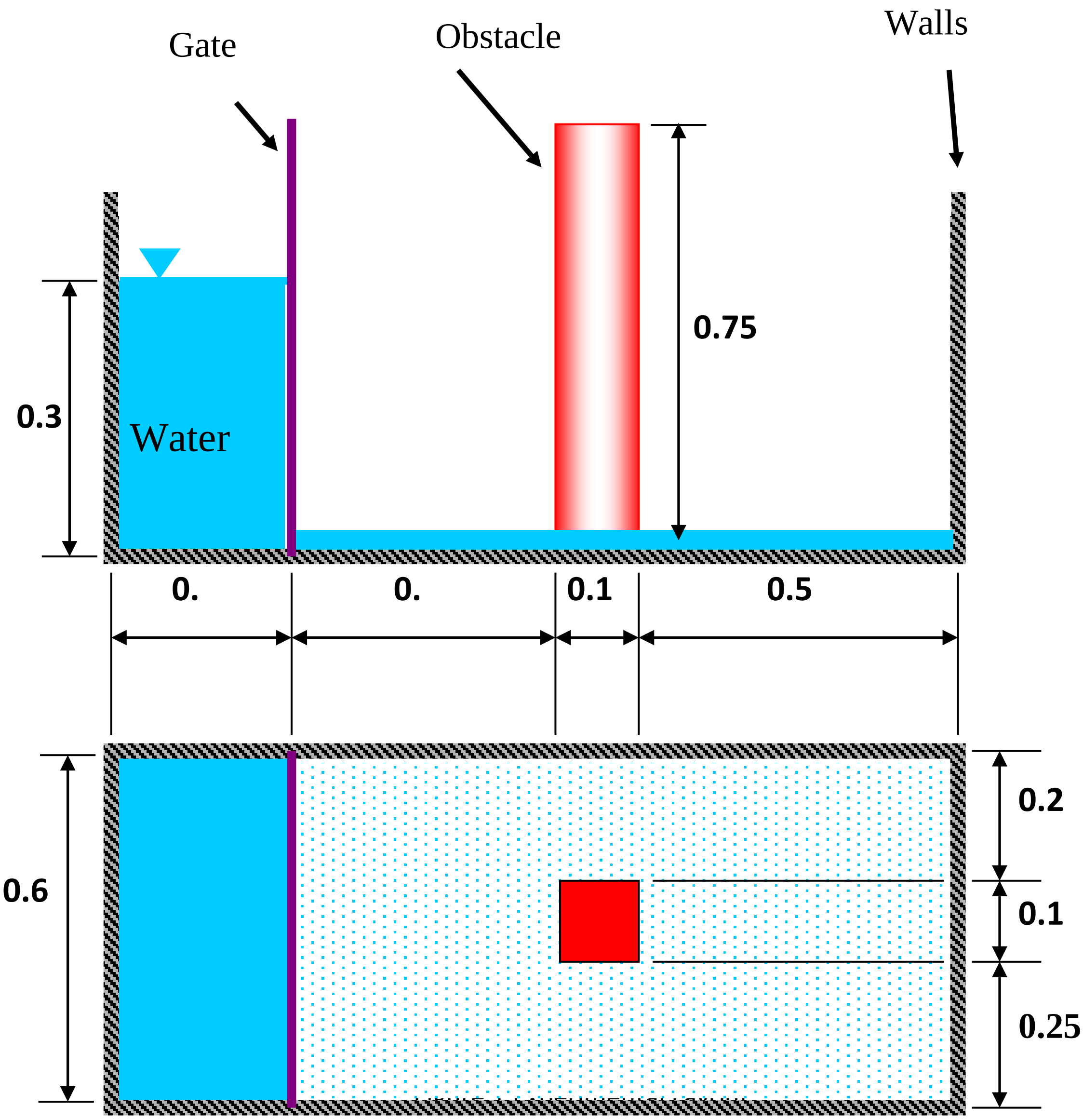

Fig 1.Experimental Layout

Simulation Methodology

The numerical simulations were set up to match the experimental geometry exactly, with all six sides of the tank and the rectangular object defined as no slip walls. The water behind the gate was represented at time zero as a block of water with a hydrostatic pressure profile. However as no gate was actually present in the simulation the pressure was atmospheric at the gate end of the block and was interpolated between the gate and hydrostatic end wall pressure. This would be the actual pressure profile milliseconds after the gates was released and so it was not considered to be of any consequence. An initial layer of water 0.75cm deep with a hydrostatic profile was defined along the remaining floor of the tank in keeping with the experiment.

The default coarse mesh was composed of 1cm sized hexahedral cells with a spacing of 160 in the x direction , 61 in the y direction and 75 in the z direction and was uniformly spaced other than for accommodating fixed points to coincide with the rectangular obstacle and sensor locations, hence a total of approximately 785,000 cells. After prescribing the initial location of the water and its viscosity, laminar time dependent simulations were carried out for a series of three progressively finer uniform meshes with cell sizes of 1cm, 0.66cm and 0.5cm and cell counts of 785K, 2.6 million and 6.1 million. A bespoke mesh was also used based on the medium 0.66cm background mesh with a nested 0.33cm mesh around the rectangular object consisting of 5.9 million cells in total.

The time step refinement study used constant time steps of $3X10^{-4}$ s, $2X10^{-4}$ s and $1X10^{-4}$ s. This can be compared to the default automatic time step controlled by stability and convergence criteria which when deployed on the coarse mesh produced time steps of approximately $1X10^{-3}$s for most of the run with a minimum value of $5X10^{-4}$ s.

The three momentum advection schemes tested were, Ist Order, 2nd order and 2nd order monotonicity preserving schemes. The two turbulence models tested and compared with the laminar simulations were the RNG and k-ω models. We have not attempted to satisfy the usual turbulence model associated constraints on the distance of the nearest node from the tank walls due to the excessive demands this would put on the mesh resolution. Also as the flow is largely chaotic and the obstacle is sharp edged the prediction of the flow separation effects will be driven by the rapid changes in geometry rather than by a gradual separation of an orderly boundary layer. Consequently, we have assumed that resolution of the boundary layer is less relevant than resolving the flow in the interior of the domain from the viewpoint of predicting the main flow features.

Unless otherwise stated all simulations used the coarse mesh and the automatically adjusted time step, Ist order momentum advection and assumed laminar flow. The solution was advanced in time using a first order time discretization along with and the automatic VOF method option and implicit GMRES pressure solver.

The simulation was carried out for a total of 3 seconds of real time and the time histories of the X-direction force on the rectangular object and X direction fluid velocity in front of it at the experimental sensors location were recorded and plotted against the experimental data. In an effort to remove the variability associated with peak measurements the fluid force was integrated on the rectangular object in time to obtain the cumulative total impulse in both the experimental and CFD calculations. The CPU and elapsed times were also recorded on the 12 core twin processor Xeon X5650 workstation with 72GB of RAM running Flow-3D V 11.0.2.03.

Results

It is noted that both the experimental force and velocity time histories have been shifted in time by -0.084 and 0.254 seconds respectively. This was justified by the fact that the experimental times given were clearly incorrect and resulted in the a velocity maximum at time zero which is physically impossible, and a force time history which otherwise would have a constant offset from the CFD results. The need to apply a time offset has been found by other authors when comparing this data with SPH simulations [10].

The mesh refinement study results are shown in figs 2,3 and 4. The instant of the highest positive force occurs around 0.34 seconds, shown in figure 5, when the fluid draws level with the rear edge of the cylinder after first impacting on it. The CFD calculation of the forces evolution closely follows the experimental time trace up to around 1.4 seconds and underestimates the peak positive x direction force by 17% or less with mesh refinement. By the time the bore returns to impact the cylinder in the negative x direction at around 1.45 seconds, shown in figure 6, it is clear that the experimentally measured force on the cylinder is fluctuating rapidly in time as the flow becomes increasingly chaotic and the smaller negative direction peak force magnitude is consistently overestimated on all meshes with no convergence with mesh refinement.

The cumulative impulse also shows good agreement up to 1.45 seconds and estimates the maximum change in momentum well on all meshes after which there is an underestimation which corresponds to the over prediction of the negative X direction force on the cylinder. The x-direction velocity before the cylinder follows a similar trend with the peak positive velocity being overestimated by up to 10%.

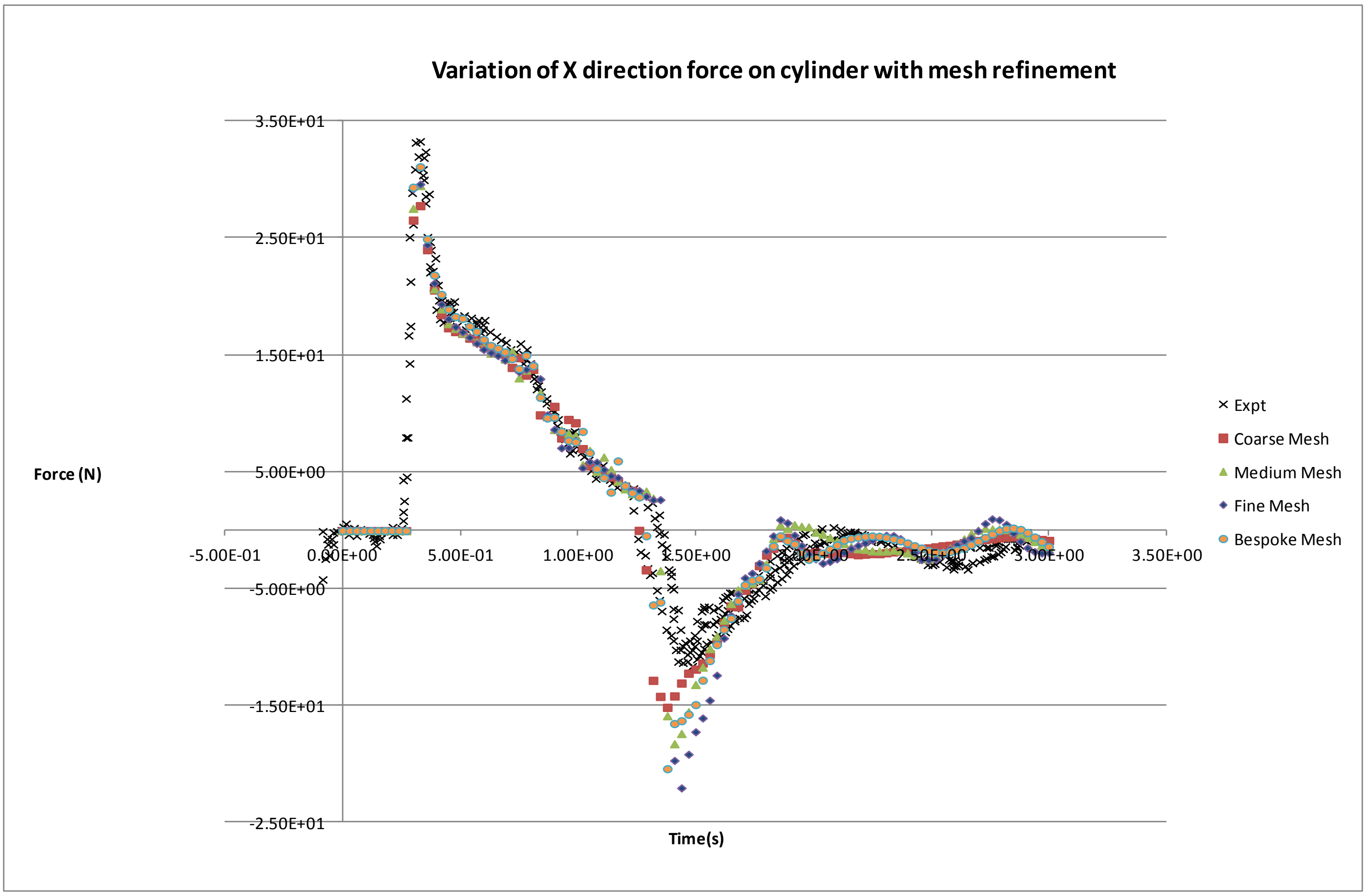

Figure 2 X direction force on cylinder with mesh refinement

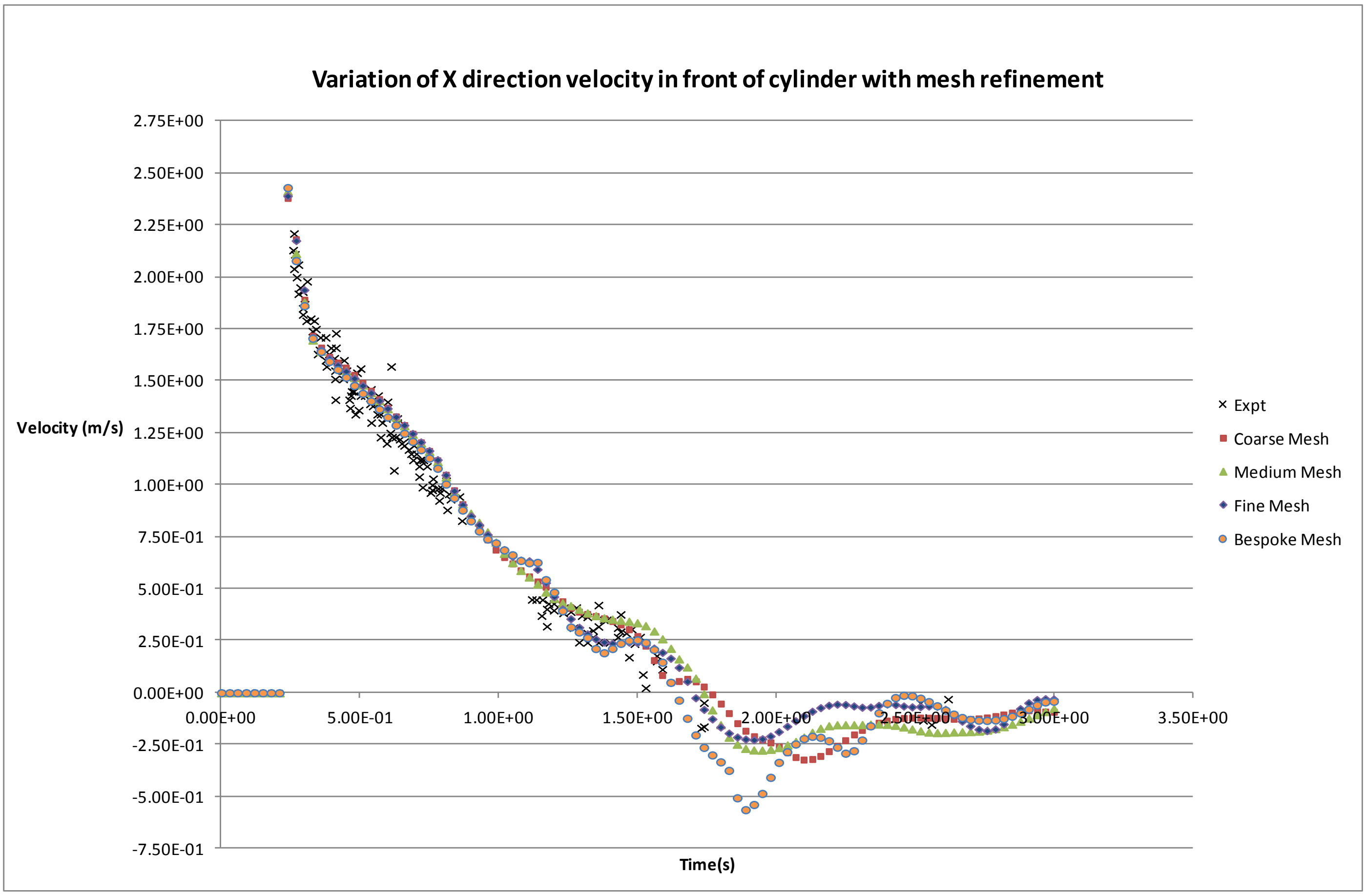

Figure 3 X direction velocity in front of cylinder versus mesh refinement

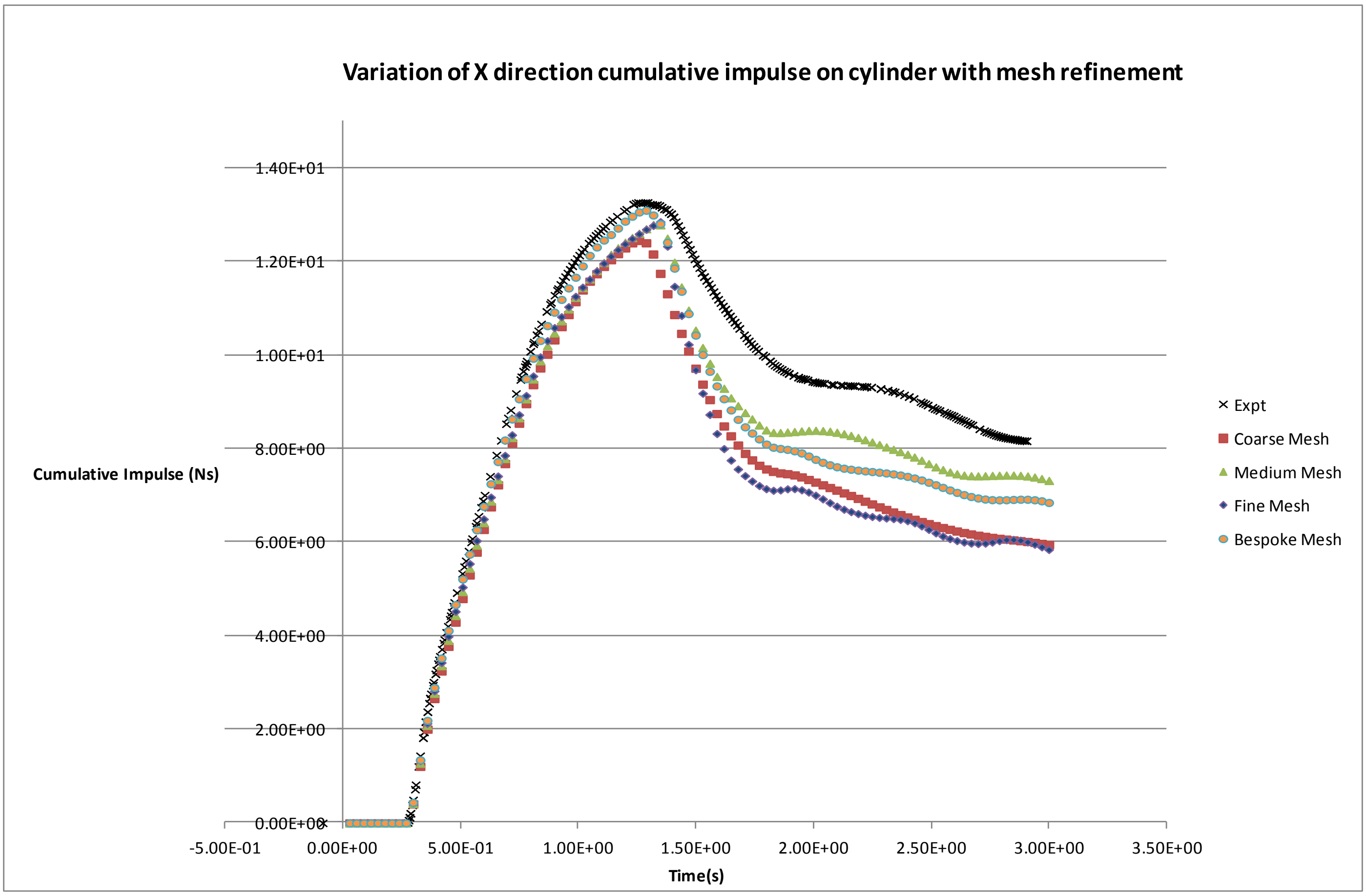

Figure 4 X direction cumulative impulse on cylinder versus mesh refinement

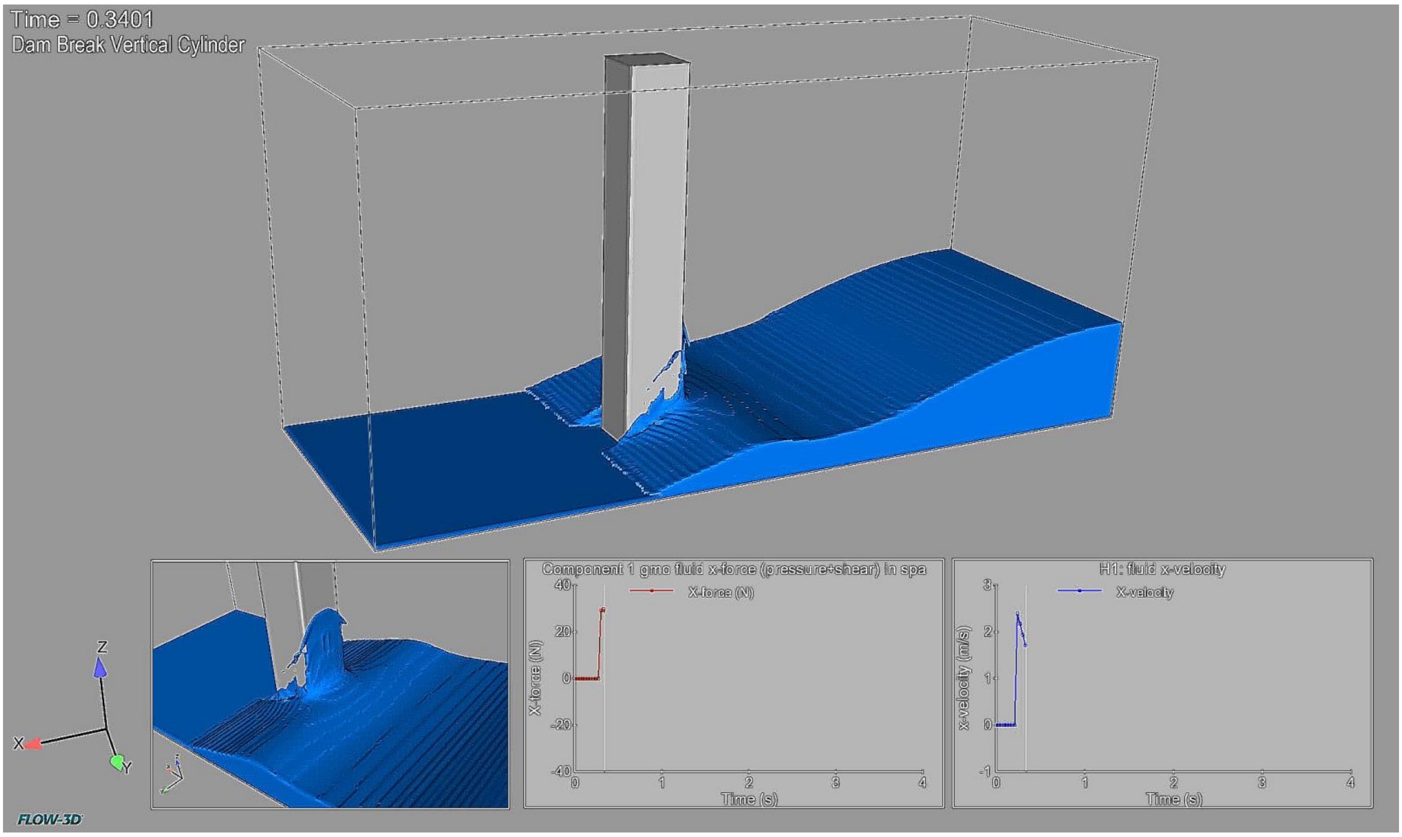

Figure 5 Instant of peak positive x direction force

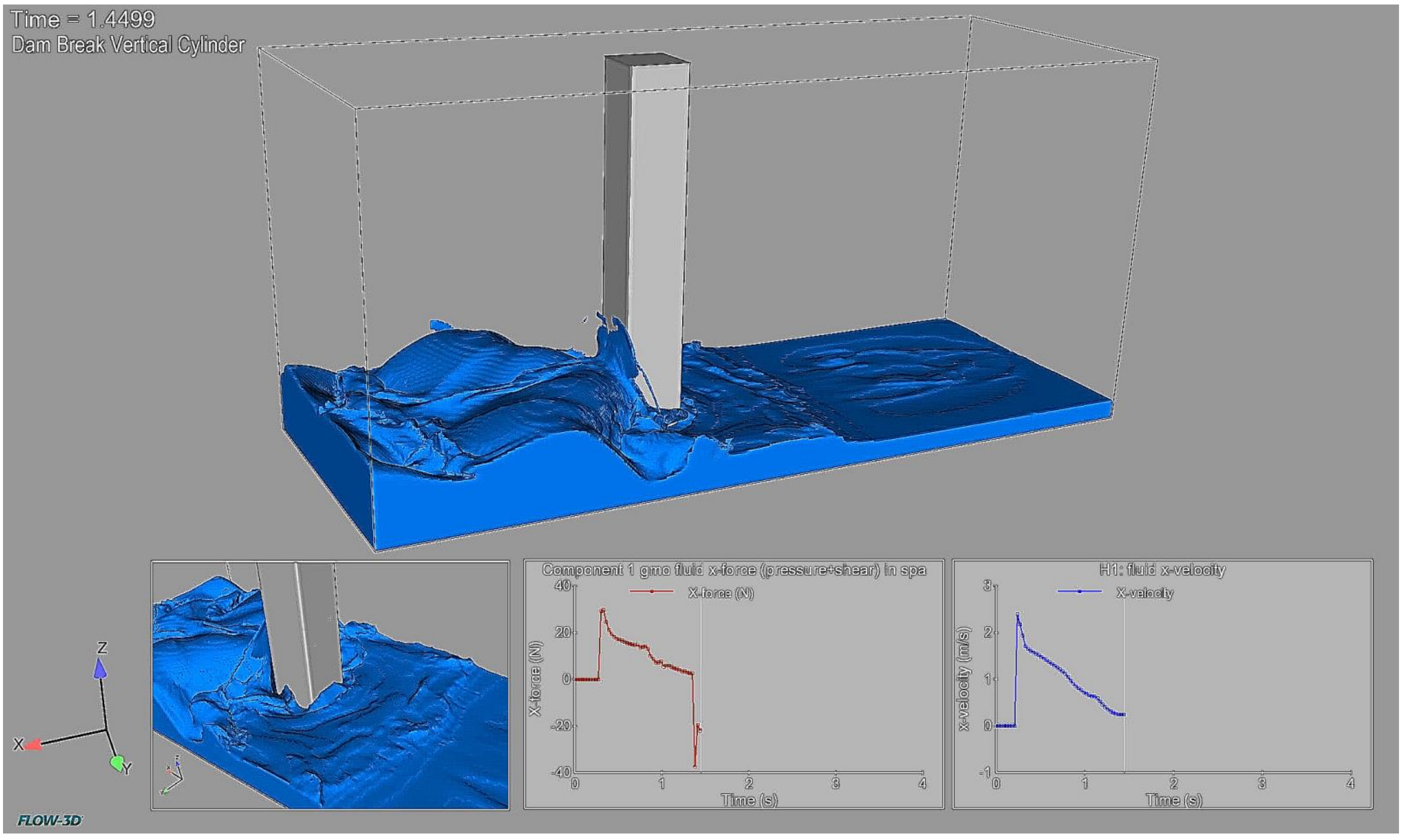

Figure 6 Instant of peak negative x direction force

The time step refinement study (figs 7,8, and 9) shows very little sensitivity to the time step size on the coarse mesh used and the solution is effectively limited by the mesh resolution. There appears to be no advantage to not using the automatic time step deployed in the default method, provided the minimum time step is set to be small enough as the run times appear to ne inversely proportional to the size of the uniform time step size.

Increasing the order of the momentum advection scheme (figs 10,11 and 12) provides no apparent convergence in the results with the higher order schemes producing more scatter in both force and velocity about the experimental points. The cumulative impulse results also appear to deteriorate for the higher order schemes, the reason for this is not clear. Similarly the use of turbulence models produces very little change from the laminar results.

Table 1 summarises the values of maximum and minimum values in the force, impulse and velocity and the associated run times of each configuration.

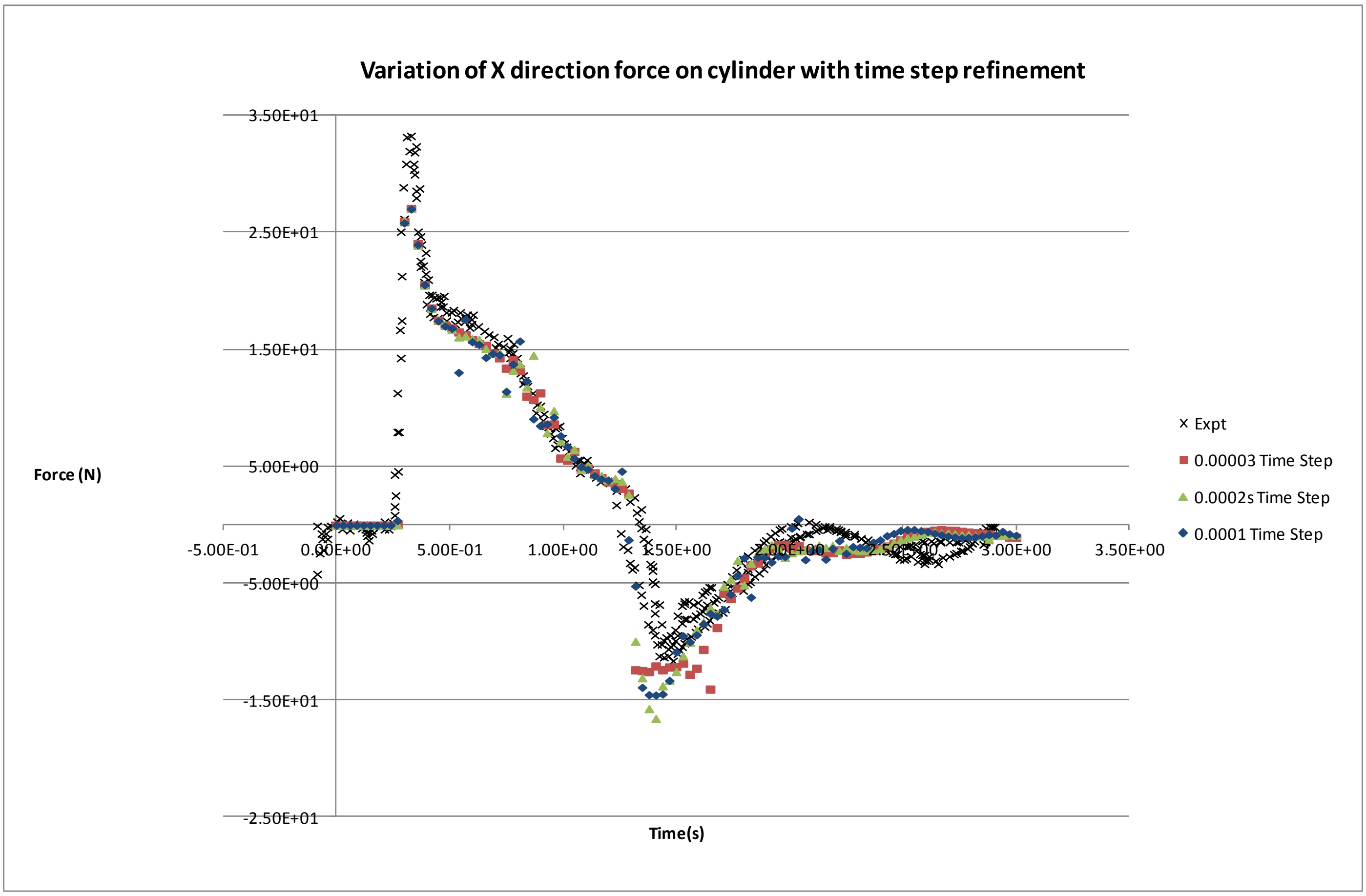

Figure 7 X direction force on cylinder versus time step refinement

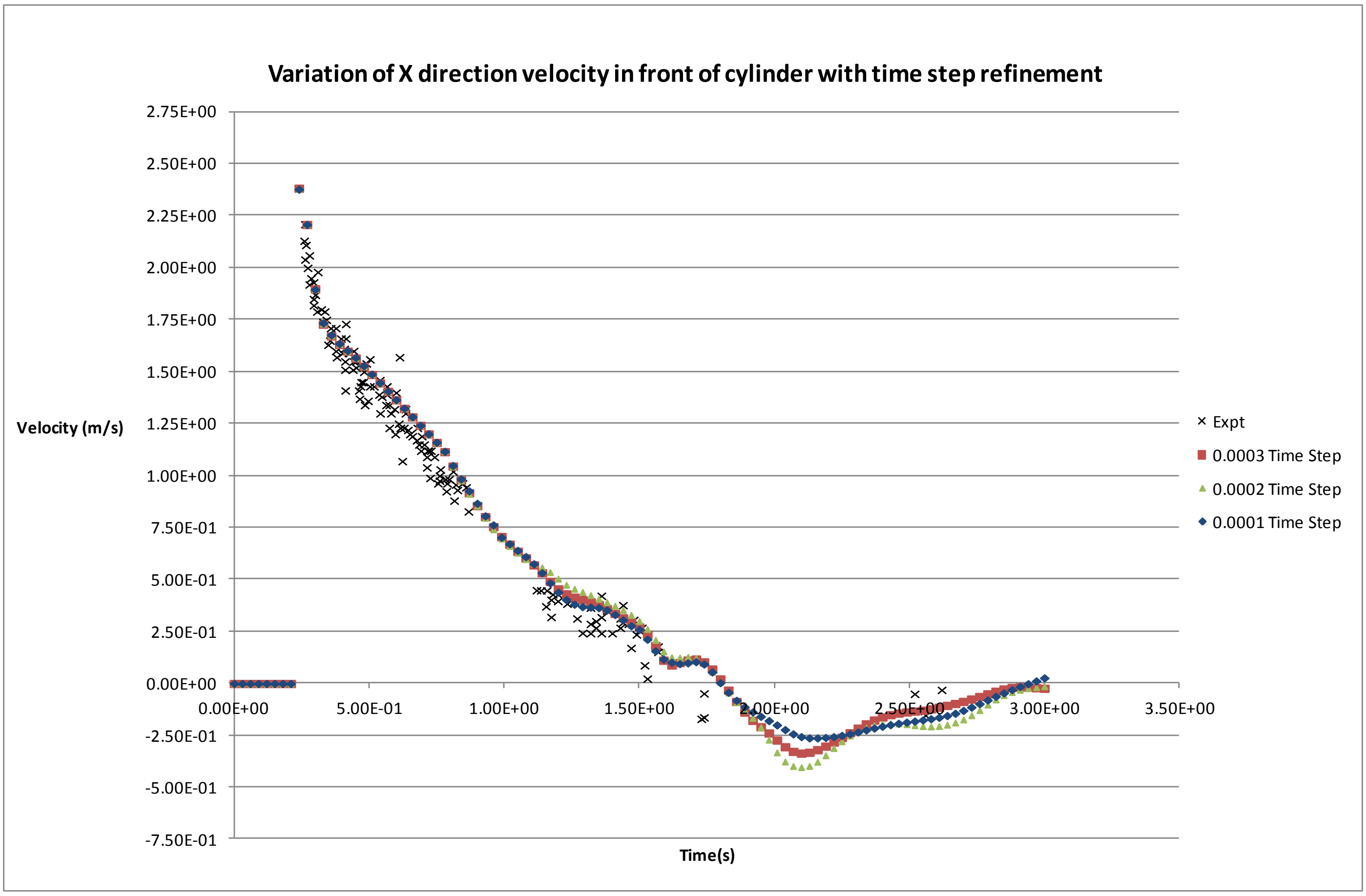

Figure 8 X direction velocity in front of cylinder versus time step refinement

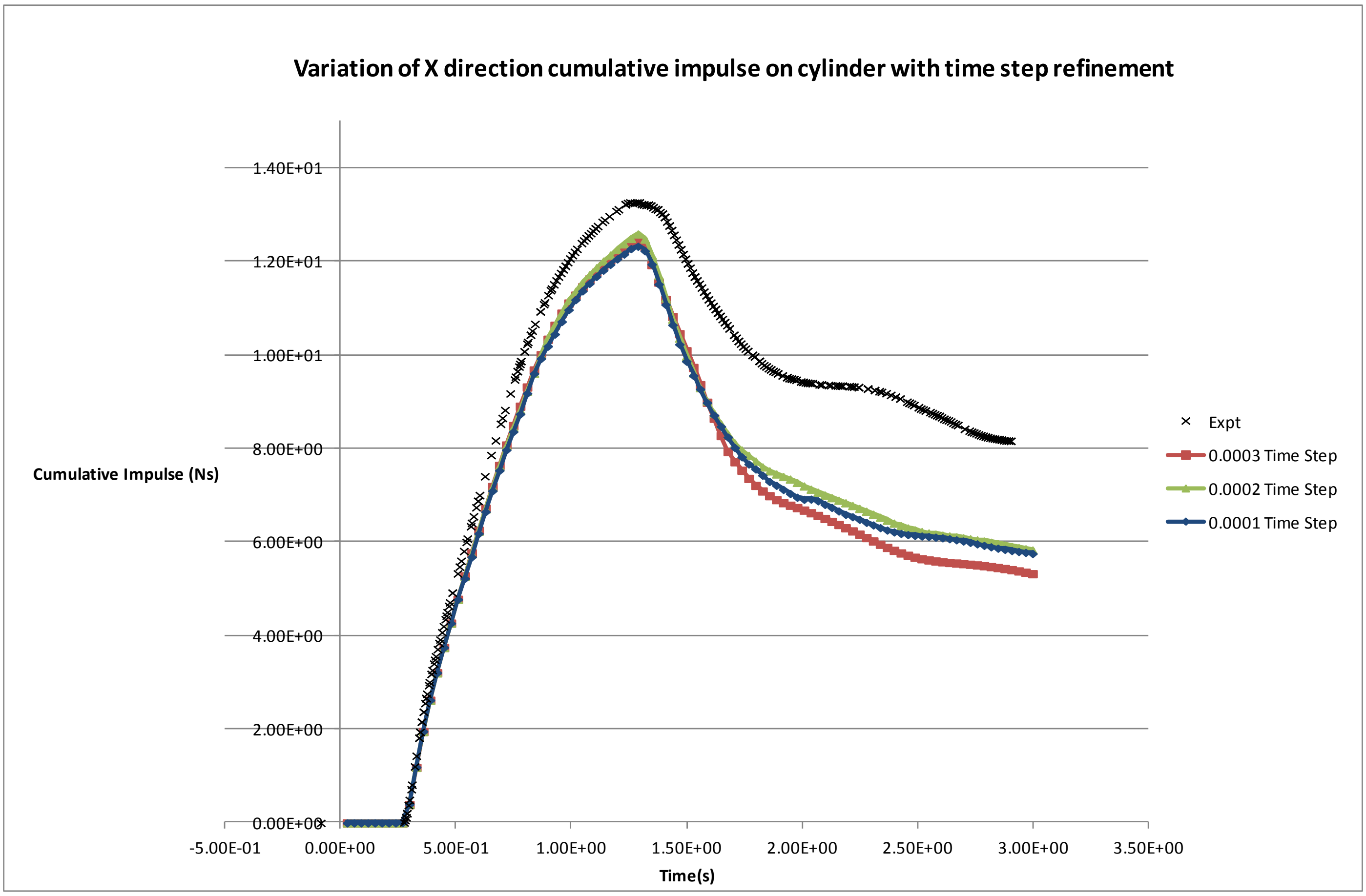

Figure 9 X direction cumulative impulse on cylinder versus time step refinement

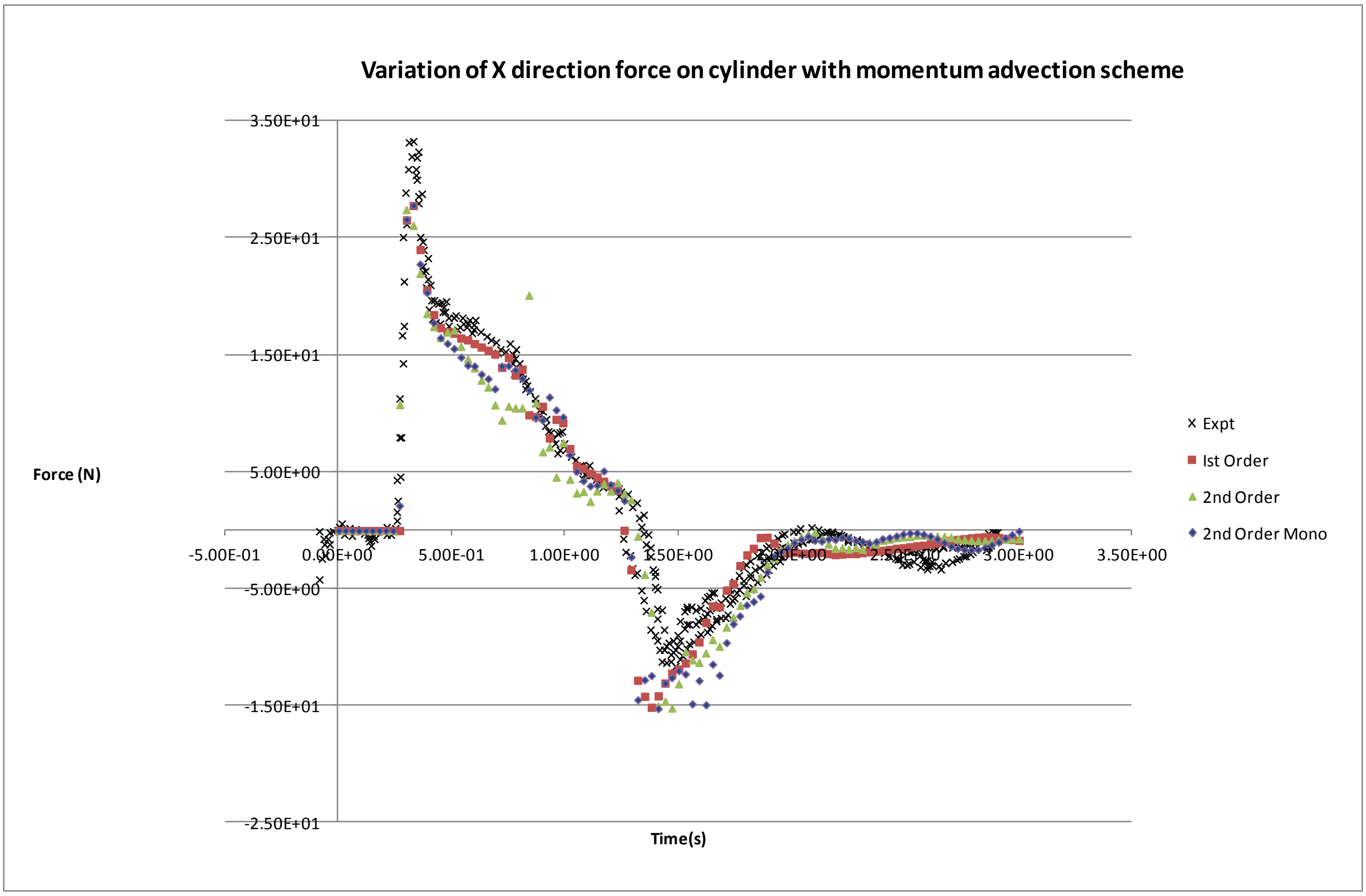

Figure 10 X direction force on cylinder versus momentum advection scheme

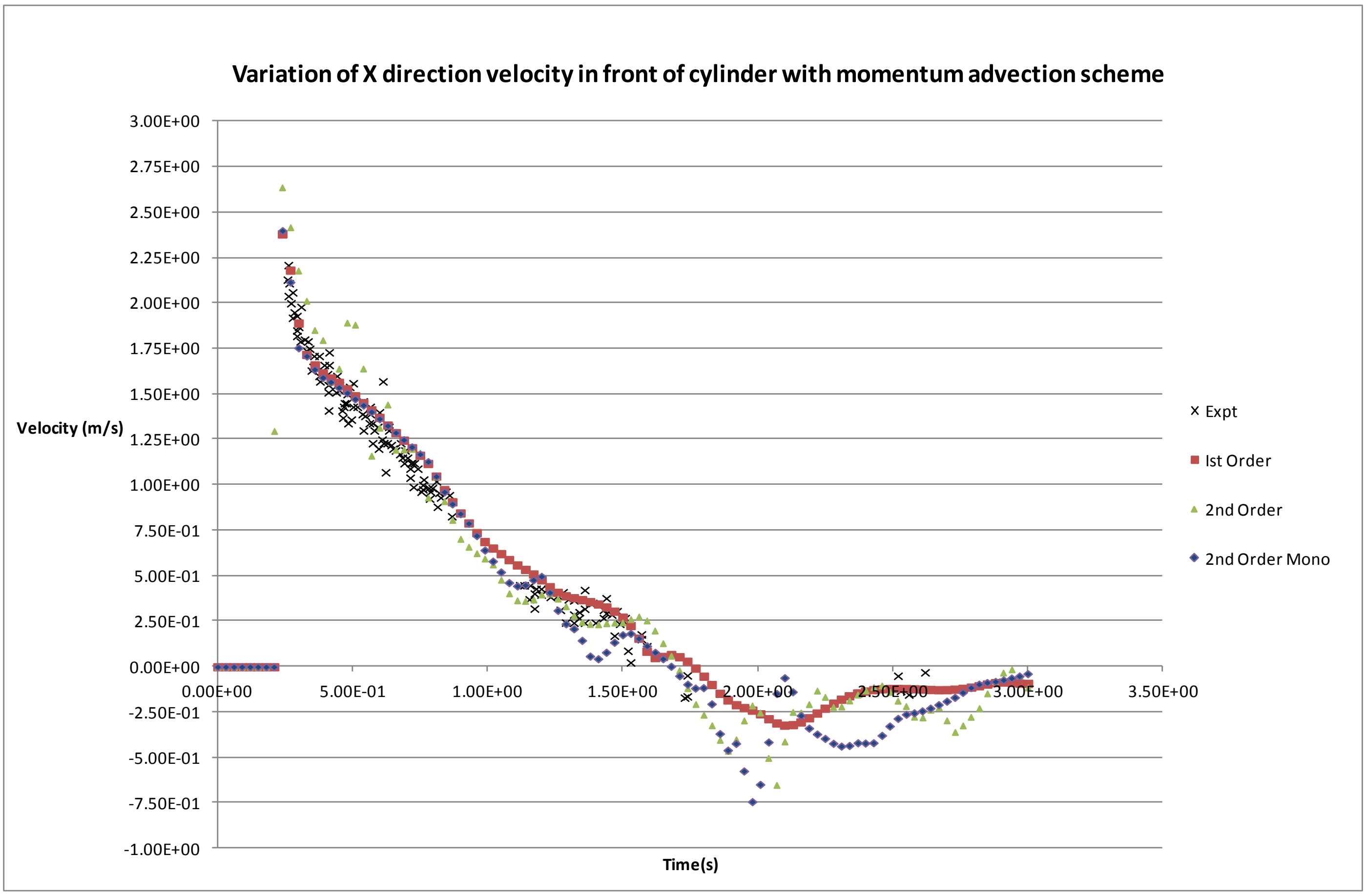

Figure 11 X direction velocity in front of cylinder versus momentum advection scheme

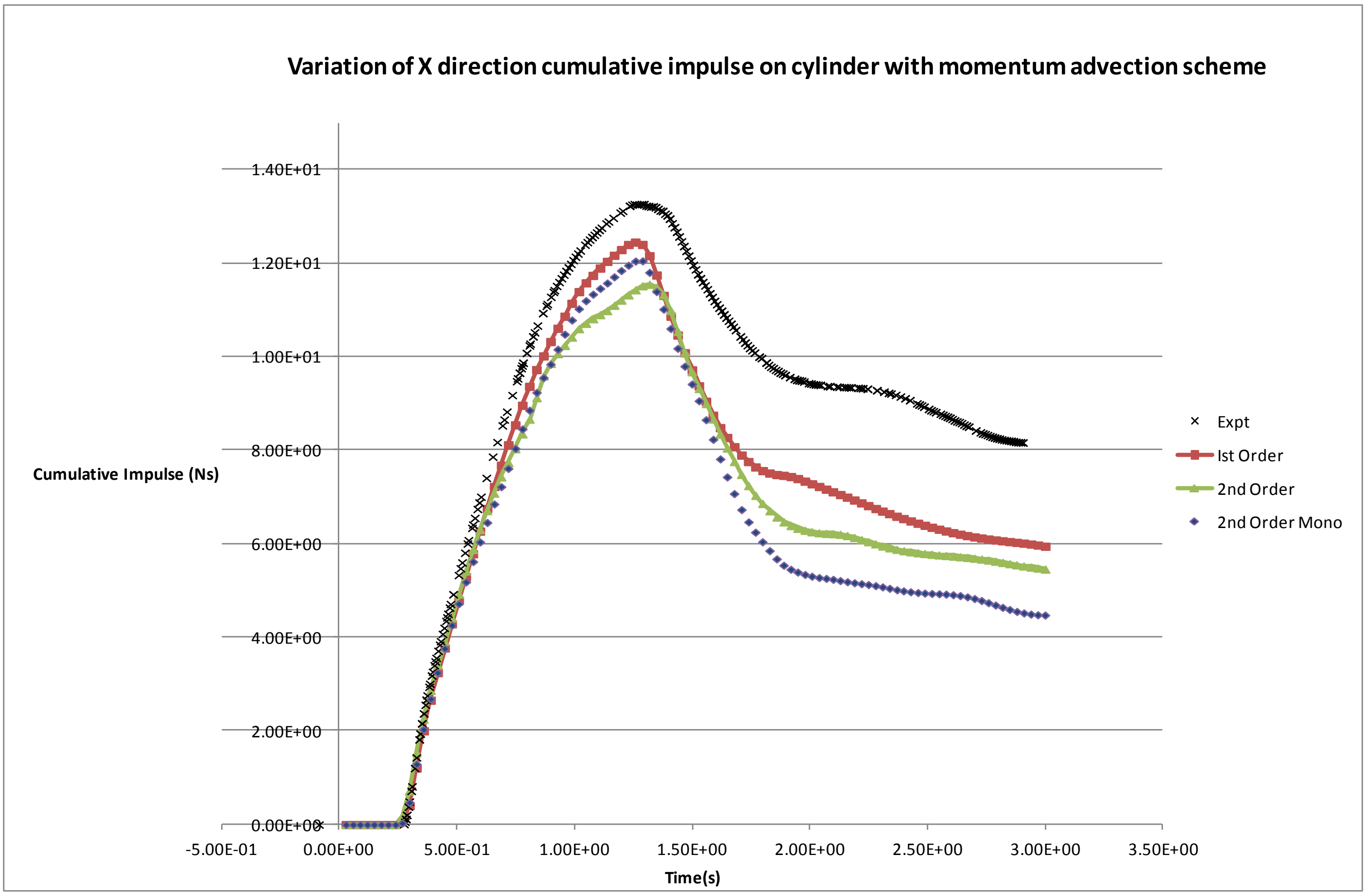

Figure 12 X cumulative direction impulse on cylinder versus momentum advection scheme

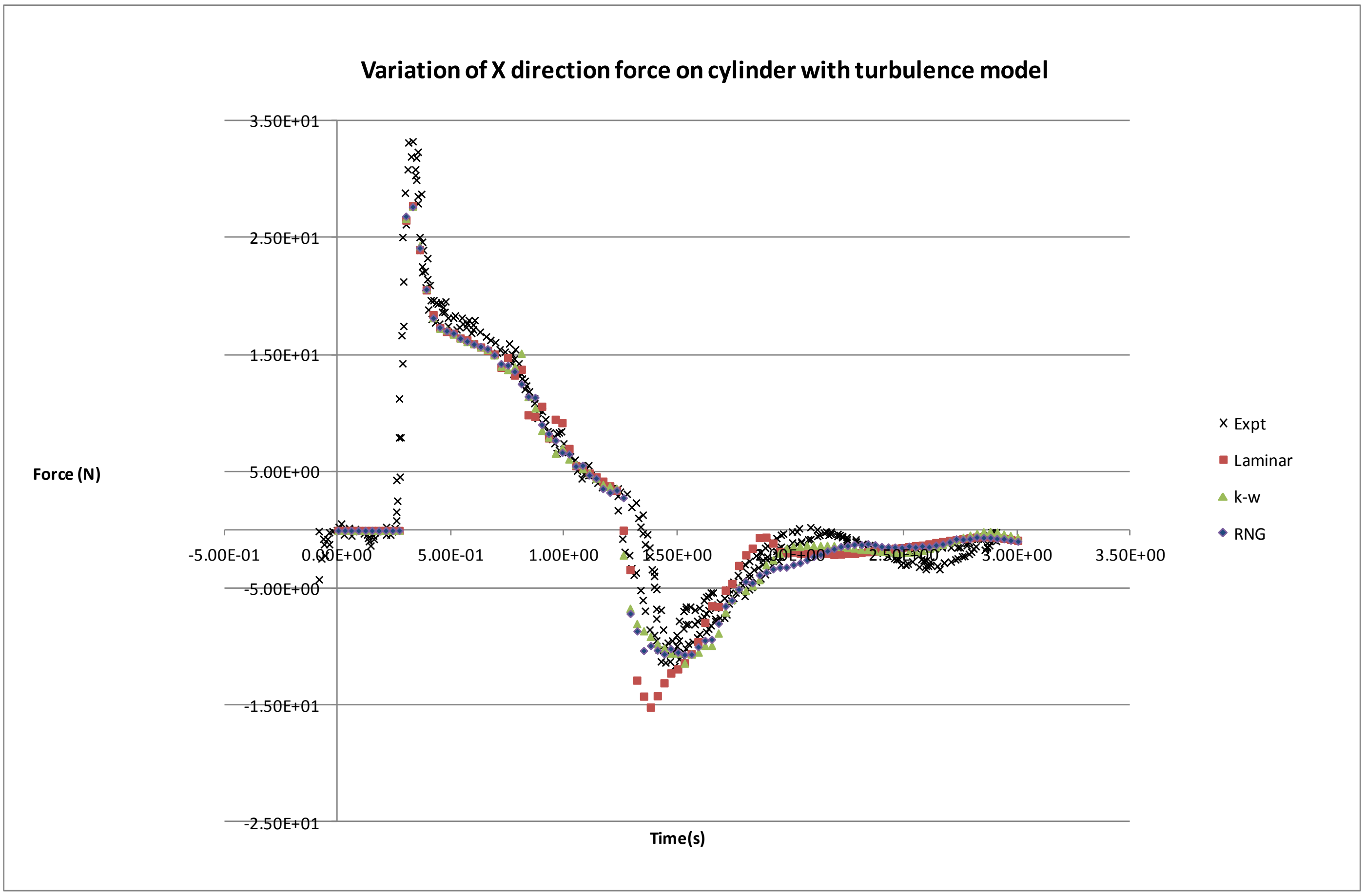

Figure 13 X direction force on cylinder versus turbulence model

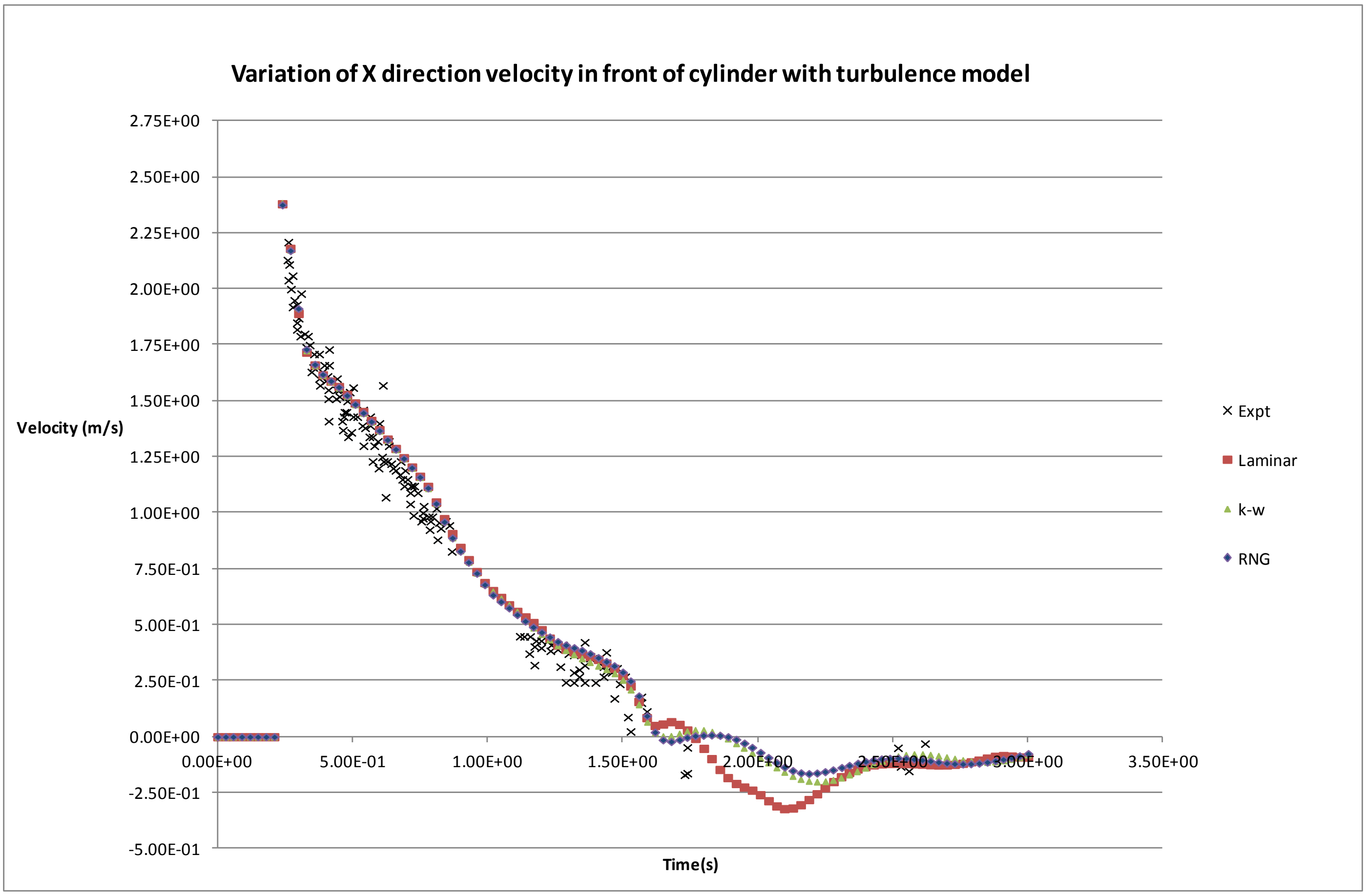

Figure 14 X direction velocity in front of cylinder versus turbulence model

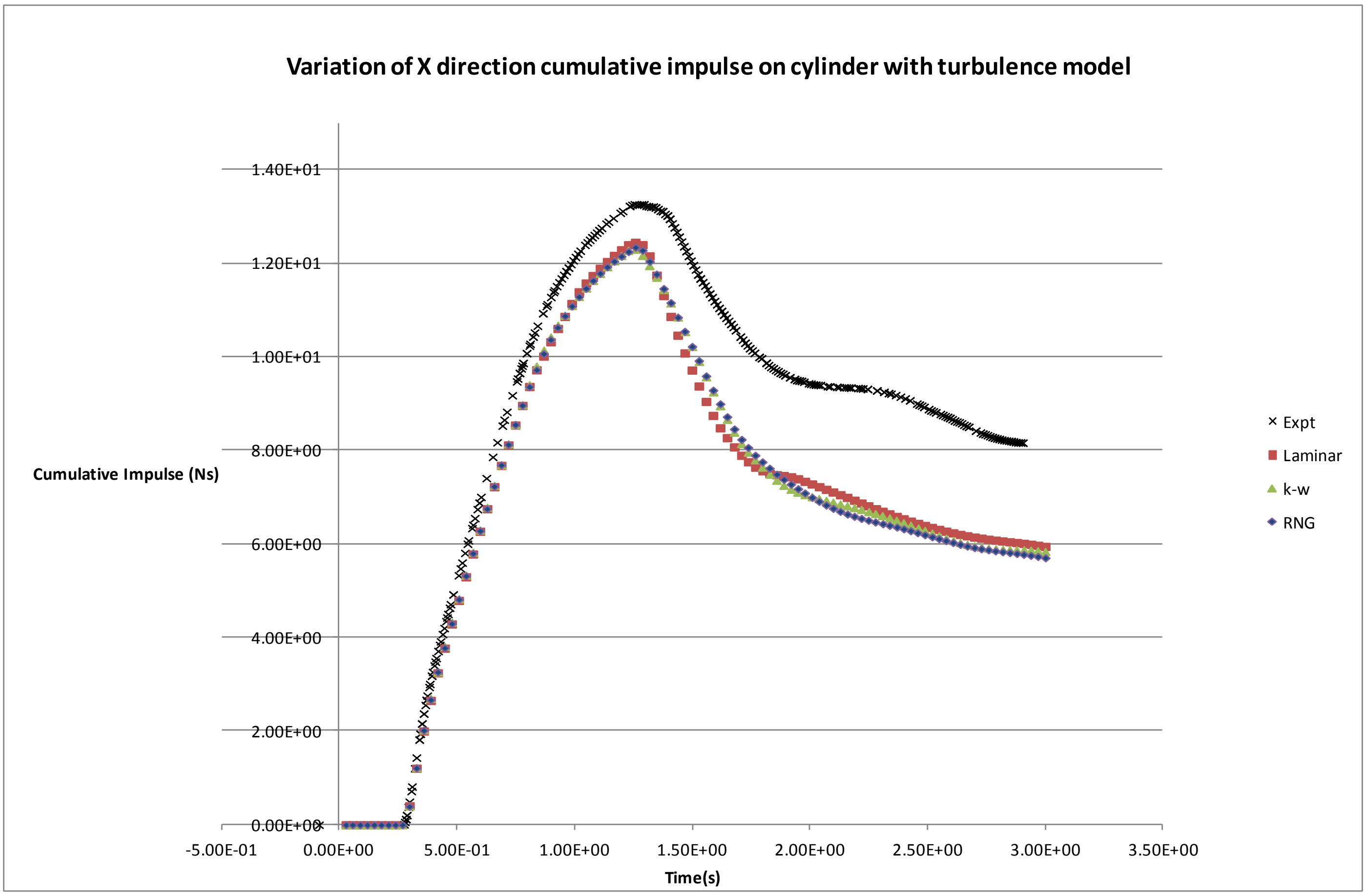

Figure 15 X direction cumulative impulse on cylinder versus turbulence model

Table 1 Mesh and time step independence tests

	Max Force (N)	**Min Force(N)**	**Max Cumulative Impulse (N.s)**	**Max Velocity(m/s)**	**Min Velocity (m/s)**	**Elapsed Time (s)**	**CPU Time(s)**
Experiment	33.3	-11.6	13.3	2.21	-0.17	n/a	n/a
Coarse Mesh	27.79	-15.12	12.5	2.38	-0.32	1266	7116
Medium Mesh	29.52	-18.23	12.8	2.41	-0.28	4107	32549
Fine Mesh	29.64	-37.54	12.9	2.39	-0.23	12663	110015
Bespoke Mesh	31.13	-20.37	13.1	2.43	-0.56	37103	369060
DT= 0.0003	**27.09**	**-14.02**	**12.5**	**2.38**	**-0.34**	2421	23974
DT =0.0002	**27.10**	**-16.53**	**12.6**	**2.38**	**-0.42**	3398	35632
DT=0.0001	**27.05**	**-14.53**	**12.3**	**2.38**	**-0.26**	6379	69684
Ist Order Momentum Advection	27.79	-15.12	12.5	2.38	-0.32	1266	7116
2nd Order Momentum Advection	**27.45**	**-15.19**	**11.6**	**2.64**	**-0.65**	1094	8791
2nd Order Mono Momentum Advection	**27.81**	**-15.23**	**12.1**	**2.40**	**-0.74**	1078	8607
Laminar	27.79	-15.12	12.5	2.38	-0.32	1266	7116
k-ω	**27.74**	**-11.35**	**12.3**	**2.39**	**-0.20**	940	7114
RNG	**27.70**	**-10.63**	**12.4**	**2.38**	**-0.17**	943	7021

Conclusion

Taking all the plots and tabular results into consideration it appears that there is a weak correlation of increased maxima with increased mesh resolution as one might expect. The reason for the increased scatter about the mean values produced by the 2nd order momentum advection scheme and the less accurate prediction of the cumulative impulse by both higher order schemes is not clear. However as discussed previously the repeatability of this particular experiment is unknown and so we cannot know for certain if our numerical simulations fall into the expected range of the experiment and whether or not the increased scatter of the 2nd order scheme has any physical basis. From a qualitative viewpoint the time histories resulting from the CFD simulations even on the coarsest mesh follow the experimental time histories remarkably closely considering the complexity of the flow field.

From an engineering viewpoint the parameter of the most importance is the maximum load on the structure and possibly the cumulative impulse up to that point, both of which are well estimated by the CFD simulations. The coarsest mesh solution with a run time of just over 15 minutes on a desktop workstation would be sufficiently accurate in terms of estimating these parameters and designing the cylinder to withstand such an impact. The point of least agreement occur when the flow complexity is at its maximum which occurs midway through the event.

References

1. Gomez-Gesteira, M., and Dalrymple, R.A., (2004) " Using a 3D SPH Method for Wave Impact on a Tall Structure, J.Waterway., Port, Coastal and Ocean Eng. 130(2) 63-6

2. http://cfd.me.umist.ac.uk/sph

3. http://cfd.mace.manchester.ac.uk/sph/TestCases/SPH_Test1.html

4. SPH European Research Interest Community SIG, Test-Case 2 3D dambreaking, R.Issa and D. Violeau. http://wiki.manchester.ac.uk/spheric/index.php/Test2

5. Arnold, P. "Validation of Flow-3D against Tank Tests for the Wavebob ", Workshop on CFD for Wave Energy Applications , NUI Maynooth , Jan 2013.

6. Lobovsky, L., Botia-Vera, E., Castellana, F.,Mas-Soler, J., Souto- Iglesias, A., " Experimental investigation of dynamic pressure loads during a dam break", Journal of Fluids and Structures, Vol 48, July 2014, pp 407- 434

7. Stansby, P. K., Chegini, A., Barnes, T. C. D., Nov. 1998. The initial stages of dam-break ow. Journal of Fluid Mechanics 374, 407{424.

8. J_anosi, Jan, D., Szab_o, K. G., T_el, T., Aug. 2004. Turbulent drag reduction in dam-break ows. Experiments in Fluids 37 (2), 219{229.

9. K.M.T Kleefsman, G. Fekken, A.E P Veldman, B. Iwanowski, and B. Buchner. A volume of fluid based simulation method for wave impact problems. J Comp Phs, 206: 363-393, 2005.10.

10. Silvester, T., Cleary, W., " Waver-structure interaction using smoothed particle hydrodynamics ", Fith International Conference on CFD in the Process Industries, CSIRO, Melbourne, Australia, 13-15 December 2006