International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163
Volume 1 Issue 5 (June 2014) http://ijirae.com

Transient Analysis of Centrifugal Pump for Thermic Fluid Application

Aniket Suryawanshi
Mechanical Engineering Department,
D.A.C.O.E., Karad.

S.M.Gunadal
Mechanical Engineering Department,
V.J.T.I., Mumbai.

Abstract*—Flow analysis and heat transfer analysis in centrifugal pumps has long been an intensive subject of research. Computational Fluid Dynamics (CFD) is the present day state-of-art technique for fluid flow analysis. Numerical simulation of 18 m3/hr capacity centrifugal pump carried out using commercial CFD package FLUENT is presented. The steady state simulations were carried out using Reynolds averaged Navier-Stokes (RANS) equations. It was found that non-uniformities are created in different parts of the pump at off-design conditions which result in the decrease in efficiency.

Key words:--Centrifugal pumps, CFD, Transient analysis, numerical simulation

I. INTRODUCTION

A centrifugal pump is a rotodynamic machine that uses a rotating impeller to increase the pressure of a fluid. They are widely used for liquid transportation in different sectors. Their operating range spans from full-load down to close to the shut-off head. In order to develop a reliable machine for this highly demanding operation, the behavior of the flow in the entire pump has to be predicted before they are put in actual use. This requires critical analysis of highly complex flow in the pump which is turbulent and three dimensional in nature. The flow analysis through experiments or model testing is considered to be time consuming, tedious and expensive. CFD is the present day state-of-art technique in fluid flow analysis. In recent years, most of the industries are using CFD as a numerical simulation tool for flow analysis of centrifugal pumps. Due to the development of CFD code, one can predict the efficiency of the system as well as observe actual behavior. One can find the root cause for poor performance by using CFD analysis of the system.

While working of pump at high temperature the heat transfer takes place and thus due to that the different components gets heated and the mechanical properties gets changed. Thus due to that it is necessary to design the every component of pump in such a way the pump can safely handle the high temp liquids with good performance. Computational fluid dynamics (CFD) analysis is being increasingly applied in the design of centrifugal pumps. With the aid of the CFD approach, the complex temperature distribution in pump, which is not fully understood yet, can be well predicted, to speed up the pump design procedure. Thus, CFD is an important tool for pump designers.

II. GEOMETRY CREATION

A. Introduction

For analysis in ANSYS FLUENT software model with correct dimensions should be drawn so that proper analysis can be done with help of software. Modeling of Pump set was done in PTC Creo v 2.0, The .stp file for whole assembly is created and then the meshing is carried into ICEM CFD software. These files then imported into tin format which can be used in ANSYS software for further processing.

Fig. 1. Geometry creation

B. Meshing

For meshing the geometry was imported then the topology was built for proper pasteurization of curves from the surface model by build topology command.

Fig.2. Meshing

Geometry imported in ICEM CFD.

C. Build topology

Need for Build Topology:

- For extraction of curves from surface geometry.
- Enabling model diagnosis.
- Curve form basis of geometry repair.

Build Topology Procedure:

I. For building topology "build topology" button is selected.
II. Under the build topology tab the minimum tolerance is defined as 0.05mm.
III. Curve healing is selected with its respective option button and then topology is built.
IV. Build topology red Lines indicating multiple surface sharing curve.

Now as the topology is available the geometry can be taken for subsequent geometry cleanup.

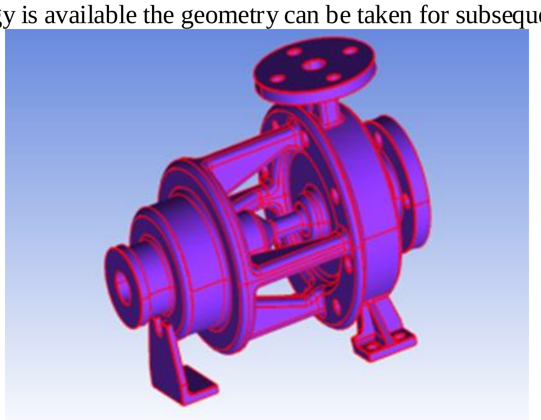


Fig.3. Build Topology

D. Geometry Clean-up

Geometry clean-up is the process of cleaning up of the geometry for making the sections closed for the analysis and in this same processes the geometry is cleaned in such a way that the simulation can be performed and hole and un patched surfaces are healed.

Need for Geometry Clean-up:

- For removing all the irregularities of the geometry such as holes, patches etc.
- For making the geometry closed for the fluent analysis.

Geometry Clean-up Procedure:

For geometry clean-up different tools are used such as:

- Close Holes
- Remove Holes

- Stich / Match edge.
- Split folded surface.
- Adjust varying thickness.
- Modify surface normal.

Now as these tools are utilised for the cleaning the geometry and reducing most of the single curves for making the geometry closed we can proceed for subsequent processes. The next main process is surface meshing.

E. Convection surface creation

Natural convection phenomenon is considered for Analysis of Pump set.

1. External surfaces of casing are selected and created named selection as convection.
2. Also the area through which Fluid is sucked by Pump is denoted as Inlet and for discharge it's named as Outlet.

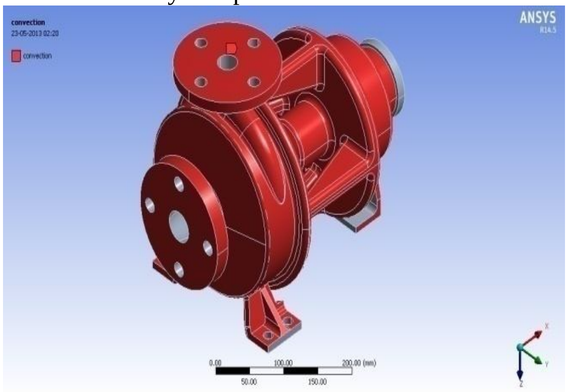


Fig. 4. Convection surface creation

Image shows Natural convection region i.e region which comes into contact with Air.

F. Surface Meshing

Surface meshing is the process of meshing the surface with different surface meshing types such as:

- All Tri
- Quad with one Tri
- Quad Dominant
- All Quad

With selection of different meshing methods such as:

1. Autoblock
2. Patch Dependent
3. Patch Independent
4. Deluanay

Once the surface is meshed the subsequent part meshing can be done now with different part meshing tools.

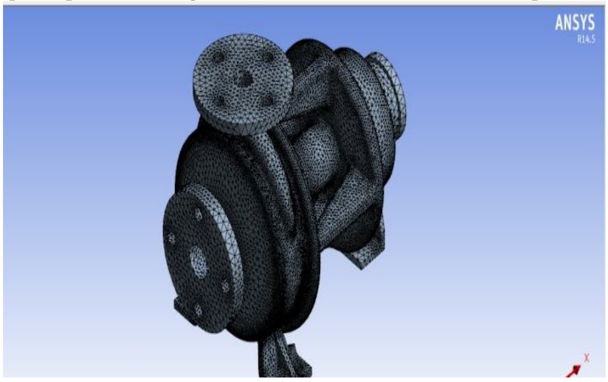


Fig.5.Surface Meshing

Image shows Meshing in ICEM CFD

G. Boundary Conditions

Boundry conditions are defined at the different spaces as undersign:

- Ambient Temperature = 300 C
- Mass Flow rate = 4.5 Kg/Sec
- Water inlet Temperature = 3000 C
- Convection Coeffiecient=10 W/m2K

H. Meshing Quality Check:

This is done for checking the mesh quality is weather good or not figure below is shoeing the mesh quality for our geometry.

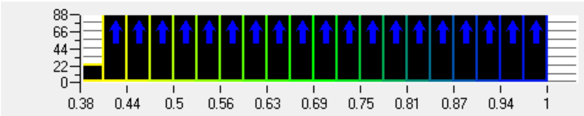


Fig.6. Mesh quality check

III. RESULTS AND DISCUSSIONS

A. Temperature contour for transient temperature distribution for time step 3600 seconds:

Max. Temp. =350 0C Min. Temp. =199 0C

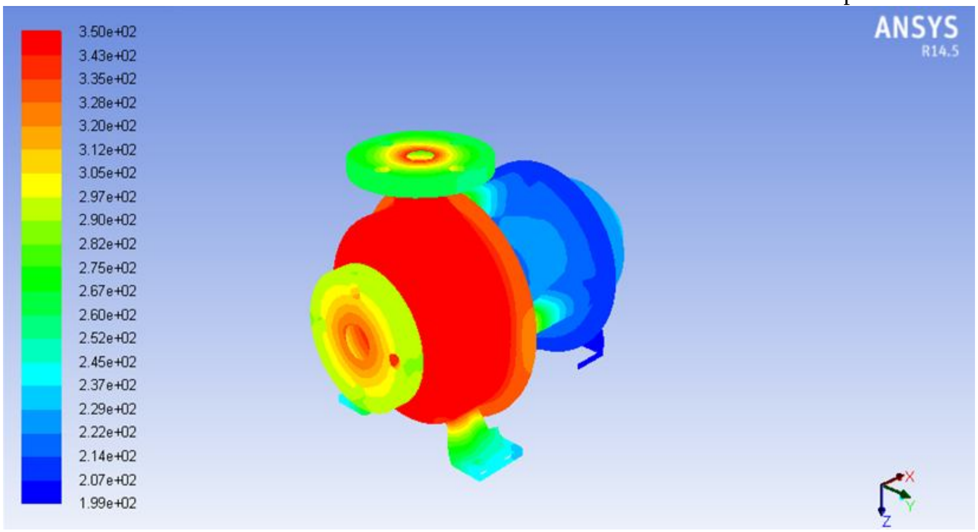
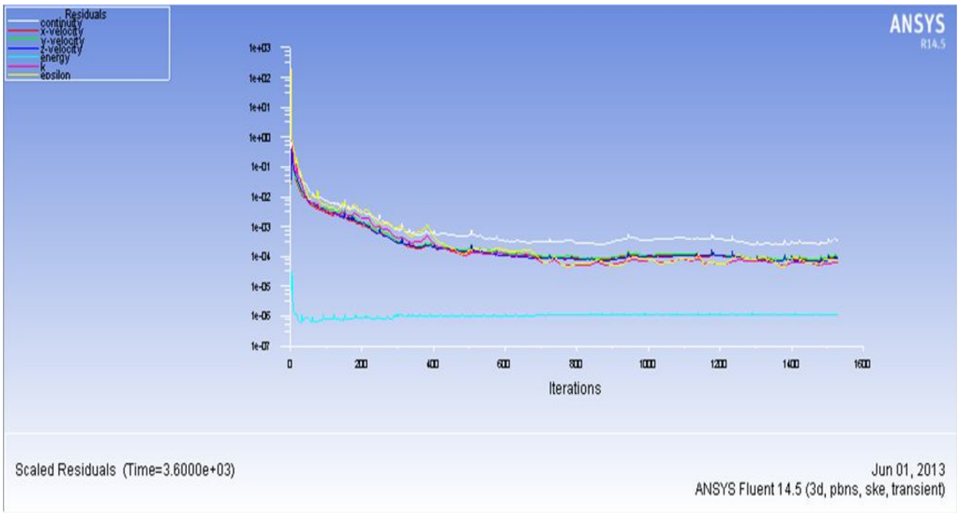


Fig.8. Temperature contour for transient temperature distribution for time step 3600 seconds



Graph.2. Scaled residue Vs no. of iterations for 3600 seconds.

B. Transient temperature distribution for time 3600 seconds:

On X-Axis – Scaled residue. (Exponential increment) On Y-Axis – No of iterations.(Standard increment)
Total no. of iterations= 1530 No. of Time step =60 seconds.
Time step size =60 Energy line (line color) OR (Temperature line)

We resulted that scaled residue for this calculation is converged
Total time= 3600 seconds. Total no. of iterations= 1530
Steady state reaching iteration number= 710
Threfore Steady State after 1670 seconds is achieved.

CONCLUSION

A numerical model of pump has been successfully generated and the complex internal flow fields are investigated by using the ANSYS-FLUENT software. Simulation results are obtained at surface of components of pump. The temperature increases gradually and continuous from the steady state temperature. The temperature of material of component increases slowly and continuous from steady state. Temperature of the working fluid decreases from suction nozzle to motor end side bearing.

Results were concluded on the basis of the following points:

- Temperature of material should not exceed beyond the melting points of the corresponding components.
- Temperature from the bearing housing changes abruptly due to low thermal conductivity materials used.

REFERENCES

[1] Erik dick, jan vierendeels, Sven serbruyns and john vande voorde "PERFORMANCE PREDICTION OF CENTRIFUGAL PUMPS WITH CFD-TOOLS" tq0405e7/579 26 I 2002 BOP S.C.,
[2] Weidong Zhou, Zhimei Zhao, T. S. Lee, and S. H. Winoto (2003) "Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics". International Journal of Rotating Machinery, 9(1) 49–61, 2003 Copyright 2003 Taylor & Francis 1023-621X/03
[3] L. Pullum, L. J. W. Graham, and M. Rudman (2007) "Centrifugal pump performance calculation for homogeneous and complex heterogeneous suspensions". The Journal of The Southern African Institute of Mining and Metallurgy, Volume 107, Referred paper, June 2007.
[4] M. H. Shojaee Fard, F. A. Boyaghchi and M. B. Ehghaghi (2006) "Experimental Study and Three-Dimensional Numerical Flow Simulation in a Centrifugal Pump when Handling Viscous Fluids". IUST International , Journal of Engineering Science, Vol. 17, No.3-4, 2006, Page 53-60
[5] Khin Cho Thin, Mya Mya Khaing, and Khin Maung Aye, (2008) "Design and Performance Analysis of Centrifugal Pump". World Academy of Science, Engineering and Technology 46 2008.
[6] Akira Goto, Motohiko Nohmi, Takaki Sakurai, Yoshiyasu Sogawa, (2002) " Hydrodynamic Design System for Pumps Based on 3-D CAD, CFD, and inverse Design Method" published by EBARA Corporation, Tokyo, Japan.
[7] Gandhi, B. K., Singh, S. N., and Seshadri, V., (2001), ''Performance Characteristics of Centrifugal Pumps,'' ASME J. Fluids Eng., 123, pp. 271–28.
[8] ANSYS CFX-Manual, (2006), Published by, ANSYS CFX, Release 11.0, December, 2006, ANSYS, Inc.
[9] American standard society for "Rotodynamic (centrifugal) pumps for design and application" sponsor by Hydraulic

