International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 4 Issue 11, November-2015

Fluid Flow Optimization Procedure for a Pump intake using Coanda Effect: A Numerical Approach

Karanbir S. Sidhu
Mechanical Engineering Dept.
Punjabi University
Patiala, India

Ankit Goyal
Mechanical Engineering Dept.
Punjabi University
Patiala, India

***Abstract*—The geometry of a pump intake structure is responsible for providing an efficient water supply to a series of pumps; failure of which causes undesirable-turbulent flow patterns, flow separation; which further leads to generation of unacceptably high magnitudes of swirl, undesirable cavitation of the impeller in conjunction with excessive mechanical vibrations in the pumps.**

The flow characteristics in pump intakes having multiple pump installations is involved, which consequentially demands for an empirical, as well as, a numerical methodology for the design of its intake geometry. The numerical study carried out in this paper aims at optimizing the overall fluid flow in a pump intake by the use of a commercially available CFD code. The test cases pertain to an intake with, 4 identical pumps working at duty point at the lowest water level (LWL), with a severely non-uniform and turbulent flow resulting due to the inherent constraints imposed by the intake geometry, arising due to site specific conditions. A total of three test cases are discussed, two of them have blocking arrangements aimed to optimize the fluid flow behavior. The flow is successfully optimized and a control over the flow is introduced by the unique application of the well-established fluid phenomena- "wall attachment" or the "Coanda effect" in the design of a curved surface blocking arrangement used in the ultimate case.

Keywords—Sandgrain Roughness; Surface Roughness; Coanda Effect; Wall Attachment; Velocity Vectors; Swirl; Swirl-angle; Single-phase; Curved Surface Blocking

INTRODUCTION

The purpose of a pump intake is to provide a controlled and adequate supply of water to the large-scale pumps installed in the pumping station; additionally, it must ensure that the fluid flow provided, must be swirl free for improved pump efficiency. Due to the involute nature of the fluid-structure interactions that take place inside the pump intake domain, the geometry of the pump intake which includes the approach channel, the fore bay and the pump-chambers enclosing the bell-suction of the pumps is critical for a steady, proportionate and swirl free fluid delivery to the pumps.

The foremost problems associated with the design of pump intakes comprise of- a turbulent, asymmetrical flow, high approach velocity of the fluid into the pump chambers, impeller cavitation, and unwarranted vibrations of the pump in addition with diminished pump efficiency and performance. The asymmetrical flow may lead to excessive wear of the pump impeller, strenuous load on the bearings, greater swirl magnitudes, deposition and ingestion of dispersed sediments which ultimately bring about enlarged maintenance costs and declined operational life of the pump.

The complications associated with poor fluid flow are well known and understood; however, the methods to be implemented for the rectification of these issues are not concrete- the geometry of every pump intake is unique due to its site specific considerations and its intended purpose. The recommendations [2] from the Hydraulic Institute Standards (HIS) provide us with a collective, but provincial approach to designing a pump intake. In the absence of a well formulated protocol for the design of pump intakes, it thus necessitates empirical analysis of a scaled, geometrically similar (distorted model in some cases) physical model of the pump intake, and the application of Froude similarity rules to warrant a robust end design of the intake geometry.

However, the experimental approach leads to an exponential increase in the costs when, one has to optimize the flow characteristics, which invariably demands dilatory geometric modifications, laboratory testing of the modified geometry under several duty points, test equipment calibration and setup procedures- all of which need to be monitored judiciously. With prodigious advancements in Computational Fluid Dynamics (CFD), and an enormous decrease in the computational costs associated with it, numerical simulation techniques are successfully supplanting experimental methods for investigating fluid flow characteristics of pump intakes.

This paper describes a numerical approach for optimizing the fluid flow characteristics of a pump intake which has a non-uniform and disproportionate flow. To establish efficient and active control of the fluid flow, and to rectify this asymmetric flow pattern, a blocking has been successfully designed around the principle of wall attachment of fluids or the Coanda effect. For accurate simulation of the fluid flow when implementing the wall attachment phenomenon, it is particularly imperative for an appropriate near-wall treatment model to be described. A sand-grain roughness value is defined for all the walls (excluding the suction and the column pipe of the Pumps) used in this paper to ensure dependable results.

In this work, the flow is compared amongst three dissimilar intake geometries. In Test Case A, the basic geometry which encompasses just the fluid flow domain without any flow controlling strategies to govern the fluid flow behavior. In Test Case B, the standard approach uses the recommended procedure of employing guide walls for flow optimizations. Ultimately, Test Case C demonstrates the use of what we refer to as, "the curved surface/blocking approach". These three test cases are equated using velocity vector plots, fluid flow streamlines, velocity magnitudes achieved at various sections of the intake.

I. MODELING OF THE CFD PROCEDURE

As this study aims at optimizing the overall fluid flow characteristics in the pump intake, and is not concerned with detection of air-entrainment or free surface vortices; a single-phase model was adopted for each of the CFD simulations. A single-phase fluid flow model was selected, rather than a multi-phase-VOF (Volume of Fraction) model, because it is computationally much less intensive and is evidently most suitable for simulating and comparing a number of iterations of the geometric design variations.

A. Geometry of the Fluid Domain

The CFD analysis was performed on a scaled and geometrically similar model of the prototype intake. The ratio of, the geometric scale of the model, to that of the prototype was analytically calculated to be 1:6 using Kinematic and Dynamic modeling principles of Fluid dynamics. Also, as recommended by the Hydraulic Institute Standards, the model flow was simulated at 1.5 Froude Number (Fr).

The amount of water that flows through the pump chambers in the model intake is 774.9gpm, with the prototype intake flow rate of 45772.2gpm. The diameters of the bell-suction and the column pipe in the model intake are 0.18m and 0.192m. The submergence of the bell-suction at the LWL was kept at 0.5m, with a bottom clearance of 0.2m. The distance between the center of the suction pipe and the back wall of the chamber is 0.125m.

Our primary objective is to study the flow pattern from the approach channel till it enters the pump chambers. The deployment of curtain wall(s) and other chamber geometry modification strategies are not gainful considering the scope of the present study.

TABLE I MODEL DIMENSIONALESS NUMBER VALUES.

Model parameters			
Scale	***Froude No.***	***Reynolds No.***	***Weber No.***
1 : 6	1.5	103720	992.074

Table I depicts the values of the governing dimensionless numbers for the model pump intake. It is to be noted that the values achieved conform to the values recommended by the Hydraulic Institute Standards, thus, the scaled geometry of the model intake is suitable for replicating the actual flow conditions of the prototype intake.

B. Mesh Generation

The current study deals with the optimization of the fluid flow, hence, naturally it requires iterations of design variations to be tested and compared. Therefore, it is important for the mesh generation process to be computationally less time intensive, while simultaneously providing good quality volume mesh having a higher seed density near the walls, vicinity of the bell-suction and the column pipes of the pumps- as the variations in velocity gradients are highest in these regions.

A hybrid unstructured mesh was generated using first, the Octree algorithm to generate a preliminary mesh, which was followed by refinements and smoothing of the generated mesh. Finally, the Delaunay algorithm was used to generate the refined volume mesh for each of the test case intake geometries. The volume mesh thus generated incorporates- Hexahedral elements in the bulk portion of the fluid domain. Whereas, the regions near the wall and the suction pipe are meshed using smaller tetrahedral and prism elements. This approach ensures a much higher seed density near the regions of interest. Approximately, the total numbers of elements vary from 13.58 to 15.1 million, whereas the numbers of nodes vary from 9.24 to 10.12 million for the three test cases reported.

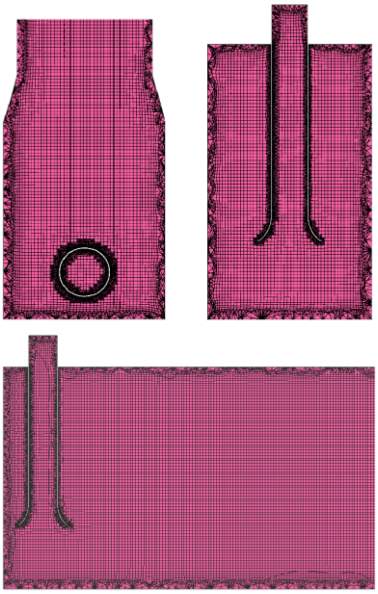

Fig. 1. Sectional views of the volume mesh prepared.

C. Pre-processing

The physics of the simulation domain is the most important aspect of a CFD simulation. It requires precise and accurate setup of the boundary conditions, definition of the wall treatment, selection of an apt turbulence-model and setting up the solver control parameters. The κ-ε turbulence model with scalable wall functions was selected as it has been successfully used on numerous occasions for pump intake simulations. The domain was initialized as Non-Buoyant, Stationary Fluid domain with water as the working fluid at 398K with the reference pressure set to 1atm pressure.

As the test case simulations involve a single-phase fluid domain with steady-state and incompressible state of the fluid flow, the domain surface corresponding to the free-surface of water was setup as a symmetric boundary condition. The inlet boundary condition was specified as a mass-flow inlet with 48.97kg/s of water flowing normal to the domain inlet surface. The outlets of the four pumps were setup as openings with specified velocity vector magnitude of 0.695m/s in the Z-axis direction. The surfaces corresponding to the bell-suction and the column pipe were setup as smooth-walls with no slip, whereas the remaining intake surfaces were setup as rough walls. This is very important as the wall attachment phenomenon is very susceptible to the smoothness of the walls, so a standard smooth-wall approach will not yield dependable results due to excessive wall attachment.

The roughness of the walls is specified by assigning a sand-grain roughness (ε_o) value in millimeters. The analytical approach adopted to calculate the value of the sand-grain roughness, requires the surface roughness values in R_{ZD} of the intake wall surfaces intercepting the fluid flow in the final prototype construction. The mathematical relationship between, the surface-roughness (ε_o) and R_{ZD} is given by (1):

$$\varepsilon_o = 0.978\, R_{ZD} \qquad (1)$$

Considering, that R_{ZD} value for a rough concrete construction varies from 0.3millimeter to 0.75millimeter, a value of 0.55millimeters was elected as the value for the sand-grain roughness of the walls. The discretization scheme is setup as high resolution; this enables the output results to be accurate and dependable till the second order. The solver was run with the RMS value of the residual convergence target specified as 10^{-5}. The solver was run in distributed parallel using Intel-mpi for more than 30hours for each of the test cases.

II. TEST CASES

A. Test Case A: The Elementary Intake Geometry

The geometry for the first test case does not include any optimization strategies. As such, it has been presented to discuss the fluid flow problem associated with the intake design geometry. Fig. 2 shows the overall geometry and the necessary dimensions associated with the original intake geometry.

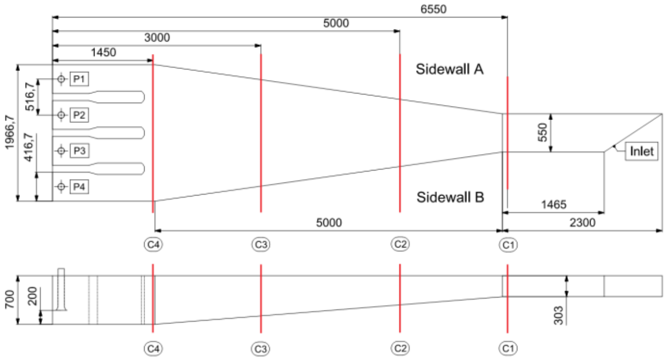

Fig. 2. Original intake Geometry (scale 1:6)

The markers from C1-C4 signify the sections used for sampling the velocity data of the fluid flow. The distance of each of these sections is clearly defined in Fig. 2 from the back-wall of the pump chambers. The velocity profile at section C4 is by far the most important aspect of the study; the flow entering the pump chamber from the forebay has to be of low velocity, minimum swirl and uniformly distributed throughout to ensure adequate and robust supply of water to the pumps.

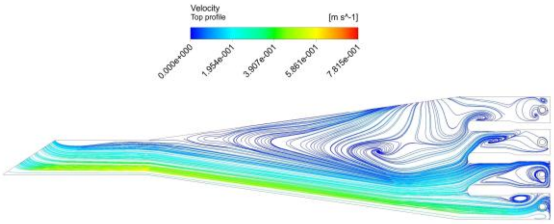

Fig. 3. Top view of the flow velocity streamlines.

The fluid flow streamlines Fig. 3 for the basic pump intake delineates the asymmetrical flow produced throughout the entire pump intake. This is as expected due to the inlet of the intake placed at very odd angle. The large swirl produced is due to the heightened difference between the velocity magnitudes of the fluid flow near the sidewalls A and B. The velocity of the flow near the sidewall A being at a much larger magnitude throughout causes the flow to circulate as it moves towards sidewall B.

The velocity profiles for the basic intake geometry Fig. 5 shows a very strong asymmetrical flow profile throughout the entire domain of the intake. Such a fluid flow trend makes the intake susceptible to high deposition of dispersed sediments towards one of the problematic regions, which immensely escalates the maintenance costs, wear and tear of the pump impeller due to excessive ingestion of the sediments.

At section C4, the flow characteristics at various depths are depicted in Fig. 6. The flow entering the pump chambers is disproportionate and has a high velocity magnitude towards the sidewall A which peaks at about 0.36m/s. This non-uniform flow augments the swirl strength of the water, which further results in greater swirl angles produced in the column pipe of the pump, reduction in efficiency, performance and operational life of the pump.

The velocity vector plot for section C4 shown in Fig. 4, best depicts the fluid circulation taking place just before the flow enters the pump chambers. The side profile of the vector plot reveals that the fluid near the bottom of the flow domain is circulated backwards with a high velocity rather than entering the pump chambers. Due to these high velocity magnitudes near the free surface and the bottom of the water having opposite flow directions, results in the generation of the swirl at the end of the forebay channel as shown in Fig. 7 depicting the vertical flow streamlines.

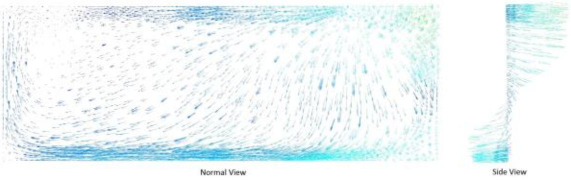

Fig. 4. Velocity vector plots at Section C4 (Test Case A)

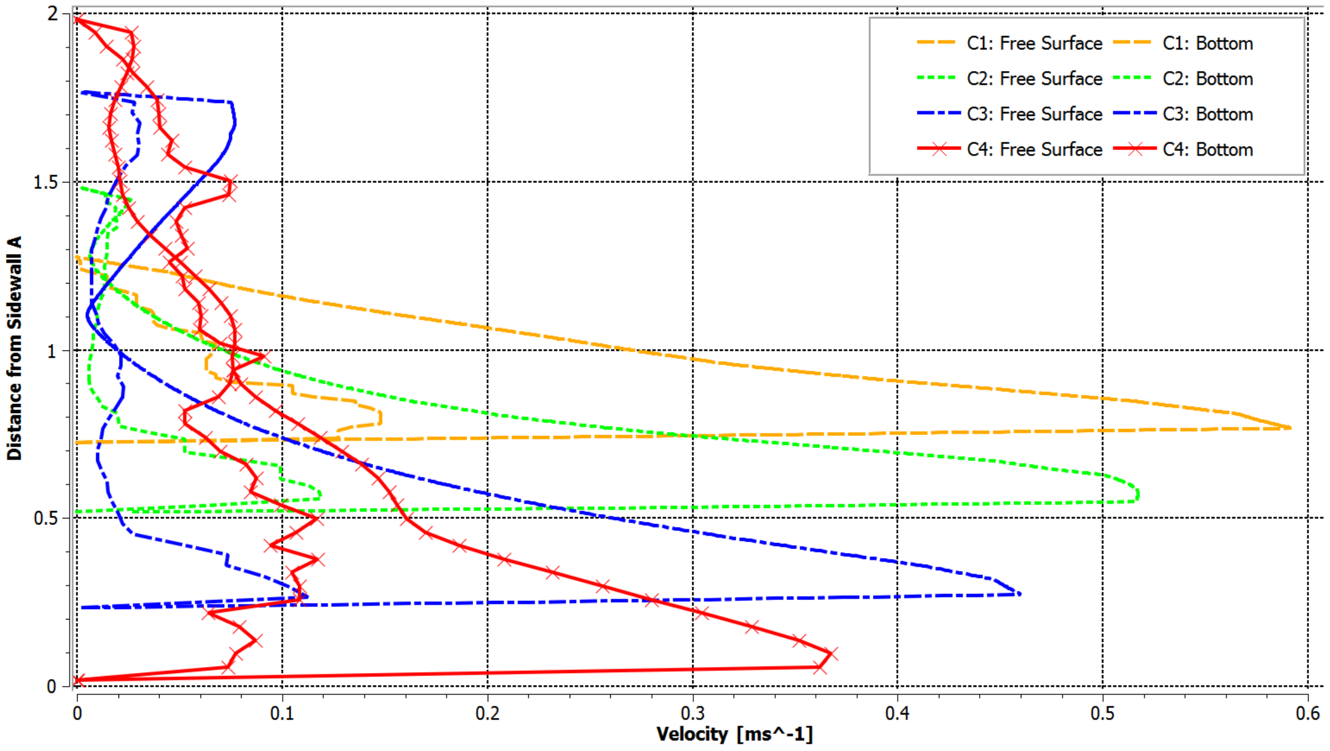

Fig. 5. Overall Velocity profiles (Test Case A)

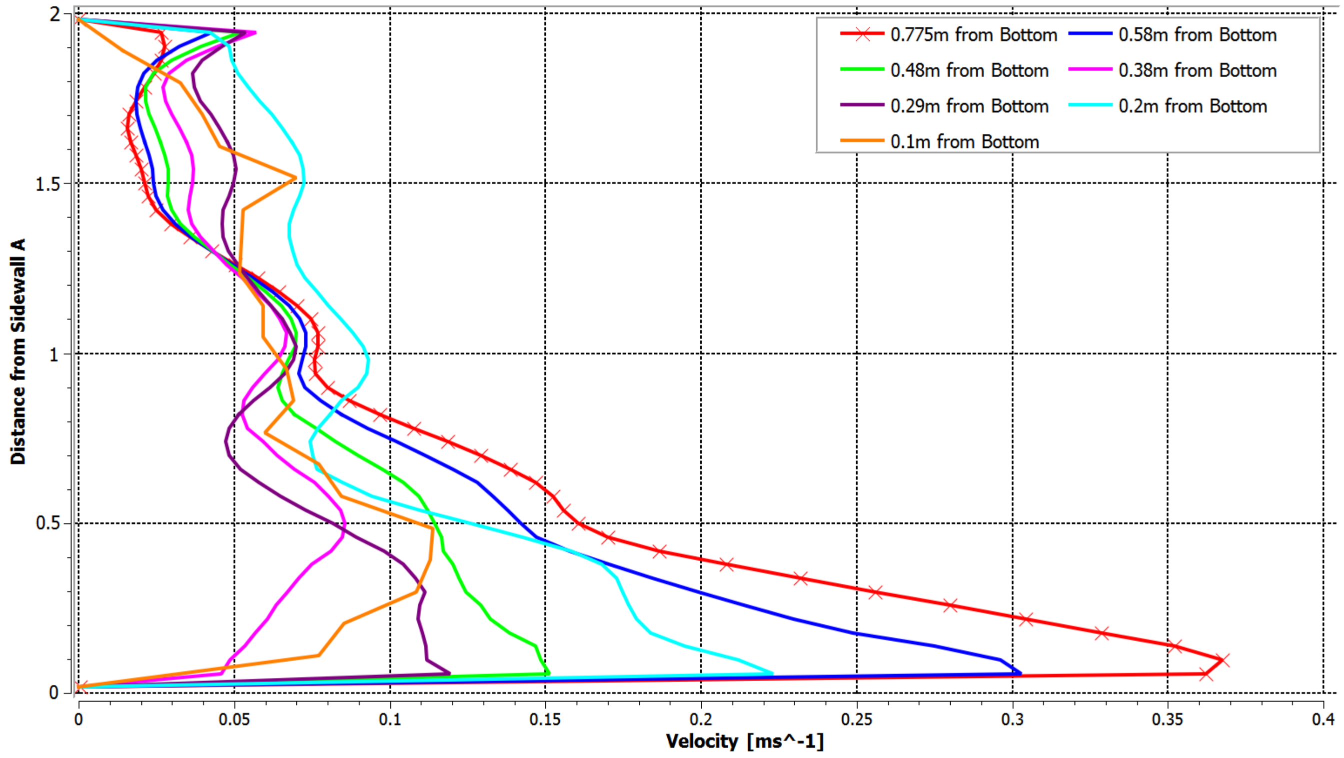

Fig. 6. Velocity profiles at Section C4.

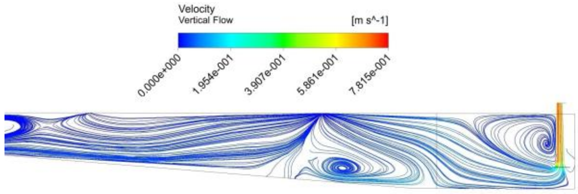

Fig. 7. Fluid flow velocity streamlines along a vertical longitudinal plane (Test Case A)

The vertical velocity streamlines at a plane passing through the center of Pump (P2) further elucidate that due to the high flow velocity entering the pump chamber, flow separation takes place before it reaches the pump chambers as the fluid sticks to the bottom walls, and the flow from above, having higher velocity, starts to flow downwards, some of which flows directly towards the bell-suction. However, due to the higher overall velocity, the flow starts to recirculate in an upward trend when coming from the back-wall instead. The swirl engendered thus, is formed exactly on top of the bell-suction of the pump, which escalates the magnitude of swirl generation in the pump chamber and the column pipe of the pump.

B. *Test Case B: The Standard Blocking Approach*

A number of methods can be adopted to mitigate the asymmetrical-turbulent flow pattern of the basic intake geometry. Various methods as recommended by the Hydraulic Institute Standards were simulated and the results compared. However, because of the adverse angle that the inlet surface makes with the longitudinal plane of the intake geometry, such methods fail to sufficiently control and uniformly streamline the fluid flow.

The results of the most efficacious of the many standard geometry variations tested has been reported in Test Case B. The geometry prepared includes the use of 4 guide walls as shown in Fig. 8.

The very first guide wall that intercepts the water flowing into the approach channel from the inlet is placed at an angle to direct the bulk-flow of the water towards Sidewall B. The water is further directed by the guide walls placed at the entry of the fluid into the fore bay. These guide walls have an elongated length to control the flow for a longer period of time, and thus give the fluid time to attain a more uniform and steady flow pattern.

In Fig. 9, the fluid flow streamlines of the modified geometry indicate and improvement in the flow characteristics as compared to the basic geometry of the intake. The flow is reasonably uniform from the approach channel up until just before it reaches the mid-forebay; the fluid flow, separates from the Sidewall B as the directional momentum of the water flowing in from the inlet towards Sidewall A is not sufficiently appeased to produce a uniform fluid flow.

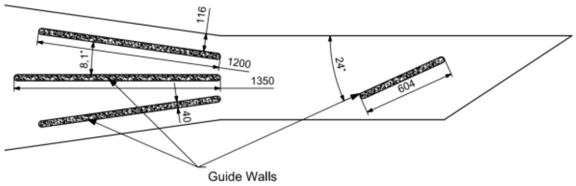

Fig. 8. Geometric Modification for Test Case B

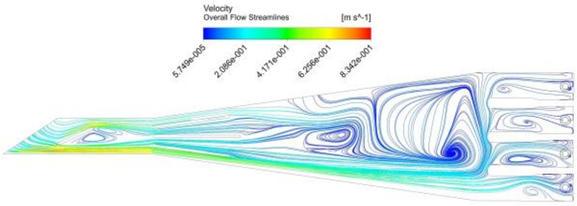

Fig. 9. Top view of flow velocity streamlines (Test Case B)

Fig. 12 depicts the overall velocity profiles produced at the sections C1-C4, gives a clear indication of the improved, yet, asymmetrical flow characteristics. The velocity profiles generated at sections C1 and C2 substantiate the contention of the flow being partially uniform till it reaches the region-midway of the fore bay. The velocity profiles at Sections C3 and C4 indicate the flow to be non-uniformly distributed, with the peak velocities around 0.35m and 0.20m from the Sidewall A.

The flow velocity profiles at section C4 shown in Fig. 13 noticeably indicate that the flow entering the pump chambers is non-uniform and possess higher velocity magnitude near sidewall A. The highest velocity magnitude peaks to value of about 0.25m/s; comparatively, the average velocity magnitudes near sidewall B are very low, this causes a strong recirculation of water from sidewall A towards sidewall B.

In Fig. 10 the Normal View of the velocity vector plot generated at section C4 shows a moderate reduction in swirl; this is due to the decreased velocity magnitudes entering the pump chamber, which in-turn reduces the magnitude of flow velocity between the two sidewalls. The Side View of the velocity vector plot shows a marked decrease in the overall velocity magnitudes for the flow near the free surface, in addition to the bottom recirculation.

The vertical velocity streamlines are very similar to the ones presented in Fig. 11. The only noticeable improvement is the variation in the placement of the second swirl which occurs due to recirculation of the flow from the back-wall of the pump chambers. Comparatively, this now originates far away from the bell-suction- reducing the magnitude of swirl in the vicinity of the bell-suction of the pump.

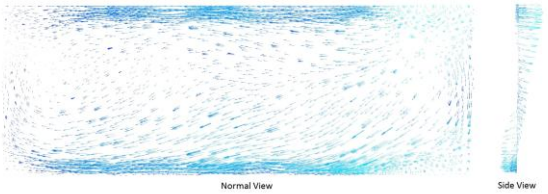

Fig. 10. Velocity vector plots at Section C4 (Test Case B)

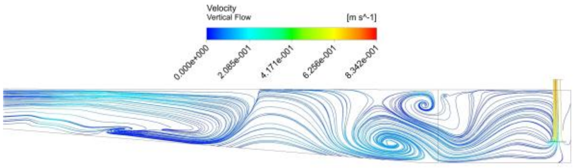

Fig. 11. Fluid flow velocity streamlines along a vertical longitudinal plane (Test Case B)

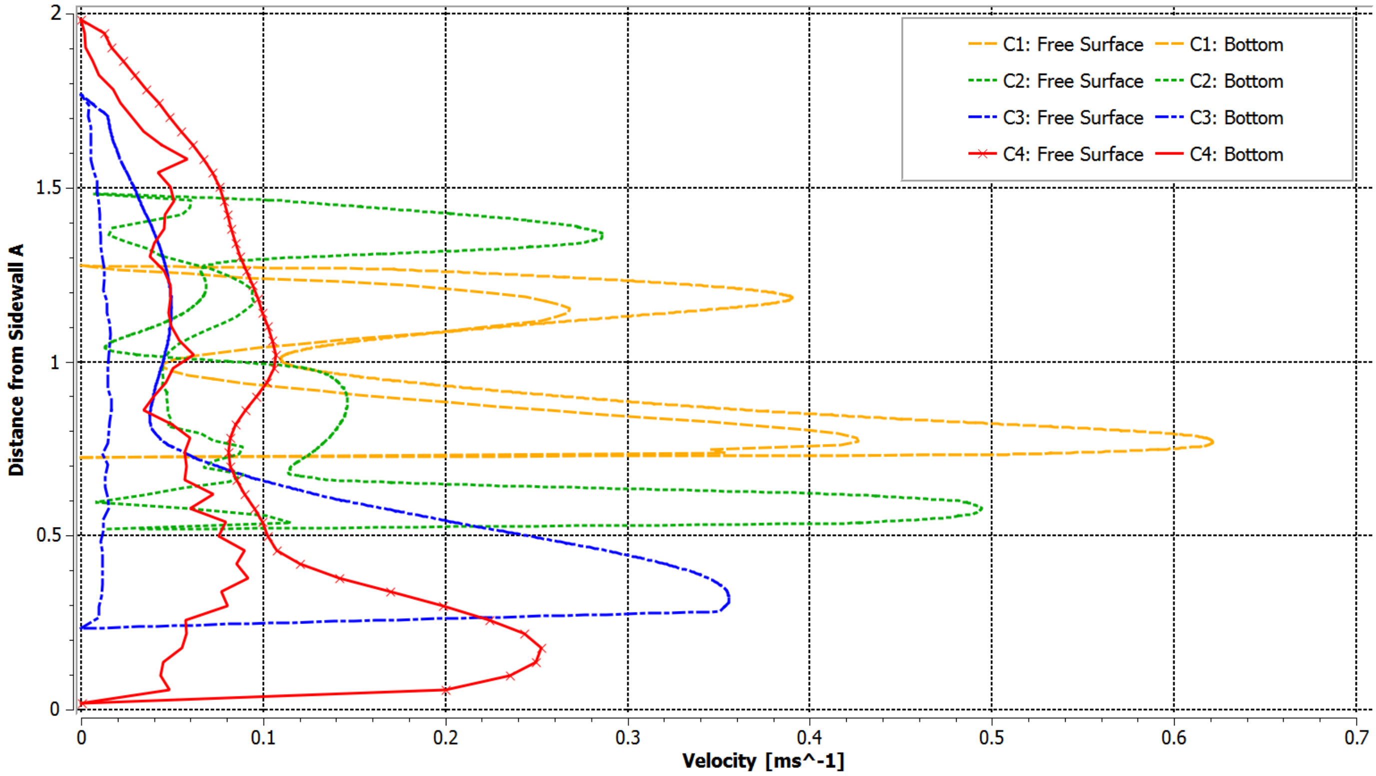

Fig. 12. Overall velocity profiles (Test Case B)

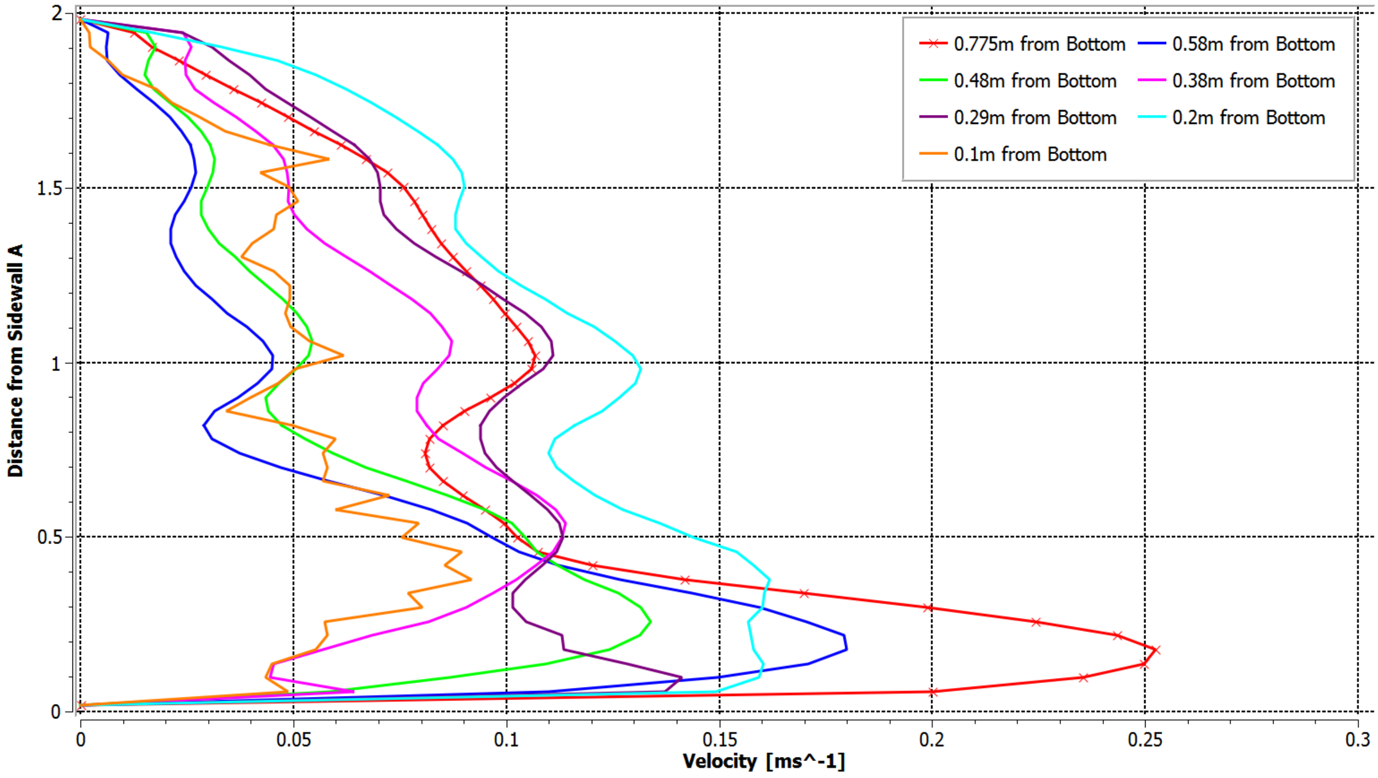

Fig. 13. Velocity profiles at Section C4 (Test Case B)

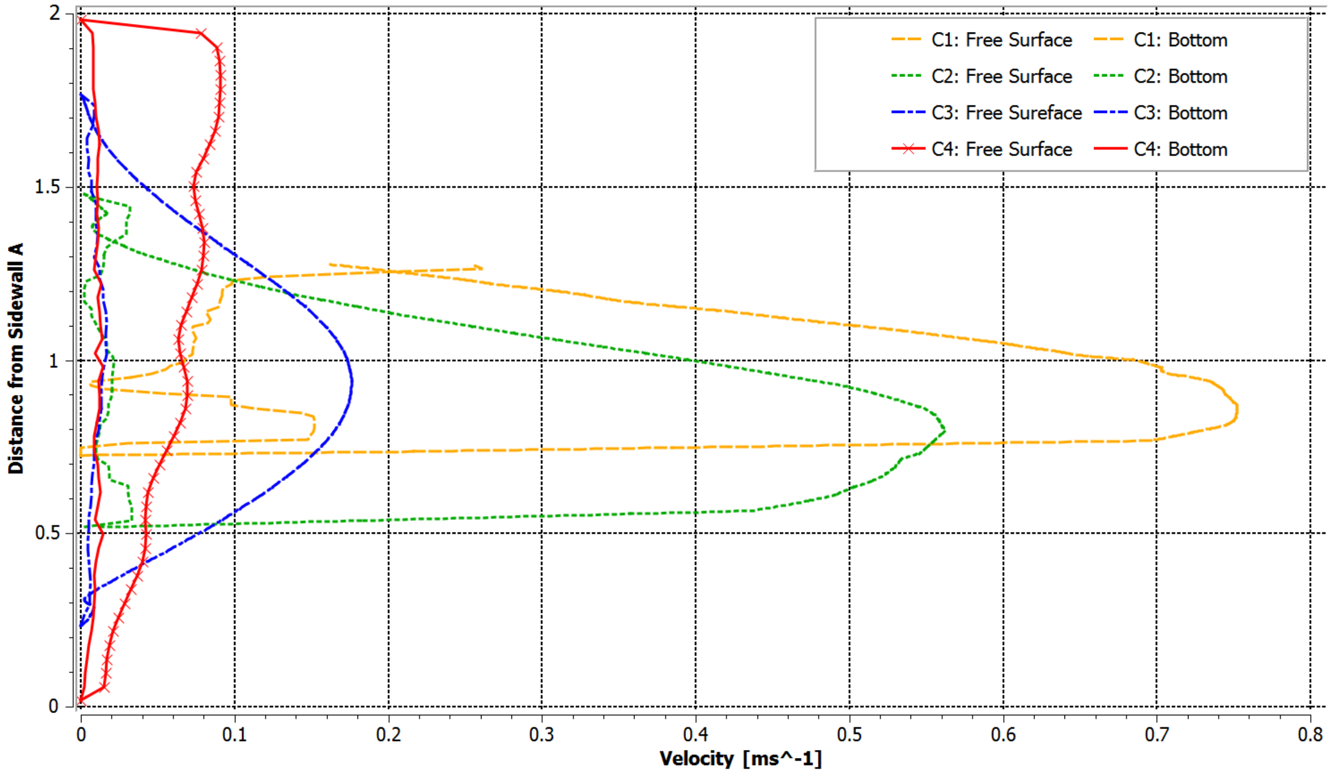

Fig. 14. Overall Velocity profiles (Test Case C)

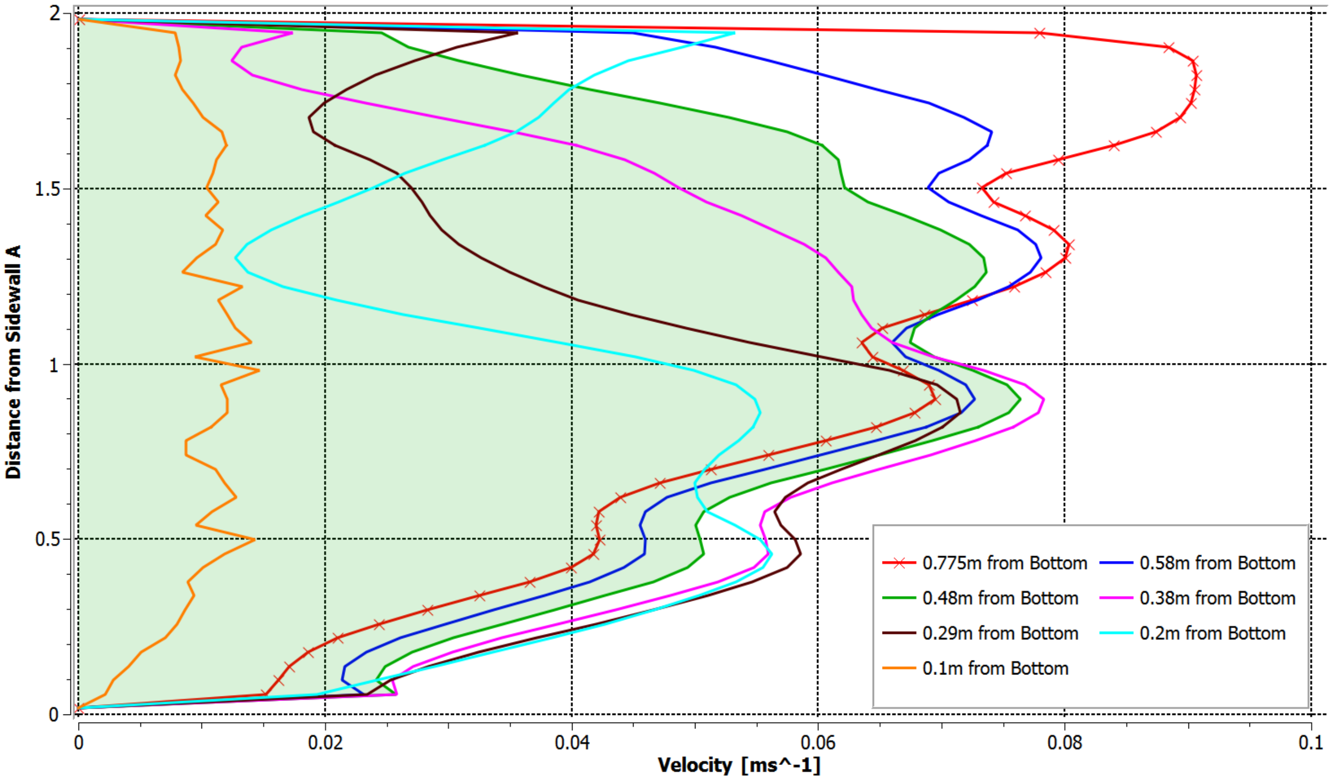

Fig. 15. Velocity profiles at Section C4 (Test Case C)

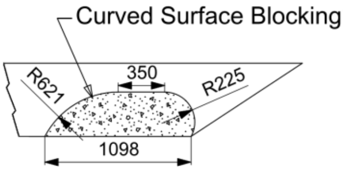

Fig. 16. Geometry of the Curved surface blocking.

C. Test Case C: The Curved Blocking Approach

Due to the inherent complex geometry of the pump intake, it is impractical to control the flow behavior using the standard blocking approach.

To optimize the fluid flow in the final intake geometry, the phenomenon of Wall Attachment was implemented to design a unique blocking arrangement to “actively” control the fluid flow behavior.

The construction of the curved blocking prepared involves a single block, Fig. 16; having two curved-parametric surfaces in contact with the moving fluid, separated by a flat surface placed between them, which provides a longitudinal surface for the fluid to adhere to and in-turn provides time to the flowing water to optimize its directional momentum for uniform flow. The two curved surfaces control the direction of flow by allowing the water to latch onto them and thereby follow them along their curvatures. The extent to which water trails along their curved surface depends on the shape and the smoothness (or roughness) of the surface.

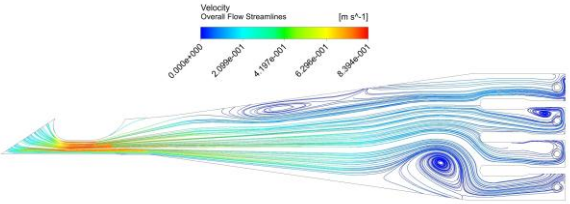

Fig. 17. Top view of flow velocity streamlines (Test Case C)

In Fig. 17, the velocity streamlines achieved with the use of a curved surface blocking show a tremendous improvement in the overall flow profile. The flow is sufficiently diverted form Sidewall A to be minutely directed towards Sidewall B. The recirculation of water produced in-front of the pier-wall between P1 and P2, has a very low intensity and as such, imparting minute effects on the overall flow characteristics.

The velocity profiles generated at Sections C1 to C4 corroborate the remarkable improvement noticed in uniformity of the flow, as well as the extent of the decreased velocity magnitudes. The curves of the velocity profiles produced in Fig. 14, show a marked increase in fluid flow symmetry. Indeed, the velocity profile generated at section C3, closely emulates a ‘Normal/Gaussian distribution’ curve. This gives a strong indication of the effectiveness of employing a Curved blocking approach.

Further, in Fig. 15 the velocity profiles sampled in section C4 at various depths show a massive drop in the overall flow velocity; the peak velocity magnitude of 0.09m/s is achieved 1.8m from the Sidewall A. The green-shaded area represents the practical near perfect flow distribution- 0.48m above the bottom surface. This consecutively produces lower swirl magnitudes and decreased deposition/ingestion of dispersed sediments.

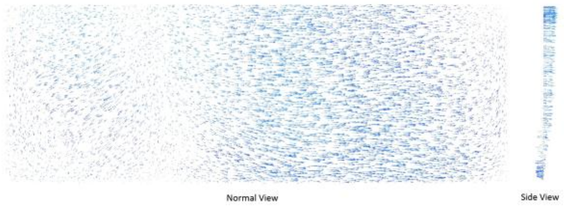

Fig. 18. Velocity vector plots at section C4 (Test Case C)

Fig. 18 shows the Normal View of the velocity vector plot with a decreased amount of swirl in the fluid flow. Additionally, the magnitude of the velocity vectors is low and very much uniform throughout section C4. The side view of the vector plot in Fig. 18, confirms the decrease in the flow velocity magnitudes throughout the depth of the fluid flow domain. A feeble back circulation of flow having very low velocity magnitude can be observed at the bottom of the fluid flow domain as seen from the side view of the vector plots.

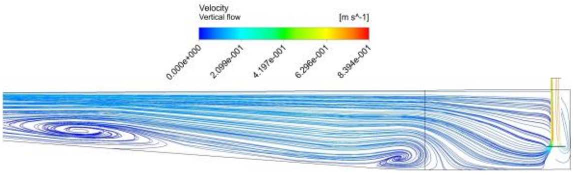

Fig. 19. Vertical velocity streamlines (Test Case C)

Due to the uniformly distributed flow and a decrease in the velocity magnitudes throughout the flow, there is a more discernible improvement in the vertical flow streamlines, Fig.19. The regions of fluid recirculation formed are of very low swirl intensity compared with Test cases A and B. The major improvement realized is the thorough elimination of the secondary swirl as was formed due to the water back-circulating from the back-wall of the pump chambers. This results in enhanced pre-swirl reduction throughout the pump chambers and successively near the bell-suction of the pumps.

RESULTS AND DISCUSSION

A unique, site specific approach is required for optimizing the flow pattern for each individual pump intake. The extent to which the flow is to be optimized largely depends upon the costs, method of construction and the amount of time it shall take to construct the intake geometry; which includes implementing the flow blocking arrangements.

Test case C demonstrates the efficacy of implementing a single curved blocking constructed in the approach channel of the intake, compared to the inefficient and time consuming implementation of multiple-standard guide walls. Considering the undesirable-inherent flow characteristics of the original intake geometry, a large number of guide walls would need to be constructed to optimize the flow, which greatly increases the construction costs and is an impractical strategy to implement.

The results obtained by using a single curved blocking approach are far more superior. This is primarily because the fluid changes its direction as it follows the slope of a curved surface, whereas, in the case of standard guide walls- it is forced/guided against its natural flow, which leads to augmented turbulent flow characteristics.

One of the more intriguing and convenient aspects of using a curved blocking is that, the fluid flow behavior can be controlled not only by the geometry of the curved blocking, but also by the smoothness of the curved surface in contact with the fluid flow. The smoother the surface construction, the fluid will follow along the surface more readily. Consequently, rather than limiting our approach to just varying the geometry of the blocking, the variation in surface roughness value (for the same geometry) can also be explored for fine tuning the fluid flow; this additional control parameter can be of great use, especially when the geometry of the intake does not allow much design alterations.

Compared to Test cases A and B, the strength of swirl formation is greatly reduced throughout the intake domain for Test case C; this is due to the noted radical descent in the velocity and the augmented uniformity of the fluid flow in Test case C.

CONCLUSION

The following conclusions can be readily drawn from the study conducted:

1. A Curved Blocking approach is superior and comparatively more efficacious than standard blocking arrangements in optimizing the fluid flow characteristics.
2. A Curved Blocking can be implemented cost effectively for intakes having a disproportionate flow pattern.
3. Keeping the same curved blocking geometry, the flow can be controlled by the variation in the wall surface roughness of the curved surface blocking in contact with the fluid flow.
4. In addition to the design of flow blockings, the curved surface approach can readily be employed at various flow separation regions of the intake geometry for streamlining the fluid flow.
5. Experimental studies are required to better understand the effects of using a curved blocking approach and also to compare the numerical results obtained with the experimental observations.

ACKNOWLEDGMENT

We humbly thank Mr. Sandeep Golekar, R&D Department-Kirloskar Brothers Limited, Kirloskarvadi for all his practical expertize and comprehension of the subject.

REFERENCES

[1] Adams Thomas, Grant Christopher and Watson Heather A simple algorithm to relate measured surface roughness to equivalent sand-grain roughness [Journal] // International Journal of Mechanical Engineering and Mechatronics. - [s.l.] : Avestia Publishing, 2012. - 1 : Vol. 1. - Article ID: 008. - ISSN: 2929-2724.

[2] American national standard for Rotodynamic Pumps for Pump intake design [Book]. - Parsippany, New Jersey : Hydraulic Institute (ANSI), 2012. - Vol. 9.8. - Approved American National Standard: ANSI.

[3] Bayeul-Laine A.C., Bois G. and Issa A. Numerical simulation of flow field in water-pump sump and inlet suction pipe [Conference] // 25th IAHR Symposium on Hydraulic Machinery and Systems. - [s.l.] : IOP Publishing Ltd, 2010.

[4] Cravero Carlo [et al.] CFD analysis of unsteady multiphase flows in sea water pumping stations [Journal] // Termomeccanica Pompe- La Spezia.

[5] Fletcher B.P. Pumping station, field performance [Report] : Technical Report HL-79-7. - Vicksberg, Miss : U.S. Army Corps of Engineers, 1979. - Waterways experiment station.

[6] Fraser W.H. Hydraulic problems encountered in intake structures of vertical Wet-pit pumps and methods leading to their solution [Journal] // Trans. ASME. - 1953. - 4 : Vol. 75.

[7] Hirsch Charles Numerical computation of internal and external flows [Book]. - [s.l.] : Elsevier, 2007. - Second : Vol. 1. - Fundamentals of computational fluid dynamics.

[8] Jones Garr M. [et al.] Pumping station design [Book]. - [s.l.] : Elsevier, 2006. - Third.

[9] Lin-guang Lu, Zhi-gao Cao and Ji-ren Zhou Study on hydraulically optimum design of pump sumps [Journal] // Journal of Hydrodynamics. - 1996. - pp. 42-51. - Ser.B, 8(2).

[10] Padmanabhan M. A parametric study of the hydraulic performance of containment emergency sumps with vertical double outlets [Report] : Technical report: 48-82/M398F / Alden Research Laboratory ; Worcester Polytechnic Institute. - 1982. - Holden Mass.

[11] Prosser M.J. The hydraulic design of pump sump and intakes [Report]. - [s.l.] : British Hydromechanics Research Association, 1977.

[12] Schlichting Dr. Hermann Boundary-Layer theory [Book] / trans. Kestin Dr. J.. - [s.l.] : McGraw-Hill Book Company, 1979. - Seventh Edition.

[13] Taylor J.B., Carrano A.L. and Kandlikar S.G. Characterization of the effect of surface roughness and texture on fluid flow: past, present and future. [Journal] // International Journal of Thermal Sciences. - 2006. - pp. 962-968.

[14] Tu Jiyuan, Yeoh Guan Heng and Liu Chaoqun Computational Fluid Dynamics: A practical approach [Book]. - [s.l.] : Elsevier, 2008. - First. - Near wall treatments.