Proceedings of the ASME 2009 Fluids Engineering Division Summer Meeting
FEDSM2009
August 2-6, 2009, Vail, Colorado USA

FEDSM2009-78571

MEANS FOR IMPROVING PUMP EFFICIENCY

R.Karvinen
Tampere
University of Technology
Tampere, Finland
reijo.karvinen@tut.fi

M. Koivikko
Sulzer Pumps Finland Oy
Kotka,Finland

S.Virtanen
Sulzer Pumps Finland Oy
Kotka,Finland

ABSTRACT

The design of pumps has been mainly based on results obtained from experiments. Because there has been no detailed information concerning very complicated flow fields inside pumps, only by accident have some new ideas been found. At present, numerical modeling of the flow offers an opportunity to understand basic phenomena which make it possible to find new connections between hydraulic and geometric parameters. With the help of numerical modeling in a long-term project, new design instructions have been successfully developed. It seems that some existing ideas on how different factors affect pump performance should be re-evaluated. An important finding is that the effect of specific speed on efficiency is quite small. Also the efficiencies of existing pumps can be clearly improved with proper design.

INTRODUCTION

Efficient energy use is of vital importance everywhere at the moment, and will surely be even more so in the near future. Hydraulic machines, i.e. pumps, fans, compressors, refiners, etc., consume a significant portion of all electricity produced. The problem is not only the price of energy but also its availability. Thus, it is important that all existing efforts be adopted to improve the energy efficiency of hydraulic machines. Traditionally the design of hydraulic machines is based mainly on experiments and on experience from actual practice due to the fact that other tools have not been available. Design has been in general essentially empirical and relies heavily upon experimental data.

Many books have been published past decades in which the best design methods on the basis of the knowledge of that time are presented [1,2,3]. This trend has continued up to our own day [4 ,5,6]. The latest achievements, and also problems, have been presented in conferences. In Europe the Pumpentagung organized every fourth year arranged (in 2008 the name was International Rotating Equipment Conference) has been an excellent opportunity to follow the development in pumps. Special volumes of high- quality journals have also focused on pumps [7].

As mentioned above, design instructions have been based very much on practical experiences obtained by performing and analyzing tests, because it has been very difficult to get detailed information concerning complicated fluid behavior in pumps. At present CFD provides a powerful opportunity to study the influence of different parameters on pump performance. The first attempts were made with it already long ago, but computational resources have restricted its use as a design tool. The utilization of CFD in pump design has been a long-term task and now it has been successfully transferred from academic research institutes to the industrial environment [8,9,10].

The paper shows the approaches of the authors which have led to the successful development of very high- efficiency semi-open impeller pumps. The aim is to give an idea of how CFD can be utilized in the design process. It must be noted that CFD is not the only remedy; it only helps in understanding some features and can give the best combination of different parameters. It has helped in understanding basic fluid dynamics inside pumps so that the geometry could be changed in such a way that losses are small. In addition to computational modeling, also experimental verification is needed beforethe performance of a pump can be guaranteed.

DESIGN WITH TRADITIONAL METHODS

In order to fix the main dimensions of a pump, experience from actual practice on how the type of pump depends on the specific speed

$$n_q = nQ^{1/2} / H^{3/4},\ [n]=1/\text{min},[Q] = \text{m}^3/\text{s},[H]=\text{m} \qquad (1)$$

is usually utilized. In the equation above, Q is the flow rate, H the head and n the rotational speed. Engineers in industry prefer to use the dimensional specific speed above instead of a non-dimensional definition in which the gravitational acceleration is included in the denominator. In addition to the specific speed, also the suction specific speed can be fixed, which is analogous to the definition in Eq. 1, except that head H is replaced by $NPSH$. In order to continue the design, the values of hydraulic η_h and of volumetric η_v as well as disk friction efficiencies must be assumed. There exist in the literature instructions on how to fix the impeller eye diameter D_{1s}, the hub diameter at the impeller eye D_{1h}, the blade inlet angle β_1, the number of blades z, and the meridian velocity c_{1m} when the specific speed is fixed. The nomenclature is shown in the Appendix.

Even more important than the situation at the blade inlet, which controls cavitation, is the flow field at the blade outlet, because this determines the head produced by the impeller. When a one-dimensional approach is used, the maximum theoretically attainable head of an impeller is

$$H_{th} = \frac{u_2 c_{2u}^* - u_1 c_{0u}}{g} \tag{2}$$

where c_{2u}^* is the tangential velocity of the flow at the impeller outlet and c_{0u} that at the channel inlet. If there is no rotation of inflow, c_{0u} is equal to zero. Assuming that the direction of the flow and the blades at the impeller outlet are the same, an ideal tangential velocity can be written as

$$c_{2u}^* = u_2 - \frac{c_{2m}}{\tan\beta_2^*} \tag{3}$$

in which

$$c_{2m} = \frac{Q}{\eta_v B_2 \pi D_2} \tag{4}$$

The next essential step is to evaluate the performance of a real pump compared to an ideal one based on the Euler equation above. This is made using a slip factor (the European definition of the slip factor, in which the reference velocity is the tangential velocity of an ideal pump, is used)

$$k = \frac{c_{2u}}{c_{2u}^*} \tag{5}$$

There exist many different formulae of slip factors in the literature, the results of which deviate from each other [11,12].

In addition to the specific speed, the pressure coefficient

$$\psi_{th} = \frac{2gH_{th}}{u_2^2} \tag{6}$$

and the flow coefficient

$$\varphi = \frac{c_{2m}}{u_2} \tag{7}$$

are also used in presenting pump performance. The values of these parameters describe the internal phenomena inside the pump better than the specific speed. In the literature there are recommendations concerning the connection between hydraulic factors and geometrical parameters. These instructions are mainly based on tests of a variety of models and nothing really new has been discovered on that basis.

TYPES OF NUMERICAL MODELING APPROACHES

The flow in a centrifugal pump is turbulent, highly three-dimensional, and spatially non-uniform. In addition, the flow is transient between an impeller and a volute. Different types of approaches to solve the problem can be chosen: steady-state, quasi-steady, or transient. The steady-state method is the fastest one and the computational resources required are not great. The next step in complexity is reached in a quasi-steady approach, in which the flow field is solved for different positions of the impeller in relation to the volute and mass- averaged results are obtained from calculated values in different positions. The transient treatment is the most powerful method, but it requires plenty of computational capacity. At present, this is not a problem, and the method is an excellent tool for research and design purposes.

In the steady-state method, an impeller and volute are solved separately so that they have no interaction with each other. The computational grids for stationary calculation of impeller and volute are shown in Fig.1. Using periodic boundary conditions only one passage of the impeller needs to be modeled. In Fig.1 the periodic surfaces of an impeller flow passage are shown. They are marked in red and blue. The inflow boundary condition used for the impeller is usually a fixed mass flow rate or velocity, and the outflow boundary condition of both the impeller and volute is a constant or averaged static pressure. The averaged static pressure is more appropriate especially at part load, because it allows having strong velocity gradients through the domain outlet. The grid surface that connects the impeller to the outflow block is called the stage outflow and is marked in yellow in Fig.1. The mass-averaged tangential and radial velocities from the simulation of the impeller at the stage outflow surface are used as inflow boundary in the simulation of the volute. The connecting grid surface between the inflow block and impeller is called the stage inflow and is marked in green.

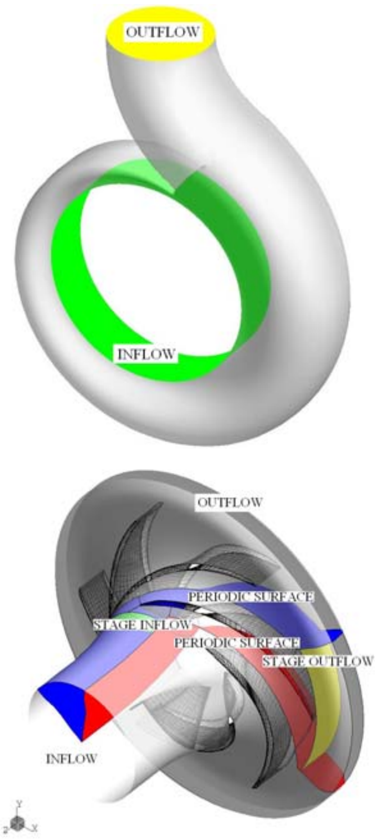


Fig.1 Inflow and outflow boundaries of impeller and volute.

In a quasi-stationary calculation the grid of a complete impeller is rotated in relation to the grid of the volute between simulations and a stationary solution is calculated at the different grid positions. The relative position of impeller and volute does not change during the simulation. The grids of the impeller and volute are combined using a frozen rotor method. The connection between rotating and non-rotating grids is made by adjusting the absolute velocities at the impeller outlet and volute inlet. All unsteady effects are neglected. The number of grid cells is approximately 10^6. The processing of results is performed in a similar way as in the case of a steady-state method.

The same grid as used in a quasi-stationary simulation of combined impeller and volute can be applied to the transient simulation. The impeller is connected to the volute using the rotor/stator grid interface at the stage outflow of the impeller and stage inflow of the volute. The grid of the impeller is rotated relative to the volute between every time step. The difference from a stationary solution is that the unsteady term $\partial/\partial t$ of Reynolds averaged Navier-Stokes equations is not zero.

RESULTS

The $k-\varepsilon$ turbulence model is used in the methods above. The calculations of the present paper are made with the commercial CFX-TASCflow code [13]. Determination of pump characteristics and efficiency from numerically obtained pressure and velocity fields can be found in the literature [10].

The results of different numerical methods were compared with each other, and also with experimental data. Fig. 2 gives an idea of the validity of numerical modeling. The results are from a large headbox feed pump of a paper machine. It can be seen that at least the transient method gives good results when compared with measured ones. When comparing the results of different methods, it was noted that in the neighborhood of the best efficiency point all methods gave almost the same result for the head. The efficiencies of the steady-state method are slightly greater than those of other methods.

Results of different slip factor formulae in the literature expressed by Eq. 5 were compared with numerically obtained values and the difference in results between different methods was considerable. On the basis of the modeling of many pumps a new and better slip factor was developed [10]

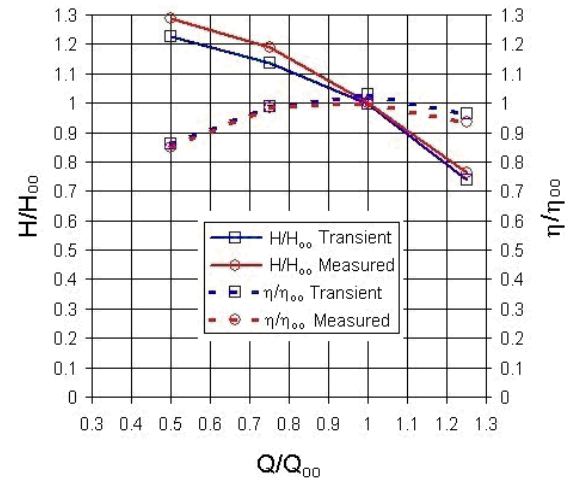


Fig. 2. Comparison of measured and modeled results

$$\frac{c_{2u}}{c_{2u}^*} = \frac{0.0137\ z}{\varphi^{0.5}(c_{2u}^*/u_2)(c_{0m}/u_{1m})^{0.1}} + 0.347$$

$$\begin{cases} 4 \le z \le 6 \\ 0.017 \le \varphi \le 0.16 \\ 0.65 \le c_{2u}^*/u_2 \le 0.95 \\ 0.26 \le c_{0m}/u_{1m} \le 0.42 \end{cases} \qquad (8)$$

In the equation above, $\varphi = c_{2m}/u_2$ and

$$c_{0m} = Q/(\eta_v \pi ((D_s/2)^2 - (D_h/2)^2)) \qquad (9)$$

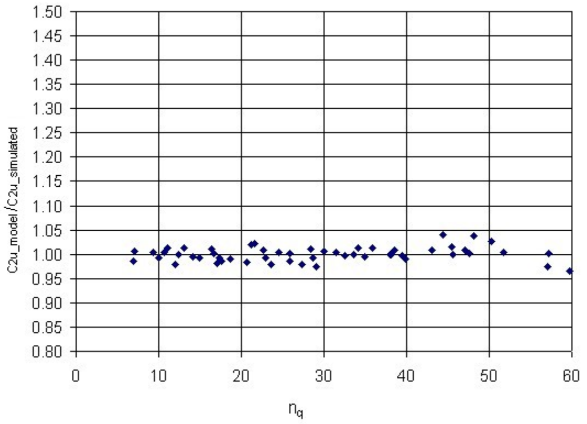


Fig. 3. Tangential velocity at impeller outlet (Eq.8) divided by numerically obtained value.

Fig. 3 shows a comparison between Eq. 8 and numerically obtained results as a function of the specific speed.

The validity of Eq.(8) was also tested experimentally using the very carefully machined process pumps with semi-open impellers in Fig. 4, where one half of a volute is shown. Altogether 9 very carefully machined aluminum impellers were measured. The evaluation of tangential velocity is very difficult in measurements because it would require the velocity distribution at channel outlets and also the distribution of different efficiencies. Therefore, it is more convenient to define lower and upper limits of the tangential velocity. The lower limit is obtained by assuming that none of the losses are head losses, and the upper limit by assuming that all losses are head losses.

The method on the basis of which it was attempted to understand the performance of pumps has been briefly described above. A preliminary rough design was based on Eq. (8) and the geometrical details were fixed after that, using numerical modelling. A special library indicating the best combinations of geometrical parameters was created by grouping geometrical parameters into dimensional variables in Eq. (8). In this way the design procedure for a new pump can be shortened, because there is no need to model every new pump numerically.

Fig.(5) gives an idea of the efficiencies of some semi-open impeller pumps which have been designed using the procedure above, and also manufactured and measured [14]. A very interesting observation can be noted concerning pumps with high specific speed: the efficiency is considerably higher than that which should be attainable according to a statistical evaluation of data collected from questionnaires to European pump manufacturers concerning pumps with closed impellers [15].

The same fact is also true for pumps with small specific speed according to preliminary results of the authors. Efficiencies of closed impeller pumps are slightly higher than for those with semi-open impellers, but even with semi-open impellers the values in Fig. 5 can be exceeded. When the results of modeling and also experimental data of high specific speed pumps were analysed the suspicion arose that the effects of different phenomena on pump performance found in the literature should be re-evaluated. It seems that design instructions can be modified if high efficiency is the goal.

The design, testing, and manufacturing of a new pump type is a long-term project. The research leading to the pumps in Fig.5 was started many years ago. Finally, the main factors concerning how pump performance can be made more energy-efficient than that of the pumps on the market were discovered. A reliable method for designing large specific-speed pumps for $n_q \geq 40$ is now available. Same basic laws of fluid dynamics are also valid for small specific-speed pumps. The authors have designed new pumps in the range of n_q =10-40, the efficiencies of which are significantly better than those which can be achieved according to the literature, as shown in Fig.5. Because the experimental verification is not yet ready absolute values are not shown in the figure.

.

Fig. 4.Prototype pump used in measurements.

MAIN FACTORS AFFECTING EFFICIENCY

The overall efficiency is traditionally divided into different parts, for instance the hydraulic, volumetric, disc friction and mechanical efficiency. It is a common believe that the maximum efficiency is achieved by big pumps when the specific speed is about 60. The hydraulic loss distribution of an impeller and a volute is traditionally considered to be strongly dependent on the specific speed. This result is achieved on the basis of a historical background by putting the emphasis on such

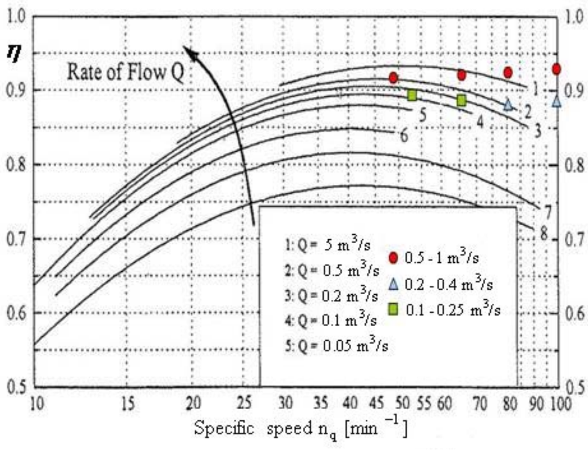


Fig. 5. Measured efficiencies of semi-open impeller pumps and comparison with maximum attainable ones in literature (solid lines) [15].

hydraulic properties which give a strong specific speed dependency. However, the hydraulic losses as a function of specific speed can be made different from those according to the statistics of existing pumps by rearranging the priority of hydraulic properties. In addition to the hydraulic design, also manufacturing is important. The volumetric, disc friction and mechanical losses are all strongly dependent on the quality of manufacturing. It is also important that the interaction of different components, i.e. impeller and volute, be correct. The overall efficiency can be destroyed if a very high-efficiency impeller is mounted with an unsuitable volute.

CONCLUSIONS

The need to improve the energy efficiency of pumping requires, in addition to the proper design of a piping system, that efficiencies of pumps be high. Numerical modeling of very complex flow in a pump has given a new possibility to study the flow field in detail. Traditionally pump design has been based mainly on experiences from actual practice, and something really new has been found only by accident.

When numerical modeling is used as a tool, the transient method is most accurate and can present the pump performance everywhere, also outside the best efficiency point. The steady-state model gives the pump characteristics correctly in the design point, but it overestimates the efficiency. The two-equation model of turbulence is enough.

Numerical modeling has helped to understand complex flow phenomena inside pumps. With the modeling, a new slip factor correlation, which is superior to earlier ones, has been developed. Also a library connecting the best combinations of hydraulic and geometrical parameters has been generated, which gives the possibility to rapidly optimize pump geometry. On that basis, new types of high specific speed pumps have been developed, the efficiency of which is clearly better than that of earlier pumps. In the range of small specific-speed pumps also new pumps, the efficiency of which is higher than attainable according to the literature, has been found. The experimental verification is being worked on. This means that the general belief that the specific speed has a strong influence on efficiency must be re-evaluated

REFERENCES

1. Pfleiderer, C., Die Kreiselpumpen für Flüssigkeiten und Gase, Springer-Verlag, 1955.

2. Lazarkiewicz,S., Troskolanski,A.,T., Impeller Pumps, Pergamon Press, 1965.

3 Schulz, H., Die Pumpen, Springer-Verlag Berlin Heidelberg New York, 1977.

4. Neumann, B.,The Interaction between Geometry and Performance of a Centrifugal Pump, Mechanical Engineering Publications Limited, London, 1991.

5. Japikse, D., Marscher, W., Furst, R., Centrifugal Pump Design and Performance, Thomson-Shore, Inc., USA, 1997.

6. Gülich, J.F., Kreiselpumpen, Ein Handbuch für Entwicklung, Anlagenplanung und Betrieb, Springer, 1999.

7. ASME Journal of Fluids Engineering,Vol.121, June,1999.

8.Treutz, G., Numerische Simulation der instationären Strömung in einer Kreiselpumpe, Dissertation Technical University Darmstadt, 2002.

9.Gugau, M.,Ein Beitrag zur Validierung der numerischen Berechnung von Kreiselpumpen, Dissertation Technical University Darmstadt, 2004.

10. Koivikko, M., Modelling the Effect of Geometric Parameters on the Performance and Efficiency of Centrifugal Pump Impellers, PhD Thesis, Tampere University of Technology, 2006.

11. von Backström, T.W., 2006, A Unified Correlation for Slip Factor in Centrifugal Impellers. Trans.ASME:J. Tubomachinery128, pp.1-10.

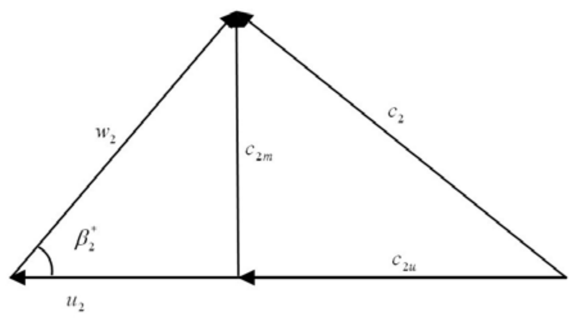
12. Huang, S., Liu,Z., Lu, Y.,Yan. Y.,Lian.X., Validations of some slip factor models for mixed-flow impellers, ASME Fluids Engineering Conference, August10- 14, 2008, Jacksonville, Florida USA.

13.AEA, CFX-TASCflow Theory Documentation Version 2.12, AEA Technology Engineering Software Ltd., Waterloo, Ontario, Canada, 2002.

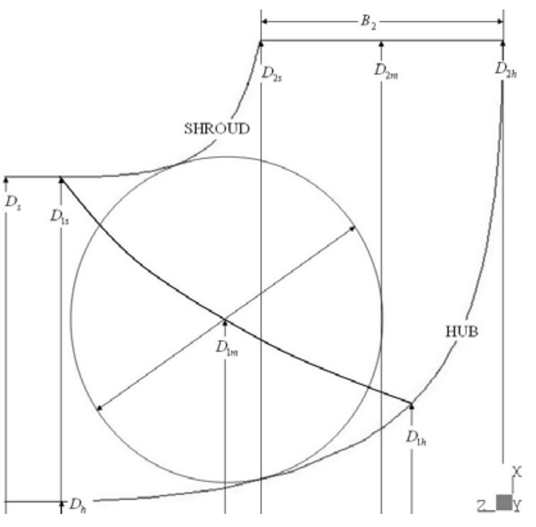
14. Karvinen, R., Koivikko,M., Virtanen,S., Design Procedure for High-Efficiency Pumps, International Rotating Equipment Conference, Pump Users International Forum, Düsseldorf October 28-29,2008.pp.228-237.

15. Stoffel, B., Lauer, J., 1998, Attainable Efficiencies of Volute Casing Pumps, European Association of Pump Manufacturers Working Group 9 “Pump”.

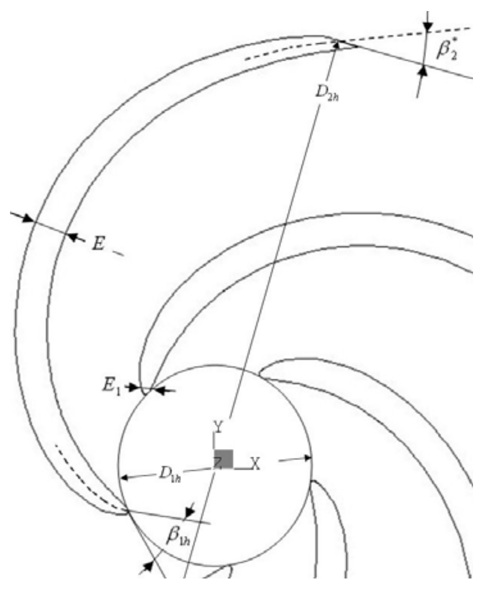
APPENDIX: Impeller notations



Velocity triangle at impeller outlet



Meridional view of centrifugal impeller



Ground view of blade profile at hub

