Available online at www.sciencedirect.com

ScienceDirect

Energy Procedia 101 (2016) 742 – 749

Energy Procedia

71st Conference of the Italian Thermal Machines Engineering Association, ATI2016, 14-16 September 2016, Turin, Italy

CFD Analysis of a non-Newtonian fluids processing pump

Nicola Aldi[a], Carlo Buratto[b], Michele Pinelli[a], Pier Ruggero Spina[a], Alessio Suman[a], Nicola Casari[a,]*

[a]*Engineering Department in Ferrara (ENDIF), University of Ferrara, Via Saragat 1, 44122 Ferrara, Italy*
[b]*Fluid-a s.r.l. , via Benedetto Zallone 19, 40066 Pieve di Cento (BO), Italy*

Abstract

Pumps are among the most spread machines in industrial facilities. In this work a comparative CFD analysis using different software is presented. The three-dimensional flow in the semi-open impeller and volute of a centrifugal pump is numerically simulated. The main advantage of semi-open impeller centrifugal pump is its efficiency which can be considered constant thanks to the clearance adjustment. In addition this kind of impeller is less likely to clog with solid bodies (important in case of slurry-processing). The open impeller has all the parts visible, so it is easier to inspect for wear and damages. Eventually it is lighter than a shrouded impeller: it can spin faster. The stress due to centrifugal force is indeed a limit for the speed of this machines. On the other hand its main disadvantage if compared to a shrouded pump is its lower efficiency due to the heavier tip leakage. In addition it cannot be employed in case of explosive products: the risk of contact between impeller and volute causing sparks is not negligible.

The simulations have been carried out using both open-source and proprietary software: OpenFOAM®, PumpLinx ® and ANSYS-CFX ®. The performance of the machine handling both Newtonian and non-Newtonian fluids are also investigated. The numerical models and the results of the different computational strategies were compared with the experimental data and the accuracy of different software is evaluated in the case of Newtonian model.

It is well known that the performance of a centrifugal pump drops processing a viscous fluid. Even so the behavior during the pumping of non-Newtonian fluids has not been investigated so far. The non-Newtonian fluid processed is a shear-thinning fluid (the apparent viscosity decreases with an increase stress). The slurries which are usually processed in the food industries, chemical plants and oil&gas processes show a usual behavior which correspond to this kind of model.

Peer-review under responsibility of the Scientific Committee of ATI 2016.

Keywords: turbomachinery ; centrifugal pumps; semi-open impeller; non-Newtonian Fluid;

* Corresponding author. Tel.: +39-053-297-4964; fax: +39-053-297-4970.
E-mail address: nicola.casari@unife.it

Peer-review under responsibility of the Scientific Committee of ATI 2016.
doi:10.1016/j.egypro.2016.11.094

1. Introduction

Semi-open impeller centrifugal pumps are widely used in industrial facilities, especially when the risk of clogging (slurry processing pumps) is not negligible. This kind of pumps differs from the "closed" one due to the lack of the front shroud. This category of pump benefits from this feature: both the wider range of high constant efficiency and the less clogging likelihood derives from the adjustable gaps between the tip of the blades and the casing. Although this machine is spread in several plants, it has not been investigated deeply, particularly if matched with the processing of a non-Newtonian fluid.

It is well known that the performance of a centrifugal pump drops processing a viscous fluid. Several analysis have been carried out for the performance variation, when the handled fluid is different from water. Among the other, Li [1-2] investigated the effect the oil has on the losses and the flow field in centrifugal pumps. His work concerns the modeling of this category of turbomachinery and its behavior whit high viscosity oil. He supported his numerical results with a wide range of experimental measurements, showing that the handling of viscous fluid has major consequences on the behavior of the machines, such as: increase in the disc friction losses (affecting the slip coefficient), wider wake near the suction side of the blades and reduction of flow in the impeller and volute.

On the other hand, the behavior of the pump in case of the handling of viscous fluid is strongly dependent on the geometry. Changing some geometric characteristics of the impeller, as shown by Shojaeefard [3], improve their performance. Particularly, major variations in the efficiency and head given by the pump is seen in case of changes made to the outlet angle and to the impeller passage width.

Few examples are available in the open literature for centrifugal pumps processing a non-Newtonian fluid. Buratto et al. [4] analyzed a large-size industrial food processing centrifugal pump designed for tomato paste. They simulated the behavior of the machine when the non – Newtonian fluid processed is different from the design one. Their study has shown that a fluid viscosity increase affects pump performances coherent with literature data. Then, pump curves and 3D flow structures obtained with non-Newtonian fluid are similar to those obtained with high viscosity Newtonian fluids. In particular, the comparison of the impeller curves of total pressure rise, disc friction losses and efficiency has highlighted a very close trend between tomato paste and Newtonian virtual fluid with a dynamic viscosity value of 1 Pa s.

In addition, in this work a semi-open impeller pump has been considered. The main disadvantage of this kind of machines is the tip leakage: the fluid flows from the pressure side of the blade to the suction side through the gap between blades and casing, called impeller side clearance. As stressed by Ayad [5], this flow interacts with the secondary flow in the impeller increasing the three-dimensionality of the flow. Thus head and efficiency drops with an increasing impeller side clearance. This parameter is very important in the design and tuning of the machine. Several works, e.g. Aknouche [6], has been done to evaluate the effect of impeller side clearance on centrifugal pump efficiency and head. Aknouche report that two main reasons may affect impeller performance: side clearance flow partially mixes with the core flow when it enters the passage and the resulting lower streamwise velocity causes the flow blockage and the decreased pressure rise.

In this paper a comparative analysis using different CFD software has been carried out. The results are compared one another and with the experimental data for what concerns the processing of water. The results of steady sate calculations with frozen rotor model for the rotor – stator interaction are reported. For the non Newtonian fluid, the comparison is made only between the numerical results.

Nomenclature

k	consistency index	BEP	Best Efficiency Point
n	viscosity index		
H	head		

Greek letters

μ	molecular viscosity
μ_a	apparent viscosity
$\dot{\gamma}$	shear rate

2. Computational set up

2.1 Fluids

For Newtonian fluids the stress response to a simple shear rate $\dot{\gamma}$ is linear. The linearity constant μ depends on the thermophysical conditions of the fluid (his temperature and pressure). In this article water has been chosen as representative of Newtonian fluids.

The non-Newtonian fluid model does not follow this rule: the stress response to a shear rate is non linear. There are many classes of fluids that do not follow the linear behavior, so different that a universal model suitable for all situations is not available (see Steffe [7]). A well-known formulation for a time-independent non-Newtonian fluid is the Power Law fluid model that is defined in Eqn. (1).

$$\mu_a = k\,\dot{\gamma}^{n-1} \quad (1)$$

The fluid used for the simulation is a low concentrated tomato paste (the one can usually be found in the first stage out of three in a common concentration process). The rheological behavior of the fluid can be modeled using Eqn. (1). This fluid fits the power law if the consistency index is equal to 100 Pa s and the viscosity index is 0.326. The density of the paste is considered constant and equal to 1100 kg/m^3. The power Law model presents a singularity for zero shear strain rate and the apparent viscosity tends to zero for high value of the shear strain rate (the fluid considered is a shear thinning). It is common practice to bound the values the viscosity can assume, though. The following rule is usually implemented:

$$\mu_a = \begin{cases} k\dot{\gamma}_{\min}^{n-1} & if \quad \dot{\gamma} < \dot{\gamma}_{\min} \\ k\dot{\gamma}^{n-1} & if \quad \dot{\gamma}_{\min} < \dot{\gamma} < \dot{\gamma}_{\max} \\ k\dot{\gamma}_{\max}^{n-1} & if \quad \gamma > \dot{\gamma}_{\max} \end{cases} \quad (2)$$

In this case the limits are $\dot{\gamma}_{min} = 10^{-4} s^{-1}$ and $\dot{\gamma}_{max} = 10^7 s^{-1}$.

2.2 Mesh

The features of the computational grids for each software are listed below.
The computational grid adopted for the Ansys-CFX simulations is composed of tetrahedrons. It was realized with Ansys-Meshing and is composed of 10.5 million elements. The algorithm for the body fitting is the patch conforming method. The growth of the prismatic layers has been realized before the generation of the volume mesh in order to guarantee its uniformity.
For the simulations carried out with OpenFOAM, the mesh has been generated with the aid of SALOME®, an open source cad with several meshing algorithm implemented as well. According to the chosen method, the realization of the 1D, 2D and 3D mesh is unique and is based on the NETGEN algorithm (see [8]). The so-obtained tetrahedral mesh has been converted in OpenFOAM readable format, and the prismatic layers close to the walls have been added with the meshing tool *snappyHexMesh.* The overall mesh is composed by 8.3 million elements. This operation is necessary for the application of the wall functions for the turbulent quantities.
For what concerns the Pumplinx simulations, the mesh is a 7.5 million elements mesh. This grid is composed of hexahedral and it is a trimmed mesh. The generation of the mesh has been done using the automatic mesh generator implemented in the software. It uses a proprietary geometry Conformal Adaptative Binary-tree (CAB) algorithm.

2.3 Numerical set-up

The simulations have been carried out with different codes which solve the 3D Reynolds averaged form of the Navier–Stokes equations by using an element-based finite volume method. The multiple reference of frame approach was used for the steady simulations; the model for the rotor/stator interface between impeller and volute is a frozen rotor. The rotation speed of the impeller is 2900 rpm.
Since pressure and temperature inside the machine are limited in a relatively narrow range, for numerical modeling purposes, the fluids are considered incompressible and isothermal. Particularly the temperature variations were neglected during this study. As boundary condition at the inlet, a constant velocity value with normal direction and a turbulent intensity equal to 5 % was imposed.
The Ansys-CFX simulations were carried out by the version 14.5. An algebraic multi-grid method based on the additive correction multigrid strategy was used. Both a first and a second order advection scheme was adopted to calculate the advection terms, while a first order scheme was used for the turbulent quantities.
The OpenFOAM results were obtained with the v3.0+. The numerical schemes chosen for the advection terms discretisation of the momentum equation belongs to the upwinded ones. The solver used for the resolution of the algebraic system of discretised equations is a Geometric Algebraic MultiGrid.
The Pumplinx (v 4.0.2) results were obtained with a Preconditioned Coniugate Gradient solver. The simulations in this case have been run with the second order numerical scheme only.

3. Results

3.1 Newtonian Fluid

In Figure 1 the results comparison in term of head are reported. One can see that, for what concerns water, the results obtained with OpenFoam – 1st order provide the highest head. Switching to the second order give results which are similar, except for very low flow rates.
In this part of the graph, the trend is different: the head calculated with a second order scheme decreases while the first is roughly constant. This difference is clear also in the case of the simulations run with ANSYS-CFX. In this case the performance are slightly lower than the previous case.
The simulations run with Pumplinx give the lower head. Also in this case the head provided at very low flow rates tend to decrease.

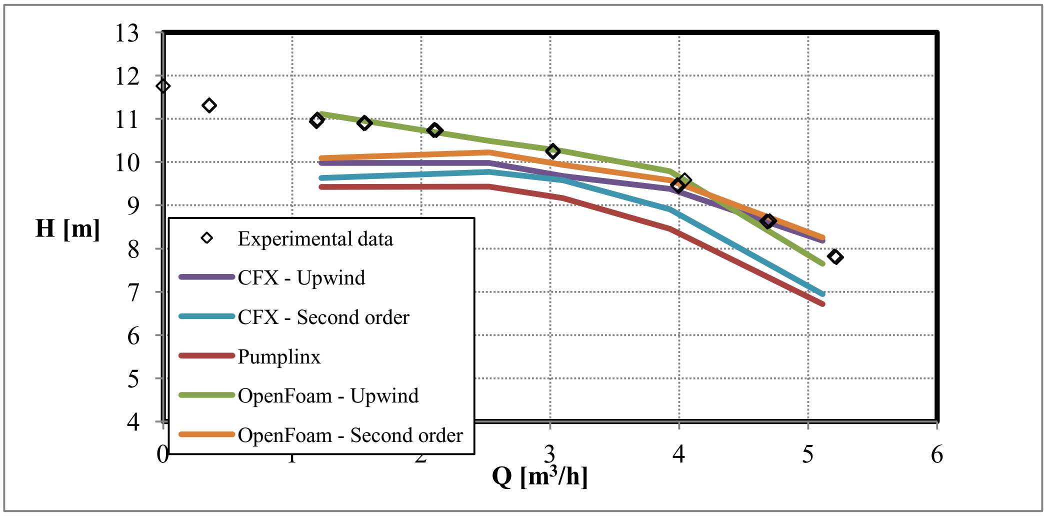

Figure 1 Newtonian Performance comparison - Head

3.2 Non-Newtonian Fluid

The non-Newtonian fluid-pumping performance are evaluated for a lower flow rate. This is in agreement with the Pullum correction theory: the BEP is shifted towards lower flow rates. The results of the simulation for the non-Newtonian fluid, are reported in Figure 3.
The simulations reported were all evaluated applying a second order upwinded numerical scheme to the advection term.
As one can easily see, the absolute value of the head provided in this case is roughly a third of the case of water. Accordingly the efficiency drops, passing from 65-70% to 10-15%. It is well clear that, among the used codes, Pumplinx predict the higher head. This result is exactly the opposite to the one obtained for the Newtonian fluid (See Figure 1). In this case the experimental data are not available, thus the comparison have been made only among the numerical results.
The results obtained with the open-source software show the lower head among the tested ones. To easily visualize the differences among the software, a mid-height contour of the pressure is reported in Figure 3.

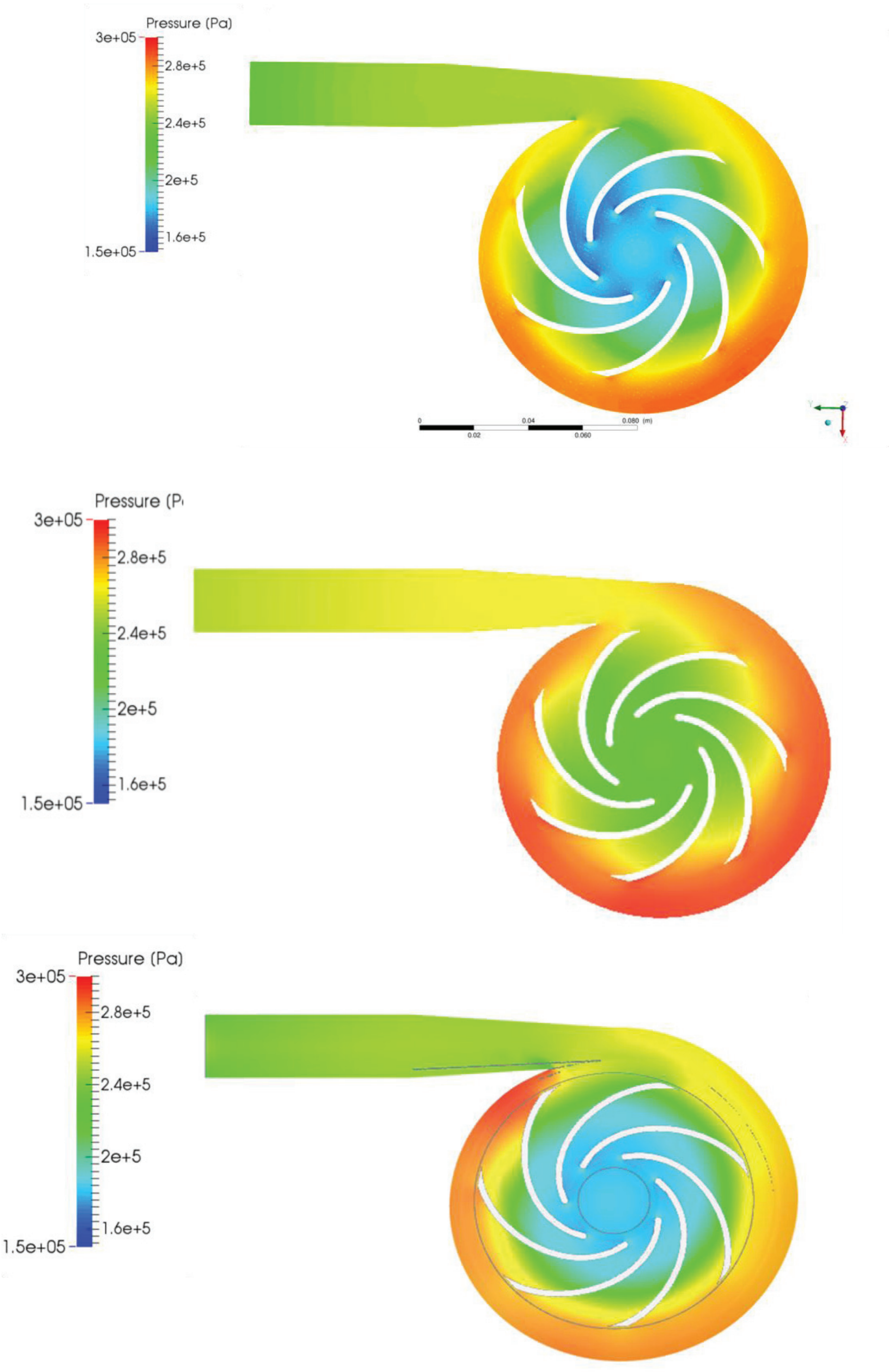

Figure 2 Mid-height pressure contour in case of tomato paste processing

4. Conclusion

Three software both proprietary and open-source have been tested and the results have been compared one another for a pump handling a Newtonian and a non-Newtonian fluid. In the first case the experimental results were available as well. The particular type of impeller of this machine make it suitable for the handling of non-Newtonian fluid (especially slurry). From this standpoint, this machine was numerically tested in order to point out the performance. The simulations have been made considering the pumping of a low concentrated tomato paste. The validation of the results obtained from the three software has been made on the experimental performance curve of water. The most accurate software in case of Newtonian fluid is OpenFOAM, using a first order upwind scheme. The experimental performance curve of the pumps differs from the numerical one at the most of the 5%. Furthermore the increase in the head provided at extremely low flow rates is reproduced.

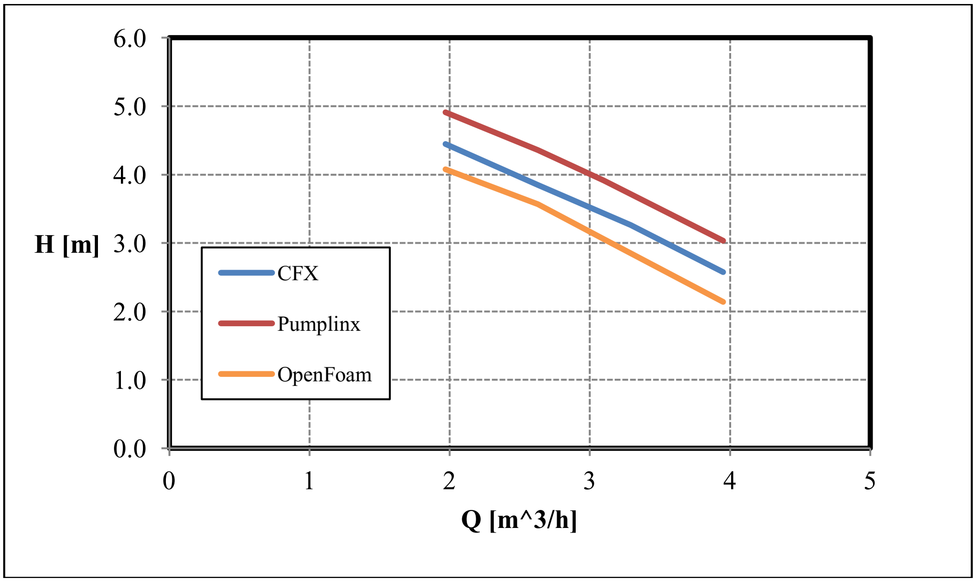

Figure 3 Non-Newtonian performance comparison - Head

The performances of the three software when the tomato paste is considered is shown in Figure 2. The overall performance of the machine decrease of roughly a third of the Newtonian fluid. All the software predict this lower head, with a maximum difference of 0.9 m (30%).

For what concerns the contours reported in Figure 2, it can be easily see that OpenFOAM predict the higher pressure at the impeller inlet, providing the smaller head among the tested software. The pattern of pressure is globally very similar in all the cases: the results are in good agreement one another. Unfortunately the experimental results are not available for this case.
Based on the results obtained in the case of Newtonian fluid, the OpenFOAM results are considered the most reliable.

References

[1] Li WG, Hu ZM. An experimental study on performance of centrifugal oil pump. *J Fluids Machines* 1997;25(2):3–7.

[2] Li WG. Numerical study on behavior of a centrifugal pump when deliveringviscous oils – Part 1: Performance. *Int J Turbo Jet Engines* 2008;25(2):61–79.

[3] M.H. Shojaeefard, M. Tahani, M.B. Ehghaghi, M.A. Fallahian, M. Beglari, Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid, *J Computers & Fluids*, Volume 60, 15 May 2012, 61-70.

[4]Buratto, C., et al. CFD study on special duty centrifugal pumps operating with viscous and non-Newtonian fluids. *Proceedings of 11 thEuropean Conference on Turbomachinery Fluid dynamics & Thermodynamics ETC11*, 2015.

[5] A. Farid Ayad, Hamed M. Abdalla, A. Abou El-Azm Aly, Effect of semi-open impeller side clearance on the centrifugal pump performance using CFD, *J Aerospace Science and Technology*, Volume 47, December 2015, 247-255.

[6] Aknouche, Sebastien. Impact of tip clearance flow on centrifugal pump impeller performance. *Diss. Massachusetts Institute of Technology*, 2003.

[7] Steffe, JF. Rheological methods in food process engineering. *Freeman press*, 1996.

[8] Schöberl, Joachim. "NETGEN An advancing front 2D/3D-mesh generator based on abstract rules." *J Computing and visualization in science* 1.1 (1997): 41-52.