INTERNATIONAL JOURNAL OF INNOVATIVE RESEARCH & DEVELOPMENT

ISSN: 2278 – 0211 (Online)

CFD Studies On Cylindrical Bodies Moving Concentrically Inside Very Long Tubes

Dr. R.Vijayakumar
Ph.D. , Indian Institute of Technology, Delhi, India
Adjunct Faculty , Naval Construction Wing, Department of Applied Mechanics at Indian Institute of Technology, Delhi, India
Lt . S K Rao
M Tech in Ocean Engineering and Naval Architecture form IIT Kharagpur,India
Pursuing doctoral research in the field of underwater explosions and its effect on the structures
Lt R.V.Shashank Shanka
pursuing a PG Diploma in Naval Construction from Indian Institute of Technology, Delhi

ABSTRACT:

The near field flow and drag estimation on cylindrical bodies moving concentrically inside tubes in an incompressible medium always fascinated Naval Platform designers while designing large projectile launchers. This paper presents the flow field numerical simulation on the published experimental work by Naval Underwater System Center, Newport, using the commercially available CFD software FLUENT based on the Reynolds averaged Navier Stokes Equations and k-epsilon turbulence model. Axisymmetric model was used in the simulation without considering appendages such as fins, rudders and propeller. Numerical analysis of such complex confined flows involving relative motion of the bodies is a challenging task. Dynamic mesh was used to solve the relative movement problem of the projectile and tube using user define programme to update the mesh. The results presented include, the characteristics of flow field in the annulus gap, pressure distribution on the moving object and the relation between increased drag and the annulus gap. The simulation results can be employed as reference in designing of such projectiles.

1.Introduction

The purpose of this paper is to demonstrate CDF simulation using commercial software in near field flow studies on the cylindrical bodies moving concentrically in an incompressible medium. The experimental work by Naval Underwater System Centre, Newport has been chosen for the CFD simulation.

Published literature, contemporary to this field of annulus flows is related to train-tunnels, where a high-speed vehicles (usually trains) travel in enclosed guide ways (usually tunnels) for significant distances. Since the vehicles travel at sufficiently low velocity it is reasonable to regard the air medium as essentially incompressible (Mach number less than 0.2).The physics of the train-tunnel and projectile motion in tube are such that fluid is forced (by the naturally developing pressure drop) to flow down the annular gap between the vehicle and the guide way to fill the void created at the back of the vehicle. Motion of the fluid is strongly coupled to the motion of the vehicle through the conservation mechanisms of mass and momentum. Specifically, an estimate of drag on the vehicle is desired to estimate the initial thrust required to eject the projectile out of the tube. Total drag is composed of two separate parts, i.e., shear drag and pressure drag and its effect due to the confined flow

Experimental studies were conducted by Richard[1] using cylindrical bodies with hemispherical noses and conical tails in a suitable drop test apparatus, using a water filled tube. The experimental set up designed and fabricated by Richard[1], similar to Hoppe and Gouse [2] and later by Nayak [3]. By closing the bottom end of the tube it was possible to simulate an infinitely long tube. In an infinitely long tube there is no induced axial flow in the tube due to its infinite resistance to flow. A suitable guide way and release mechanism was designed and constructed for the model test.

The schematic diagram of the experimental setup is shown in Fig.1. Results were obtained for gap Reynolds numbers (based on the annular gap) up to 2E4. High speed photography was used to construct a position vs. time history for the model from rest to terminal velocity. Terminal velocity was sufficiently high to ensure that turbulent flow had been established in the annular clearance.

Figure 1: Drop test experimental set up[1]

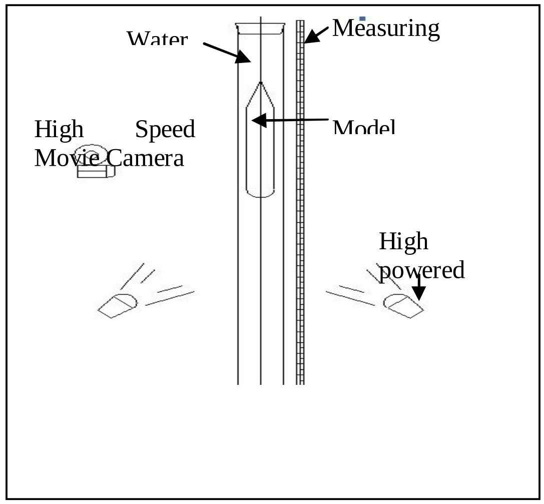


Figure 1: Drop test experimental set up[1]

The experimental results of distance vs time of falling were available for the drop test carried out for three bodies (ß=1.6, ß=1.33 and ß-1.14) while falling inside a 1.000" tube. They are called 625, 750 and 875, which corresponds to outside diameters of 5/8, 3/4 and 7/8 inches, respectively as shown in Fig 2. All the bodies had a hemispherical nose, cylindrical mid-body, and a conical tail(also referred to as the nose, straight and tail sections respectively) without any added appendages. Table 1 lists relevant model geometries and properties, which were used for experiments. This paper describes the CFD simulation of the experimental work.

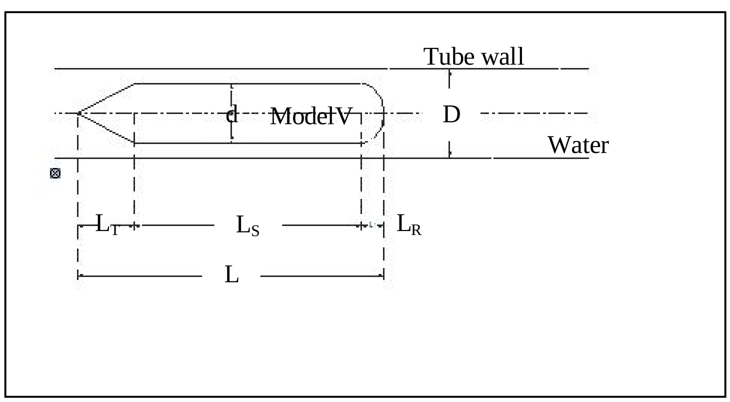


Figure 2: Model geometry

Model Parameters	Body 625	Body 750	Body 875
Diameter, d (in.)	0.625	0.750	0.875
Center hole, d_h (in.)	0.125	0.125	0.125
L_R (in.)	0.313	0.375	0.438
L_S (in.)	5.433	3.453	1.588
L_T(in.)	1.38	1.375	1.485
$L=L_R+L_S+L_T$ (in.)	7.125	5.203	3.510
Frontal area, a (in.^{2})	0.307	0.442	0.601
Displaced Vol (in.^{3})	1.64	1.79	1.37
Dry weight, W (lb)	0.1640	0.1796	0.1377
Buoyant force, F_b (lb)	0.0591	0.0646	0.0494
Net wt, W_n=W-F_b (lb)	0.1050	0.1150	0.0884

Table 1: Model Specifications

2.Computational Domain And Mesh Generation

In a flow field simulation computation domain selection is a key factor which determines the complexity and successfulness of the problem. In the present work, the axi symmetry of the tube and the body were used and a 2D axisymmetric computational domain has been generate as shown in the Fig 3. This reduces the need for computational resource and time greatly and simulation can be performed just on a PC and results can be achieved quickly.

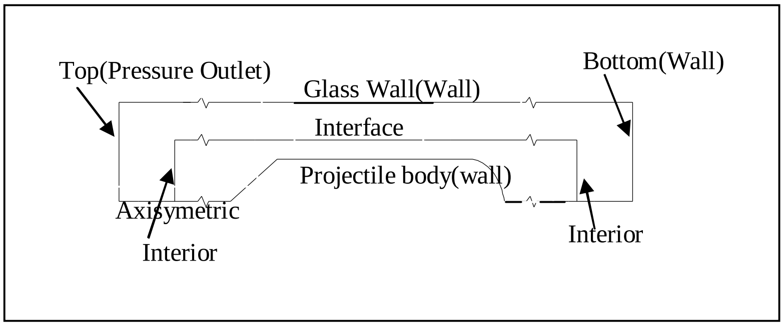


Figure 3: Schematics of the computational domain geometry

The mesh generator used in this work is Gambit2.3.16. It is an integrated pre-processor for CFD analysis from FLUENT Inc which can be used to generate unstructured meshes with high efficiency. The procedure of 2D axisymmetric model meshing in Gambit is almost same as normal 2D model, except that all the grid point coordinates should fulfill the requirements :$X\geq0$ and $Y\geq0$, and the model axis should be the X axis. The computational domain was split two sub zones and meshed with 500,000 mixed cells(triangular and quadilateral). Fig 4 shows part of the generated mesh in the tail and nose part of the model.

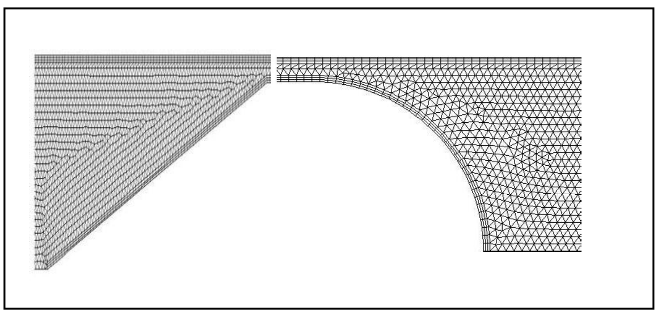


Figure 4: Generated Mesh

2.1.Numerical Solver And Boundary Conditions

In the computation the pressure based implicit unsteady solver is used and the pressure-velocity coupling algorithm is SIMPLE. For accuracy second order method was used for pressure, momentum and the turbulence viscosity discretization. User Defined Functions was used to calculate the total force acted on the body and the resulted body velocity. Moving mesh method is used in the simulation to handle the fluid structure interacting problem and the dynamic layering method is selected as the mesh updating method. The dynamic layering method is suite for prismatic (rectangular and/or triangular) mesh zones, the basic idea of it is that layers of cells adjacent to the moving boundary is added or removed based on the height of the layer adjacent to the moving surface. As in Fig 5 shows, the layer of cells adjacent to the moving boundary (layer j) is split or merged with the layer of cells next to it (layer i) based on the weight w of the cells in layer j.

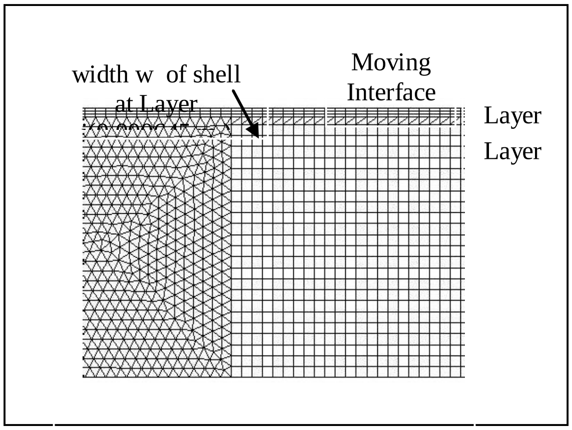


Figure 5: Schematics of dynamic layering mesh

The time step used is 0.0001 second and total time steps is 46000, the CFL number is less than 1 based on the time step and cell characteristic length and time convergence is ensured.

2.2.Boundary Conditions

The boundary condition setup in FLUENT is illustrated in Table 2.The projectile body, glass wall and the bottom are assigned as wall boundary. The Axis in and the axis out of the two domain are assigned as axi-symmetric. Top surface was set to pressure outlet to simulate the experimental condition. The interface zones created to accommodate the moving zones between the two zones with internal top and bottom as interior.

Geometry Boundary	**Boundary type**
Axis in	Axis
Axis out	Axis
Projectile body	Wall
Tube wall	Wall
Top Surface	Pressure outlet
Interface in	Interface
Interface Out	Interface
Internal bottom	Interior
Internal top	Interior
Outlet	wall
Buoyant force, F_b (lb)	0.0591
Net wt, W_n=W-F_b (lb)	0.1050

Table 2: Boundary condition

3.Result

The velocity and displacement of the projectile is recorded during the simulation. Comparison of the experimental data the computational data is made. As can be seen in Figs 6 , 7 and 8 for the three different objects with three different annular gap. The comparison results show good agreement between simulation and experiment and thus validity of the CFD model is achieved. The error were found to be less than 9% for all three models.

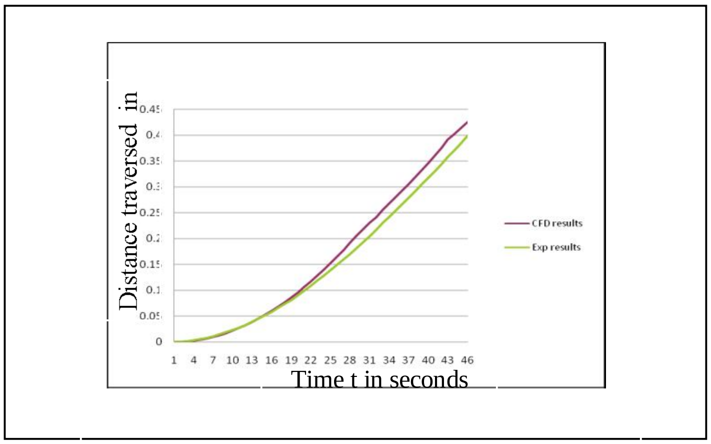


Figure 6: CFD and experimental results comparison for the 625 model

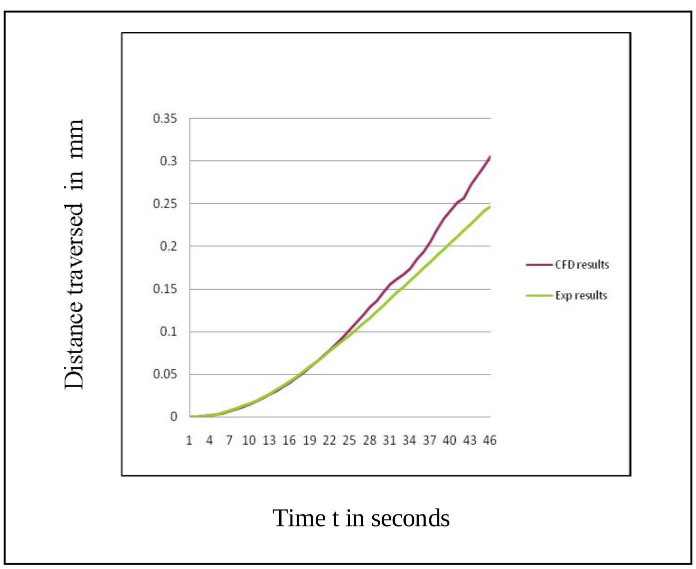


Figure 7: CFD and experimental results comparison for the 750 model

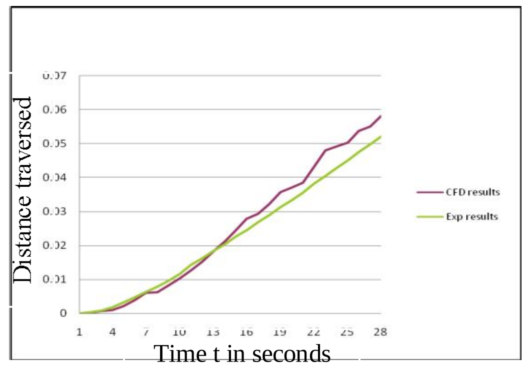
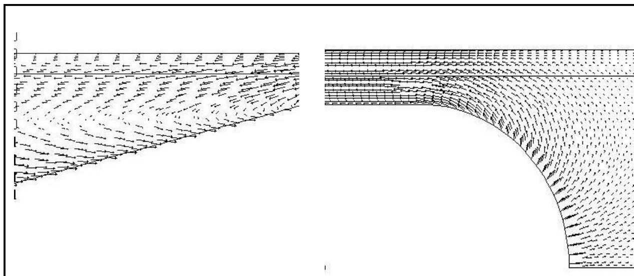


Figure 8: CFD and experimental results comparison for the 875 model

Fig 10 shows the vector plot of velocity . From Fig. 10(a) we can see that a the void created by the fwd movement of the model is filled with the water from the annulus gap. From the Fig 10(b) it is seen that, as the model moves forward , the water is accelerated and pushed back through the annulus gap between the model and the glass tube.



(a) Flow at Tail region (b) Flow near Nose

Figure 10: Velocity vectors around the model

Fig.11 shows the velocity contour around the body. The flow gets accelerated in the annulus region and shows the max velocity. The annular water jet is formed due to leakage from the annular gap between model and tube. In the model head zone a velocity increase also can be seen which results from the model movement and incompressibility of the water.

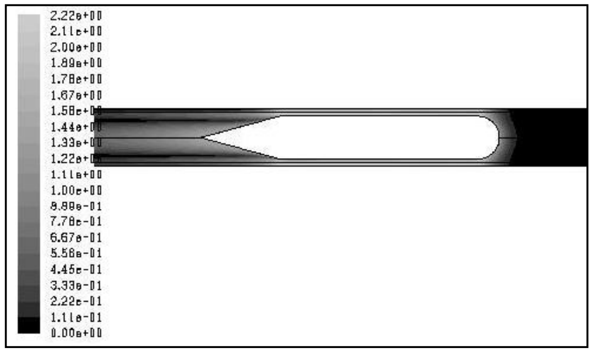


Figure 11: Velocity contours around the model

4.Conclusion

From this study of CFD application, we can conclude that CFD method is an effective way to study the flow characteristics of cylindrical bodies moving in confined tubes. The study can be extended to the projectile launch from tubes. The moving boundary problem is well resolved by using the dynamic mesh method. Future studies will focus on the transient dynamics of the pressure change and its influence on the launch system structures.

- **NOMENCLATURE**

ß= Diameter ratio D/d

D=Dia meter of the tube

D= Diameter of the projectile

L= Total length of the projectile

L_T = Axial length of the Tail

L_S = Axial length of the straight region

$L_{R=}$ Axial length of the nose

5.Reference

1. Richard F Hubble, "The near feild flow and Drag on Cylindrical Bodies Moving Concentrically Inside Very Long Tubes' Naval Underwater System Center Technical Document 7036, London Aug 1991
2. Hope R.G and Gouse S.W, 'Fluid dynamic Drag on vehicles Travelling Through Tubes' Carnegie-Mellon University Report No 1-59076-1, Aug 1969 NTIS No PB 188 451.
3. NAYAK, U.S.L, The Aerodynamic drag of Tube Vechicles Travelling at Low Subsonic Speeds, BHRA Fluid Engineering Second International Symposium on Aerodynamics and Ventilation of Vehicle Tunnels, Paper E1, Cambridge, England, March 1976
4. Fluent Inc, Fluent User Manual 2006.