.....

NUMERICAL ANALYSIS OF WATER JET INLET FLOW

LILADHAR GANESH DHOBLE¹, Dr. V. ANANTHA SUBRAMANIAN²

¹Dept. of Ocean Engineering, IIT-Madras, Chennai, India

²Dept. of Ocean Engineering, IIT-Madras, Chennai, India

ABSTRACT

The purpose of present study is to investigate three dimensional viscous flow field through water jet inlet duct. Water jet systems are frequently used in high speed marine vehicles because they have high propulsive efficiency, good manoeuvrability and low noise. The flow to the impeller has to be uniform for wide range of operating speed to get good performance. There are certain parameters which influence the duct flow such as duct angle, shaft rotation etc. The flow in the water jet inlet is analyzed in detail for a shallow angle duct. The flow pattern and the presence of shaft influence can be visualized in CFD studies. Commercial RANSE based CFD software package Star-CCM+ was employed to study the flow behavior in the inlet duct. Standard k-epsilon turbulence model with standard wall functions was used. Self-propulsion tests were done for various ship speeds and mass flow rate through the exhaust nozzle was measured. The duct flow was studied for the same mass flow rates. The complicated flow features in intake duct with and without shaft were well understood in the present study.

Keywords: Water Jet Inlet Duct, CFD, Star-CCM⁺, Inlet Velocity Ratio, Stagnation Point.

1. INTRODUCTION

Water jet propulsion system works on Newton's third law of motion. It uses reaction force of high speed jet to move marine vessels. They have high propulsive efficiency at high speed, less prone to cavitation and good maneuverability [5]. Therefore water jet system are used for high speed marine vehicles, patrol boats etc. Water jet system consists of inlet duct, impeller, stator and a nozzle. The design of water jet system is more complex by virtue of its components. Typically it has a flush type inlet which operates under 3-D complex flow conditions. Flow in inlet duct is more complex since it ingests hull boundary layer; duct angle and S-bend of inlet which influence flow distortion and separation and shaft rotation. Duct angle should be chosen in such way that the water jet system should take minimum required space and flow at impeller plane should be uniform. Therefore inlet duct angle is a compromise between uniform flow at the impeller plane and space required for system installation.

Efficiency of complete water jet system depends on the performance of its inlet duct. The inlet should not produce extra drag. The design of inlet should be based on hydrodynamic design principles [1]. Lot of experimental studies had been carried out in last two decades. The replacement of ramp inlet by flush type inlet [2] reduces inlet momentum drag. To reduce this drag, a wide, large width-to-height ratio inlet was used which spans as much as the beam at the transom. The definition of 'height' in this context is the height of the outlet nozzle centre above the inlet aperture. Propulsion coefficient is improved by 10-20 % as compared to the performance with conventional normal width inlets. This kind of inlet is well suitable for low design jet velocity ratios (JVR). The complexity of flow in water jet inlet is more influenced by hull boundary layer ingestion [4] and [7]. They have studied the effect of natural and artificially thickened boundary layer on pressures along the duct for shallow angled inlet and developed basic tool for CFD calculation. It was concluded that an accurate prediction of propulsive thrust can be made with accurate calculation of hull boundary layer.

Recent development in CFD codes and computational facilities has enabled modeling of complex flows like water jet system. However most of the work is done for large duct angle. Large duct angle saves space required for installation but experiences more flow distortion. Comprehensive experimental procedures regarding water jet-hull interaction were given by [3]. Computation of pressures along ramp and cutwater side was calculated. Large deviation in results was observed. This was due to negligence of viscous effects in the calculation. As the inlet velocity ratio increases (IVR), the stagnation point on cutwater side moves inward [6]. Inlet velocity ratio defined as ratio of ship speed to the speed at the impeller plane. A large duct angle was considered for study and methodology to calculate water jet thrust is explained. The competency of numerical scheme was checked by numerical validation of experimental results by [8]. Using concept of dividing streamline [9] presented work on the calculation of flow losses in the inlet duct.

The development and increasing use of CFD as mentioned above has resulted in use of three dimensional viscous flow methods for flow analysis of inlet duct. In the present study a shallow angle water jet inlet duct is considered. Experimentally obtained mass flow rate has been applied as boundary condition at nozzle outlet. The flow analysis is performed using commercial CFD code STAR-CCM+. Pressures at ramp side and cutwater side were measured. Further presence of shaft on total pressure inside the duct is shown. The flow analysis paves way for further improvements and development of water jet system performance.

2. GEOMETRY AND GRID GENERATION

Water jet propulsion systems mainly consists of intake duct, a pump and exhaust convergent nozzle. The present study analyses flow dynamics inside water jet inlet. A schematic diagram of inlet duct is shown in Fig. 1.

Figure 1. Inlet geometry with main parts

In the present study, inlet duct in addition to convergent nozzle is studied. The perspective view of inlet duct upto impeller plane is shown in Fig. 2 with its isolated surfaces.

Figure 2. Perspective view of the Inlet and isolated surfaces

A rectangular flush opening is considered here. As seen from Fig. 2, the rectangular box shape is transformed into circular shape at the impeller plane. This is the challenge in the modeling of inlet duct. The inlet duct angle is 22.5° and the nozzle diameter is taken as 675 mm. The shallow angle of duct reduces flow losses. The water jet inlet design depends on the 'hull rocker angle' (the rise of the keel line at the stern near the inlet). The inlet must be turned to that angle to match the flow conditions. In the present study for the candidate hull form, the hull rocker angle is nearly flat or zero. The

inlet duct design has been carried out with considerations of the local flow conditions. The data on model scale is given in Table 1.

Table 1. Design point particulars for inlet geometry (Model scale)

Design speed ($V_{s\ design}$)	2.88	knots
Inlet Length(Lin)	560	mm
Inlet Diameter (D_{pump})	140	mm
Centre Depth	96	mm
Lip Length	$X_{lip}=103$	mm
Lip Bottom from axis (0,0)	$Y_{lip}=96$	mm
Top Duct Radius	387	mm
Axis Duct Radius	317	mm
Lip Duct Radius	247	mm
Lip Centerline Radius	6	mm
Entrance Ramp Radius	1368	mm
Side Edge Base Radius	6	mm
Duct Angle (α_D)	25	degree
Rocker Angle (α_R)	0	degree
Nozzle Angle (α_N)	0	degree
Nozzle Diameter	70	mm
Scale of model	1:10	

Star-ccm+ provides auto-meshing of the computational domain using block structured hexahedral grid and trimmed hexahedral cells were used. Trimmed hexahedral cells are standard hexahedral cell shapes cut to form polyhedral cells conforming to the model geometry. This allows them to form body fitted meshes easily and quickly without having any implicit knowledge of the geometry shape

3. GOVERNING EQUATIONS AND NUMERICAL METHOD

In the present study, commercial CFD code STAR-CCM+ is used for all numerical simulations. The code uses SIMPLE algorithm to couple pressure and velocity. The unsteady, three dimensional and viscous flow is solved using finite volume method (FVM). K-epsilon turbulence model with standard wall functions (high wall y^+) was used for all simulations performed. The three dimensional incompressible Navier-Stokes equation is given as:

$$\nabla \cdot (\rho \vec{V}) = 0 \quad (1)$$

$$\rho \frac{\partial \vec{V}}{\partial t} + \rho (\vec{V} \cdot \nabla) \vec{V} = -\nabla p^* + \nabla \cdot \mu_{eff} \left(\nabla (\vec{V}) + (\nabla (\vec{V}))^T \right) \quad (2)$$

where 'p' is pressure, $\mu_{eff} = \mu + \mu_t$, the effective turbulent viscosity, μ is molecular viscosity and μ_t is turbulent viscosity.

4. BOUNDARY CONDITIONS

At the inlet of the computational domain a velocity distribution according to flat plate boundary layers distribution is given. The present calculations are based on power-law value $n = 7$ and a boundary layer thickness of 50mm (0.37D), where D is the diameter of the inlet. An undisturbed uniform velocity is prescribed in remaining cells.

At the outlet of the domain, constant pressure is applied. The walls of the domain are kept far apart from the inlet, and slip condition is applied. At the bottom slip condition is given. Finally the nozzle of the inlet is given outlet boundary condition where mass flow rate is specified, which is flow required for the pump. The static pressure distribution at the nozzle exit plane becomes part of solution.

5. RESULTS

The numerical analysis of water jet inlet was performed for with and without shaft condition in the inlet. Pressures at ramp and cutwater side, velocity at impeller plane, pressures in the duct and impeller plane are compared for two cases.

5.1 Static pressure along the ramp centre line and cutwater (without shaft)

The static pressure is measured at 31 locations along the ramp centre line and at 42 locations along the cutwater. The reference origin for the locations is placed at body fitted coordinate system. The points on the cutwater side starts from the nozzle pass along the cutwater to some distance along the top (hull) downstream of the inlet. The static pressure is made non-dimensional using the density ρ and ship speed V_{ship} as:

$$C_p = \frac{p - p_{ref}}{\frac{1}{2} \rho V_{ship}^2} \quad (3)$$

The reference static pressure p_{ref} is taken as atmospheric pressure of 101325 Pa and ship speed is taken at the inlet of the domain. The static pressure is measured for different IVR (ship speed/mean flow velocity at the impeller face). Table 2 below shows the conditions used for measurements.

Table 2 Parameters of measured quantities for pump diameter=135 mm

V_{ship} (m/s)	V_{pump} (m/s)	IVR	Re_{inlet}
0.50	0.48	1.038	7.31E+04
0.55	0.49	1.118	7.47E+04
0.60	0.56	1.066	8.55E+04
0.65	0.57	1.137	8.68E+04
0.70	0.65	1.078	9.85E+04
0.75	0.71	1.055	1.08E+05
0.80	0.70	1.141	1.06E+05
0.85	0.75	1.129	1.14E+05

The Reynolds number is calculated based on the pump velocity and inlet duct diameter. Fig. 3 shows the values of non-dimensional static pressure along the ramp center line, based on the CFD studies. Up to the plane in front of the impeller, the pressure is almost constant and then suddenly rises. This is due to the bottleneck effect of the converged nozzle. From Fig. 3 it is obvious that as IVR increases the pressure rise becomes less severe.

Figure 3. C_p distribution along the ramp for various IVR

The influence is highest at low IVR. Fig. 4 shows the pressure distribution along the symmetry plane of the configuration for IVR of 1.038(left) and 1.141(right). It is clear that the pressure distribution is constant along the ramp up to the impeller plane. There is reduced pressure at the ramp tangency point from low IVR to high IVR. This is due to rapid acceleration of flow at this point.

From the pressure distribution it is clear that the location of stagnation point (Fig. 4) changes along the lip with IVR. Therefore the minimum pressure point also moves. The graphs for cutwater pressure are divided into two groups of alternate IVR for clarity (see Fig. 5). The stagnation point moves from the outward cutwater region to inward as IVR increases. The movement of stagnation point and the minimum pressure point is related to the shape change of inlet stream-tube. Fig. 6 shows the pressure distribution along the inlet centerline. The pressure at the impeller plane is almost constant for various IVR.

Figure 4. Pressure coefficient at the symmetry plane for IVR of 1.038(left) and 1.141(right)

Figure 5. Cp along the lip

Figure 6. Cp along the duct centreline

5.2 Comparison of Total Pressure at Impeller plane

The impeller plane is the cross sectional area just upstream of the impeller at the end of the inlet duct. The total pressure in non-dimensional form is given as:

$$C_{p_{total}} = \frac{P_{tot} - P_{ref}}{\frac{1}{2} \rho V_{ship}^2} \quad (4)$$

From Fig. 7a to 7c, it is concluded qualitatively that, as the IVR increases the total pressure losses becomes severe. This indicates that the hydraulic losses in the duct increases with IVR.

Figure 7(a). Total pressure coefficient distribution for IVR=1.038

Figure 7(b). Total pressure coefficient distribution for IVR=1.078

Figure 7(c). Total pressure coefficient distribution for IVR=1.141

The total pressure coefficients are compared qualitatively for with and without the presence of shaft in the inlet duct. There is pressure drop on the upper side of the shaft, due to the shaft/ stern tube being an obstruction to the incoming flow. The consequence of this is increased hydraulic losses. These losses can be significant as seen in the pressure contours. In the practical arrangement the shaft is led through a stern tube inside the impeller.

Fig. 8 shows the flow features at the plane joining the hull bottom and the inlet. The stagnation line is shown clearly to indicate dividing path lines location.

Figure 8. Path Line Pattern at Cross Section of Inlet Plane (IVR=1.141)

6. CONCLUSIONS

The numerical flow analysis was performed for a shallow duct angle inlet to provide understanding of complex flow dynamics occurring during water jet operation. The study also checks applicability of k-epsilon turbulence model using commercial code STAR-CCM+. The present calculations showed that suction is created at the ramp tangency point and this effect increases as IVR increases. The location of stagnation point plays an important role in terms of amount water sucked in the inlet. This decides the dividing streamline. It was seen that as the IVR increases the location of stagnation point along the lip moves inward. The pressure losses in the duct due presence of shaft becomes sever as IVR increases. The analysis brought out important parametric study of design variables and paves the way to optimize the system components for improved system performance.

REFERENCES

- [1] Wislicineus, G.F., "Hydrodynamic Design Principles of Pumps and Ducting for Water jet Propulsion", Naval Ship Research and Development Center, Bethesda, Maryland, 1973.
- [2] Purnell, J.G., "The Performance Gains of Using Wide, Flush Boundary Layer Inlets on Water-jet Propelled Craft", David W. Taylor Naval Ship Research and Development Center, Bethesda, Md. 20084, 1976.
- [3] Terwisga, T.J.C.V., "Water-jet-hull interaction", PhD thesis, Delft University, Netherland, 1996.
- [4] Roberts, J.L. and Walker, G.J., "Experimental Determination of Ingestion Streamtubes for Water jet Propulsion Intakes", 13th Australasian Fluid Mechanics Conference, Monash University, Melbourne, Australia, 1998.
- [5] Park, W.G., Jang, J.H., Chun, H.H. and Kim, M. C., "Numerical flow and performance analysis of water-jet propulsion system", Ocean Engineering, Vol.32 (2005), pp. 1740-1761, 2005.
- [6] Bulten, N.W.H., "Numerical Analysis of a Water jet Propulsion System", Ph.D thesis, Technical University of Eindhoven, ISBN-10: 90-386-2988-5. Library Eindhoven University of Technology, 2006.
- [7] Brandner, P.A. and Walker, G.J., "An Experimental Investigation Into the Performance of a Flush Water-jet Inlet", Journal of Ship Research, Vol. 51, No. 1, pp. 1-21, 2007.
- [8] Guo, S. and Chen, Y., "Numerical Analysis of Water-jet Inlet Flow", Proceedings of the 2009 IEEE International Conference on Mechatronics and Automation, August 9-12, Changchun, China, 2009.
- [9] Ming, D.J. and Sheng, W.Y., "Research on Flow Loss of Inlet Duct of Marine Water jets", J. Shanghai Jiaotong Univ. (Sci.), 15(2): 158-162, 2010.
- [10] Manish Dadhich, Dharmendra Hariyani and Tarun Singh, "Flow Simulation (CFD) & Fatigue Analysis (Fea) Of A Centrifugal Pump" International Journal of Mechanical Engineering & Technology (IJMET), Volume 3, Issue 3, 2012, pp. 67 - 83, ISSN Print: 0976 - 6340, ISSN Online: 0976 - 6359.