iJRASET
International Journal For Research in Applied Science and Engineering Technology

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY

Volume: 9 **Issue:** VII **Month of publication:** July 2021

DOI: https://doi.org/10.22214/ijraset.2021.36430

www.ijraset.com

Call: 08813907089 | E-mail ID: ijraset@gmail.com

International Journal for Research in Applied Science & Engineering Technology (IJRASET)
ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

Use of CFD for Analysis of HYDRAM- An Approach

Ajay B. Mahajan[1], C. C. Handa[2], A. P. Ninawe[3]
[1]*M.Tech Scholar, K.D.K. College of Engineering, Nandanvan Road,Nagpur.*
[2]*Professor & Head, K.D.K. College of Engineering, Nandanvan Road,Nagpur.*
[3]*Assistant Professor, K.D.K. College of Engineering, Nandanvan Road,Nagpur.*

***Abstract:** This paper presents the analysis of the existing hydraulic ram pump performance using computational fluid dynamic (CFD). The study specifies that mass flow rate at inlet and outlet on CFD. These are based on a systematic study of hydraulic ram pump and testing of on hydraulic ramp pump model. For which we consider literatures reviews & some of them are used for the analysis. CFD approach is used in this paper for estimating the water discharge.*
***Keywords:** CFD, hydraulic ram pump, Water discharge, Mass flow rate.*

I. INTRODUCTION

An analysis of Hydraulic ram pump presents some of the most challenging problems faced by engineers in domestic use, agriculture use, industry use, because simulations need to address the moving parts in an appropriate manner. In this way, computational fluid dynamic (CFD) tools have become essential in aiding in the pump development. Hydraulic ram pump become more and more complex in order to address the higher output. Therefore flow characteristics are constantly evolving in order to keep CFD in pace with pump innovations. Flow characteristics help to optimize various parameters of a pre-existing.

In this study, a commercial CFD program SOLIDWORKS Flow Simulation-Computational Fluid Dynamics (CFD) Software is used. Computational Fluid Dynamics or CFD is a technique that deals with the solution of fluid flow fields through numerical analysis. SOLIDWORKS Flow Simulation is CFD software designed for the everyday SOLIDWORKS user and analyst. It provides dynamic feedback on the fluid flow and thermal performance of their products. With parametric optimization capabilities, users can automate the design and analysis process to discover the best iteration of their design within the familiar SOLIDWORKS CAD environment.

II. LITERATURE REVIEW

1) The paper entitled by Piyush B. Shende, A. P. Ninawe, Dr. S. K. Choudhary, is mass flow through the waste valve of HYDRAM is more, so to reduce it an enhancement is done and a new design is created. These are based on a systematic study of hydraulic ram pump and testing of on hydraulic ramp pump model. For which we consider literatures reviews & some of them are used for the analysis.

2) MN Harith, R A Bakar , D Ramasamy, Ma Quanjin is studied the effect of improved design to significant effect on flow analysis and simulation study. This study set out to evaluate how effective the new design towards improving the overall pumps performance. These experiment confirmed that the by adding control mechanism to the newly design component delivery and waste valve have enhance about 20% more efficiency than current design.

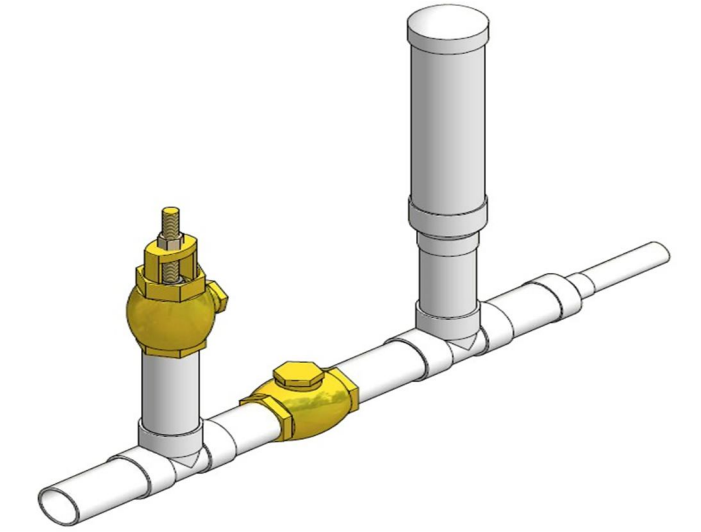
3) Wojciech Sobieski, Dariusz Grygo is studied he results of a study investigating the equilibrium of forces acting on the closing element of the impulse valve in a water ram at the end of the acceleration stage. Acceleration is one of the three main stages in the working cycle of a water ram

4) Jedsada Juruta, Nattapong Tabtimhin and Yosawat Limpongsa studied that an increase in the supply head tends to increase the supply flow rate, delivery flow rate, delivery head, and the overall efficiency of the pump. An increase in air chamber pressure tends to decrease the overall efficiency of the pump. However, there was no significant difference on the HRP performance over a wide range of flow conditions when air chamber pressure was varied. An increase in waste valve beats per minute tends to decrease the supply flow rate, delivery flow rate, and delivery head. But it tends to increase the head ratio, the flow-rate ratio, and the overall efficiency of the pump

ISSN: 2321-9653; IC Value: 45.98; SJ Impact Factor: 7.429
Volume 9 Issue VII July 2021- Available at www.ijraset.com

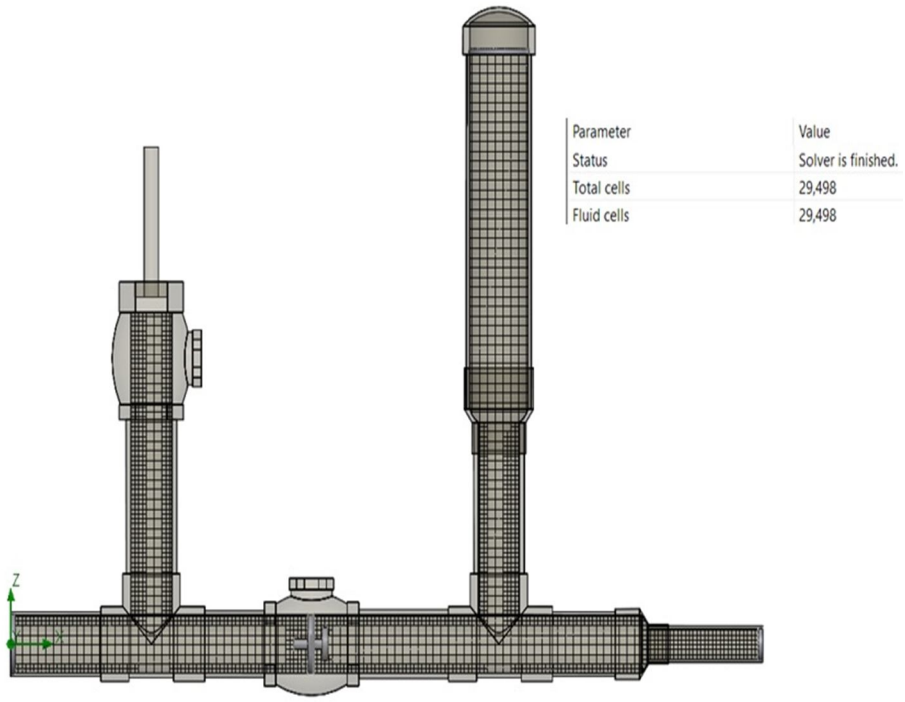
III. METHODOLOGY

The collected data is used to design the model. Designing software is Autodesk-Inventor
The domain is created in Autodesk-Inventor software and imported into SOLIDWORKS CFD software for the final results.



A. Meshing

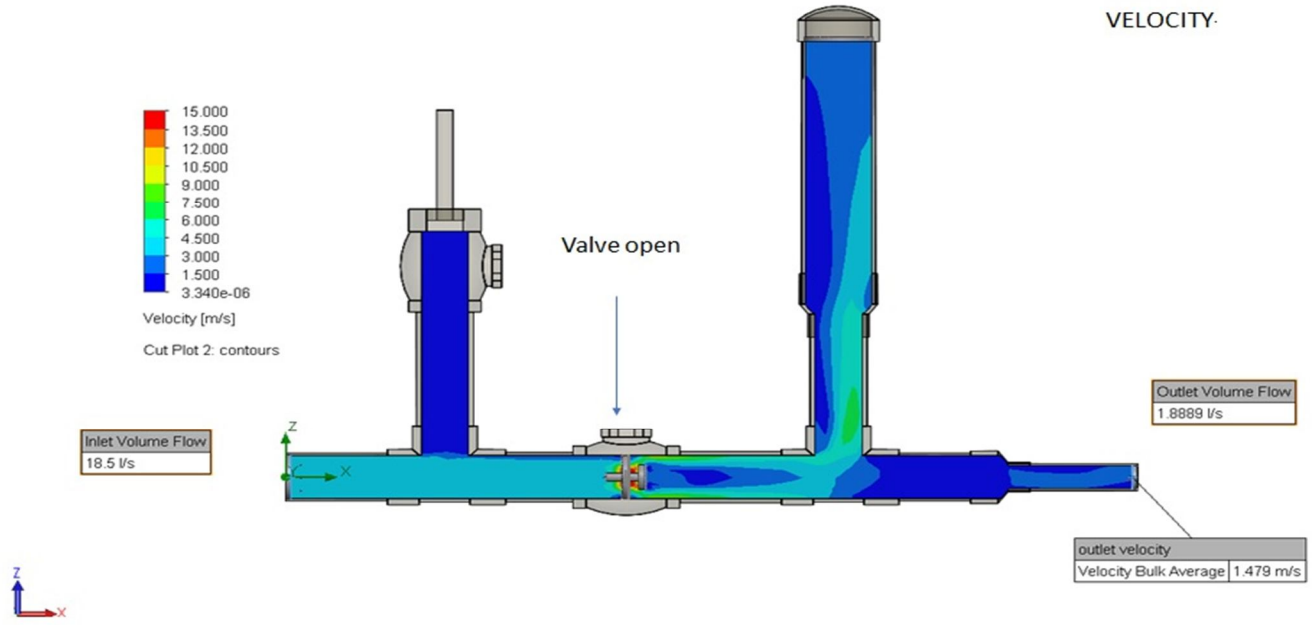
In this study unstructured mesh is used for the geometry. There are two main advantage of unstructured meshing, one it occupies a very less memory in computer and another one is it requires very less time to compute and solve the equation to get solution. Takes less time to analysis delivered appropriate and accurate result.

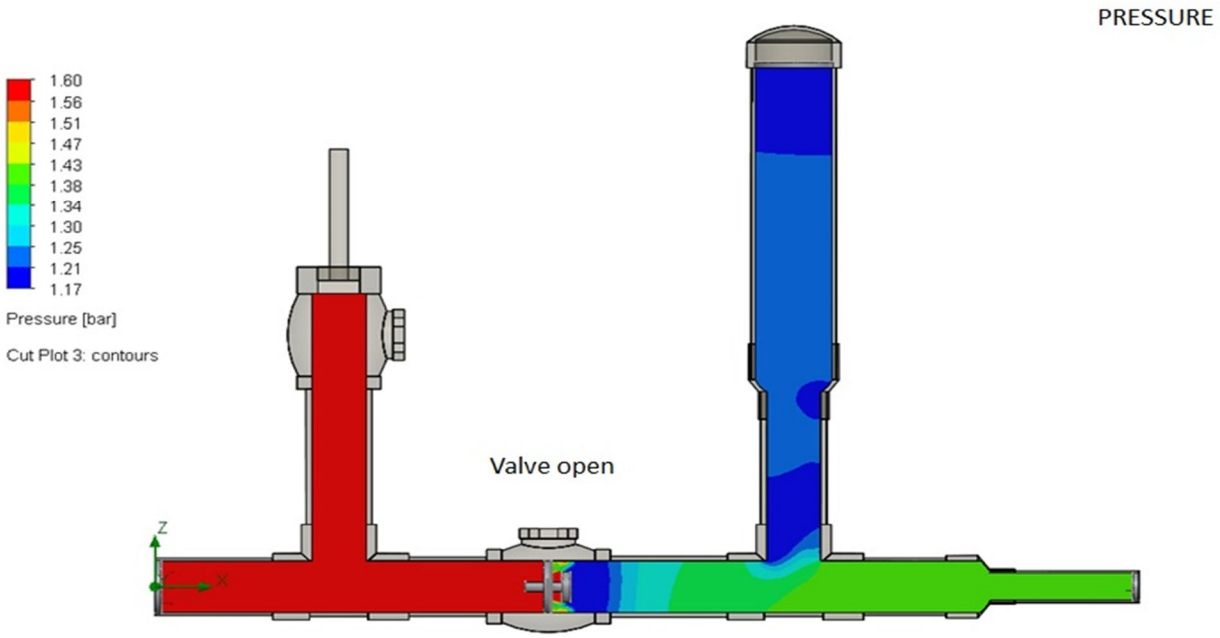


The geometry is being meshed with the help of triangular cell and quadrilateral cell. The no. Of cells 26498. All of CFD, in one form or another, is based on the fundamental governing equations of fluid dynamics—the continuity, momentum, and energy equations.

IV. CFD ANALYSIS

The dynamic grid approach is used to treat the moving piston in the computational area. In other words, the grid generation approach was used to treat the moving piston as a moving solid body in the computational domain without generating completely new grids. Piston moves upward and downward position considering open condition and closed condition of waste valve & NRV. The model structure is unstructured grid and to setup boundary condition for moving piston. Total number of computational cells was used about 26498. The above mentioned solver setting is used to get appropriate and accurate flow pattern. Following figures shows the flow pattern after analysis. Figures present the velocity vectors, pressure vectors for different heads.



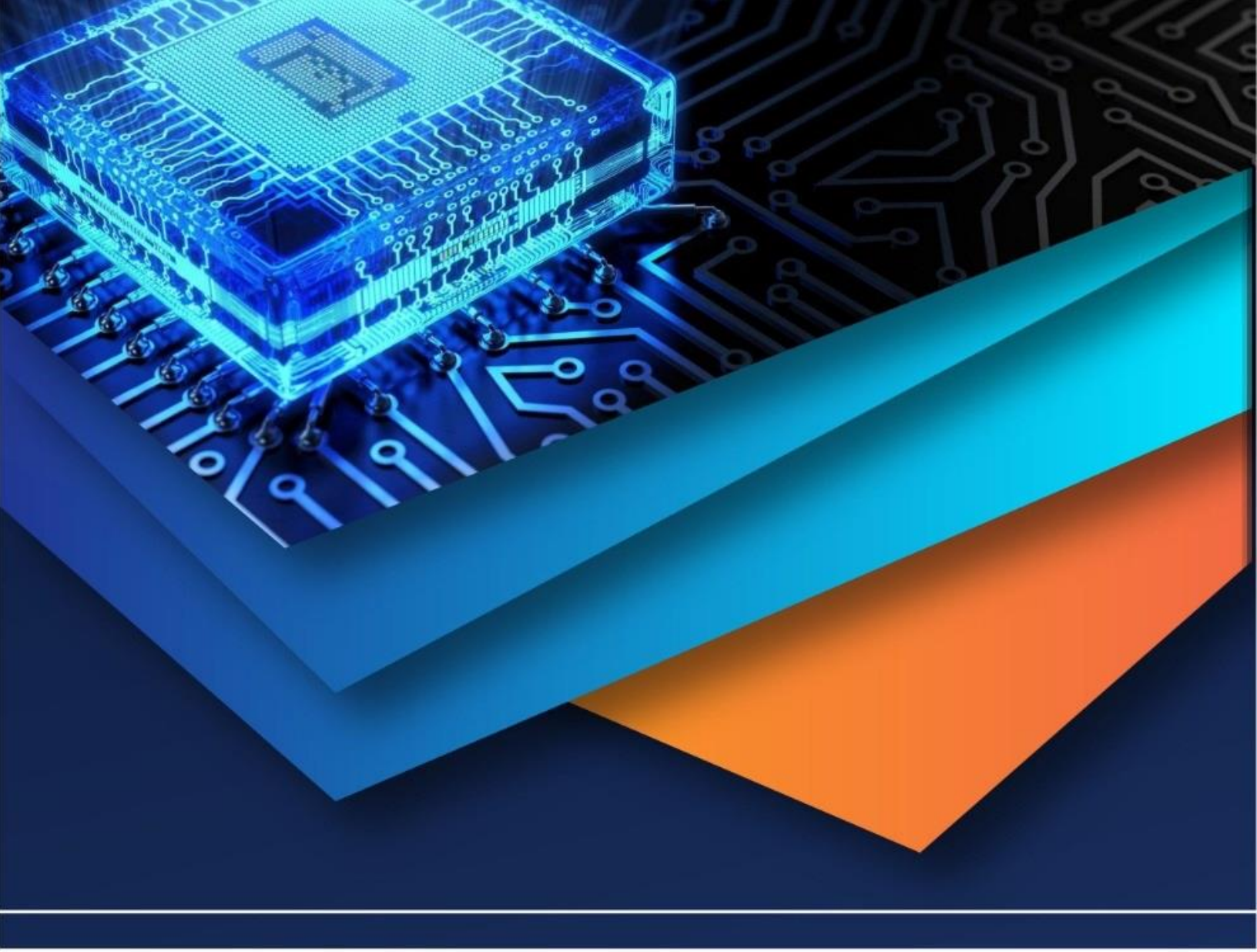


V. CONCLUSIONS

Using CFD, analysis of existing model is done. The mass flow rate at the outlet is determined. CFD analysis can be used for estimating the discharge in HYDRAM. Also various complex velocity and pressure behavior performance are identified which helps for further optimization of efficiency of HYDRAM

REFERENCES

[1] Piyush B. Shende, A. P. Ninawe, Dr. S. K. Choudhary "Analysis and Enhancement of Hydraulic Ram Pump using Computational Fluid Dynamics (CFD)" in IJIRST –International Journal for Innovative Research in Science & Technology| Volume 2 | Issue 03 | August 2015
[2] MN Harith* , R A Bakar , D Ramasamy, Ma Quanjin "A significant effect on flow analysis & simulation study of improve design hydraulic pump" in 4th International Conference on Mechanical Engineering Research (ICMER2017
[3] Wojciech Sobieski, Dariusz Grygo" FLUID FLOW IN THE IMPULSE VALVE OF A HYDRAULIC RAM" Technical science scientific journal ISSN 1505-4675
[4] Shuaibu Ndache Moammad, ―Design and Construction of a Hydraulic Ram Pump‖,Leonardo Electronic Journal of practices and Technologies,ISSN 1583-1078,Issue 11,July-Dec.2007,P.59-70
[5] H. W. Iversen, "An analysis of the hydraulic ram," Journal of Fluids Engineering, vol. 97, no. 2
[6] B. W. Young, "Design of hydraulic ram pump systems," in Proceedings of the Institution of Mechanical Engineers, Part A, vol. 209
[7] Gray CAM (1953), "The Analysis of the Dissipation of Energy in Water Hammer" Proc. Am.Soc. Ci6. Engrs., Vol.119,pp. 1176-1194.



10.22214/IJRASET

45.98

IMPACT FACTOR: 7.129

IMPACT FACTOR: 7.429

INTERNATIONAL JOURNAL FOR RESEARCH

IN APPLIED SCIENCE & ENGINEERING TECHNOLOGY