EPJ Web of Conferences **269**, 01035 (2022) https://doi.org/10.1051/epjconf/202226901035
EFM 2019

Inducer with Variable Pitch

Patrik MARCALÍK[1], Lukáš ZAVADIL[2], Milada KOZUBKOVÁ[1] and Jana JABLONSKÁ[1]*

[1]Vysoká škola báňská – Technická univerzita Ostrava, 17. listopadu 2172/15, 708 00 Ostrava-Poruba, Czech Republic
[2]SIGMA Výzkumný a vývojový ústav, s.r.o., ul. Jana Sigmunda č. 313, Lutín, Czech Republic

Abstract. This paper deals with CFD incorporation in a process of designing inducer with variable pitch. The introductory part is devoted to the theory of centrifugal pumps, cavitation and inducer itself followed by a description of the mathematical models of turbulence, multiphase fluid and cavitation. The main part of this paper consists of CFD evaluation, first, of inducer itself, later in application with centrifugal pump with regards to cavitation properties with and without the aplication of inducer, for interest, the comparison with different type of inducer, with constatnt pitch, is also included. At the end of this paper the advantages and some disadvantages are given for CFD based design of the hydrodynamic machines.

1 Introduction

At present, CFD software are inseparable tool for most fluid engineers, especially for those that deal with design of various hydrodynamic machines, such as pumps, for example [1], as will become quite evident by the end of this paper.
The importance of correct estimation of operation parameters of the resulting proposal cannot be stressed enough, which is especially true at instances where cavitation occurs. As for cavitation occurring during flow of water, which is what this paper deals with, it is phenomenon well documented in literature, both theoretically [4 to 6] and in connection to technical practice [7 to 10].

CFD methods can be used for the optimization of the machine geometry itself [11], whenever possible, or, as if shown in this paper, it can be used to design other devices which can solve the cavitation problem.

2 Centrifugal Pumps

Centrifugal pumps are a very important tool of the technical world, be it specifically in technical practice, or in everyday life, which illustrates the worldwide centrifugal market with a volume of tens of billions of dollars a year [2] with applications across all technical sectors.

They are designed with flow rates ranging from values of thousandths to several tens of $m^3 \cdot s^{-1}$ for conveying heights up to approximately 5000 m at speeds ranging from several hundreds to several tens of thousands rpm [2].

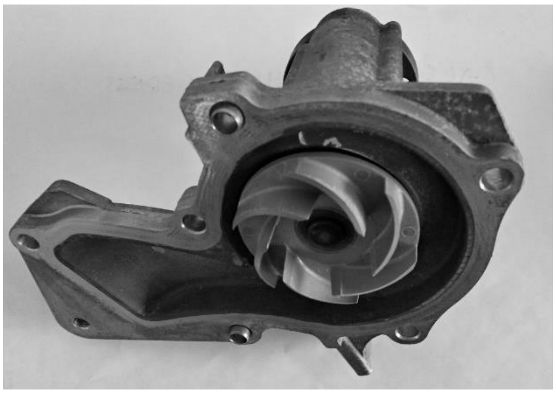

Fig. 1. Radial Impeller [3]

Centrifugal pumps are hydrodynamic machines used for transportation of liquids by raising a volume flow to a specified pressure level. The impeller transfers the mechanical energy into kinetic energy necessary to transport the fluid and accelerates it. This causes the static pressure to increase. The fluid exiting the impeller is decelerated in the volute and the following diffuser in order to utilize the greatest possible part of the kinetic, which is transformed into hydraulic energy, for increasing the static pressure.

They come at variety of types according to the shape of the impeller. As such they can be divided into 3 general types: radial, which would be used later for CFD purposes, see Fig. 1, semi-axial and axial. By the number of impellers themselves we can further divide them into single/multi-stage pumps, see Fig. 2.

* Corresponding author: patrik.marcalik@vsb.cz

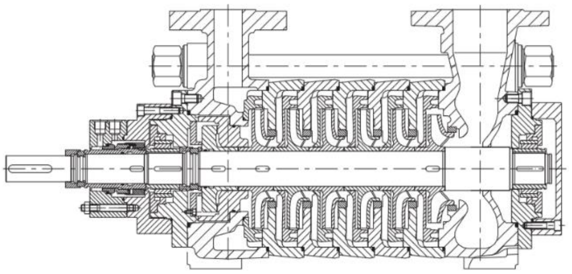

Fig. 2. Multistage segmental pump [2]

Another possible criterion is number of entries to the impeller: single/ double entry, see Fig. 3.

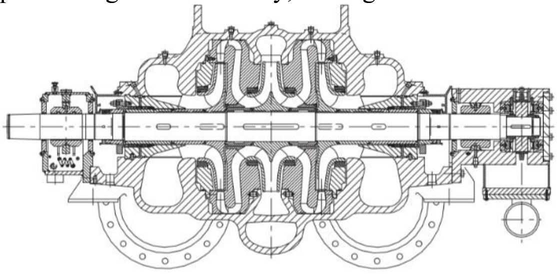

Fig. 3. Two-stage, double-entry pump [2]

For further information on centrifugal pumps and other possible divisions see [2].

3 Cavitation

In this section the problem of cavitation will be outlined. First, its physical nature will be discussed, and then solution via inducer will be presented.

3.1 Cavitation

By cavitation local multiphase flow is understood. Locally produced cavity filled with water vapor (gas phase) is transported from areas, where the static pressure drops below the saturation pressure due to the excessive velocity of the flowing liquid, downstream to regions, where the static pressure is again greater than the saturation pressure at which point cavity implodes, i.e. the vapour, that fills cavity, suddenly condenses back from the gas phase into the liquid phase, the surrounding liquid suddenly fills the newly formed space in the cavity. This phenomenon is accompanied by the existence of a significant pressure wave, possibly 100 MPa [2].

For hydrodynamic pumps typical cavitation forming area is found at the suction of the pump impeller where the liquid takes up high velocities around the wall of the suction side of the blade, cavitation cloud is then transported through the impeller canal until implosion occurs near the blade wall resulting in cavitation erosion, if left unnoticed resulting damage to the impeller might become fatal, such as in fig. 4.

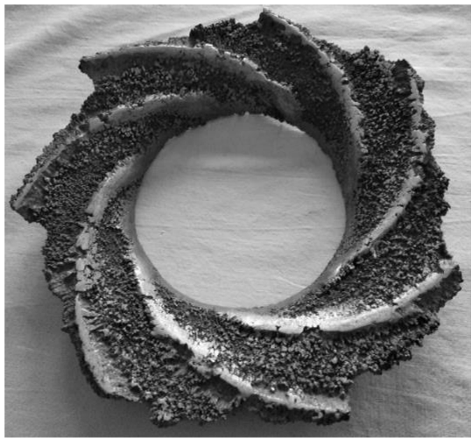

Fig. 4. Cavitation erosion [3]

Other detectable outcomes of cavitation are increased noise and vibration levels. Noise levels are particully worth mentioning as they may raise up to 90 dB and more depending on the severity of cavitation [11].

3.2 NPSH

As was mentioned earlier, the criterion for the occurrence and extend of cavitation is existence of pressure within the flow that drops below the saturated pressure. In technical applications however such pressure cannot be measured by any simple means. To that end simple quantifying magnitude was introduced – *NPSH* [m].

$$NPSH = \frac{\Delta y}{g} = \frac{p_{\mathrm{sHG}} - p_{\mathrm{s}}}{\rho g} + \frac{c_{\mathrm{sHG}}^2}{2g} \qquad (1)$$

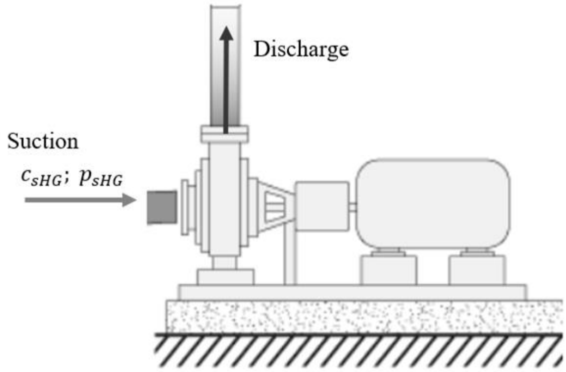

Fig. 5. Cavitation erosion [3]

The *NPSH* of the pump is the minimum ABSOLUTE pressure that must be present on the suction of the pump to avoid cavitation. In technical applications it is very useful to determine a specific *NPSH* that will imply the permitted level of cavitation. This *NPSH* is referred to as $NPSH_{\mathrm{R}}$. The index R denotes the selected criterion of the consequences of the cavitation process. As such this criterion comes with many different modifications based on the most interesting endpoint for each given application. In this paper we deal with $NPSH_3$ which is criterion of maximum allowable reduction of the pump head by 3%, which in industrial application settings is usually measurable.

4 Mathematical models of multiphase flow with cavitation

In this section we will deal with basic description of the mathematical models used in later CFD calculation, first basic flow models modified by turbulent flow will be discussed, then cavitation model will be introduced.

4.1 Turbulent flow model

The basic mathematical model consists of a spatial multiphase flow model, which uses equations expressing the basic physical laws of mass and momentum conservation defined for the mixture and equations of volume fractions of individual phases [3].

In centrifugal pump applications, turbulent flow always occurs, that is flow characterized by high Reynolds numbers. Because of that all the equations must be modified by statistical method of time averaging [3].

For the CFD calculation based on the experience when dealing with CFD evaluation of centrifugal pumps the k-ω SST transient turbulent model was selected from the many available models in today's commercial software. Because of the turbulent model all evaluations must be done with time averaged values.

4.2 Multiphase model

As was said earlier cavitation is treated as a multi-phase flow, assuming in our case a local two-phase flow of water (H_2O) as liquid phase and water vapor as compressible gas phase. Because of this the cavitation model must therefore contain equations for the transport of the gas phase and phase change between vapour and water [3].

4.3 Cavitation Model

The Zwart - Gerber - Belamri cavitation model was selected again based on previous experience. This model assumes a constant bubble radius. With this assumption the relationship between the total rate of phase change in the unit of volume and the bubble density number was obtained. Through this relationship and subsequent modification, the equations for growth and implosion of the vapor bubble were obtained [3].

5 Inducer

By the term inducer we understand axially positioned impeller upstream of the main impeller, see Fig. 6.

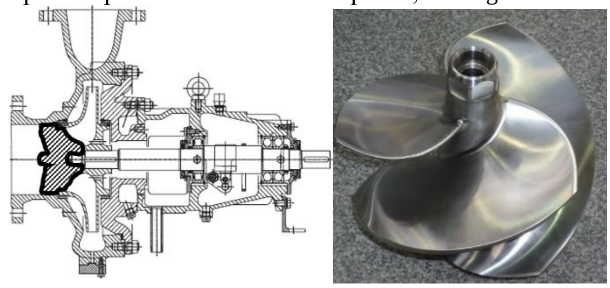

Fig. 6. Inducer (bold black line) placed in front of impeller in centrifugal pump [2] and photo of inducer

Its main function is to lower the required $NPSH_R$ up to half the value without its application by way of increasement of the static pressure in front of the pump impeller, thereby reducing the formation of vapor bubbles on the impeller blades, proportionally up to complete suppression. As a result, the pump can be operated at higher speeds, possibly with a lower suction pressure.

The profile of the blades can be designed with a constant pitch as a helical surface in an easier variant.

+ the main advantage lies in its advantageous simple manufacturing
− this approach has its limitations in achieving the necessary static pressure.

On the other hand, the design using variable pitch consist of blade angle increasement in its cylindrical cross sections in the flow direction from the inlet to the outlet in contrast to the first variant, where the blade angle over the entire length of the cross sections. This change essentially reverses the pros and cons with comparison to the first variant but is the only way to sufficiently increase static pressure before main impeller provided that reasonable dimensions are maintained.

6 Design methodology of inducer with variable pitch

In this section the methodology of inducer design process will be introduced.

6.1 Theoretical design methodology of inducer blade geometry

According to the dimensions of the pump, in the first step the basic geometrical dimensions and parameters such as inlet and outlet diameters, number of blades etc. were chosen, after which other parameters were calculated, based on [2], such as blade angle progress etc., for more information see [3], this comprises a conventional approach to the design of the inducer.

6.2 CFD evaluation and design refinement of inducer blade geometry

With thus obtained parameters [3], CFD software was approached, and 3D geometry was generated based on those parameters, after which fine enough computational mesh was generated considering the need for fine mesh for cavitation calculations (the need to capture the movement of individual bubbles).

At this point CFD analysis and subsequent evaluation itself could be realized at created computational domain, see Fig. 7, with boundary conditions (BC onward) according to the Table 1.

Table 1. BC for inducer alone CFD calculation.

Number of Phases	1 - water (H_2O)
Inlet BC	total pressure p_{total}
Outlet BC	mass flow Q_m

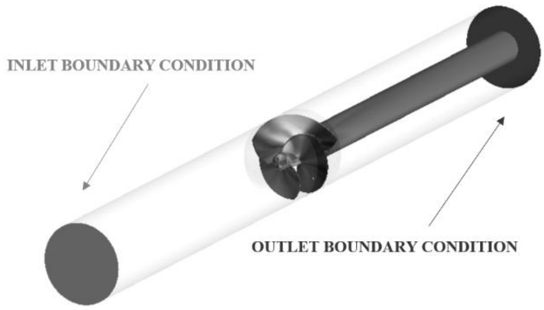

Fig. 7. Computational domain of inducer alone CFD calculation [3]

At this point the calculation was set without cavitation, as this was not yet important. Only simple single-phase, turbulent model was set with the required parameter of static head. The calculation was set as time-dependent with a time step dt = $1.11857 \cdot 10^{-4}$ s. To determine the time step, the multiplier of the inducer blades was used. After several changes of design, the optimal variant was found. The end result of a static height was over 10% better in comparison to the conventional approach [3]

7 CFD cavitation evaluations

In this section methodology of CFD cavitation evaluation will be introduced. First the computational domain will be shown with its settings after which evaluation of computed data will be presented.

7.1 Computational domain

Based on the previously obtained computational mesh of inducer and the pump itself [3] the whole computation domain could be created, see Fig. 8.

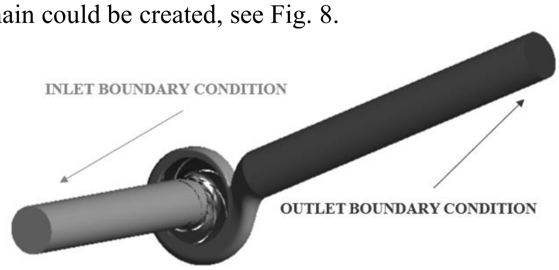

Fig. 8. Computational domain of inducer with pump [3]

Calculation was set as multiphase (two phases – liquid phase as water (H_2O) and gas phase as water vapor) with cavitation model enabled, see 3, with BC according to the Table.2.

Table 2. BC for inducer with pump CFD calculation.

Number of Phases	1 - water (H_2O), 2 - water vapor
Inlet BC	total pressure p_{total}
Outlet BC	mass flow Q_m

The calculation was made for the pump optimal mass flow and for 2 subsequent suboptimal and above optimal mass flows with relatively high value of static pressure on inlet BC to ensure non-occurrence of cavitation with gradual decreasement right up to the values where $NPSH_3$ was not met. The deciding factor was static head.

7.2 Evaluation of chosen CFD results

For the purpose of evaluation of pressure distribution and vapor volume fraction only one variant (optimal mass flow at Outlet BC with Inlet BC pressure just before arriving at the critical value for the $NPSH_3$) out of the many variants of calculation was chosen.

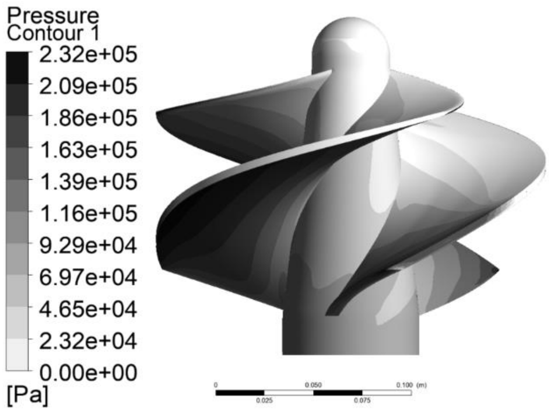

Fig. 9. Absolute static pressure at inducer domain [3]

As can be seen at Fig. 9, pressure distribution on the inducer blades clearly indicates presence of cavitation as the saturated pressure has been reached, which can be seen at Fig. 10, where the vapor volume fraction (white) of value 0.1 is shown.

Fig. 10. Vapor volume fraction of value 0.1[3]

Significant volume of vapor at the domain of inducer can be seen. In the domain of the main impeller the volume of the vapor is not as significant. This variant nears $NPSH_3$, so this is very good result which indicates that the inducer fulfils its function, which is more elaborated in next subchapter.

For interest, on the Fig. 11, comparison of the two different types of inducers is drawn:

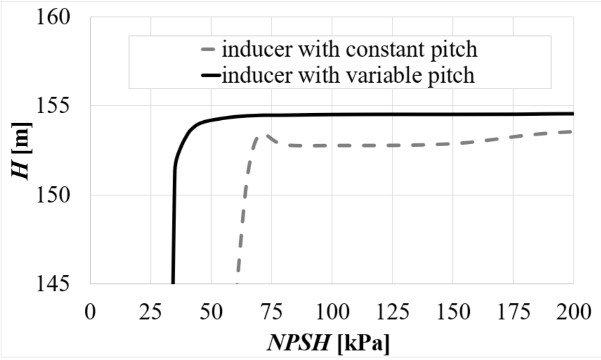

Fig. 11. Vapor volume fraction of value 0.1[3]

As can be seen from Fig. 11, inducer with variable pitch effectively doubles minimum suction pressure for this operation conditions under which the pump is still able to work properly, in contrast to inducer with constant pitch.

7.3 Evaluation of pump characteristics

On Fig. 12 a comparison of the pump static head is drawn with and without inducer, different types of inducers are also considered.

As can be seen, there are no discernible differences as the change in static pressure obtained by inducer is not as significant from this viewpoint, but inducer with variable pitch still shows best results.

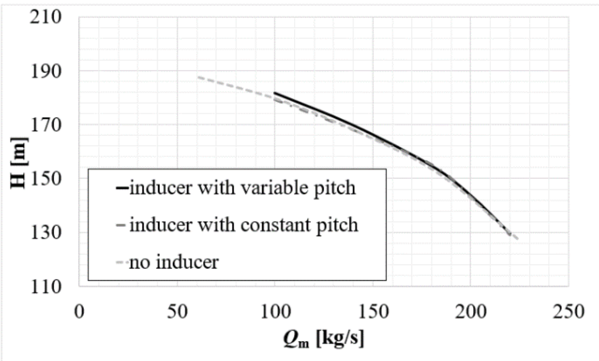

Fig. 12. *H-Q* characteristic [3]

The real difference is seen on the Fig. 13, where $NPSH_3$ is drawn. The minimal pressure at the suction is effectively doubled in comparison to pump working alone or with inducer with constant pitch across almost whole working area.

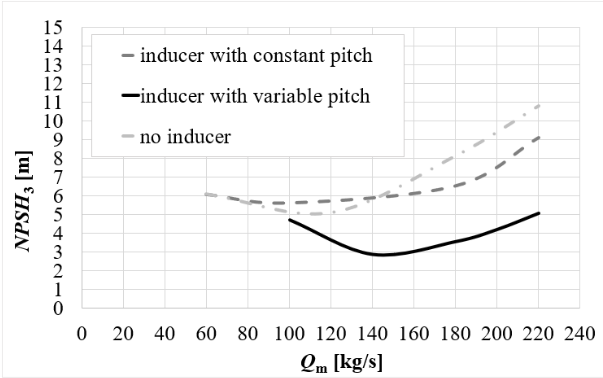

Fig. 13. $NPSH_3$ characteristic [3]

Inducer with variable pitch can raise static pressure so considerably in comparison to the inducer with constant pitch that its production costs are more than justifiable. The pump can now be operated at much lower suction pressures (higher speeds) given that good enough efficiency is still met.

8 Conclusion

This paper dealt with the incorporation of the CFD analysis into hydrodynamical design of the inducer with variable pitch.

As shown above and in more detail in [3], CFD analysis helped with the process of design significantly. If designed only by standard approaches the result would not be as effective as by incorporation of CFD methods.

From these certain conclusions can be drawn, such as:

- CFD analysis can be advantageously used in the geometric design of inducer (or any other hydrodynamic machines)
- To estimate the performance parameters with much better precision than by standard calculations alone, hence arriving at much better results by making the necessary modifications
- Integration of the common CAD formats with ANSYS
- Easily incorporable modifications to CFD geometric model based on the previously calculated results

There are also some disadvantages like:

- Need for CFD qualified worker
- Relatively long calculation time (dependent upon the geometrical complexity, mesh fineness, used mathematical models, PC hardware etc.)

CFD calculations have already proven to be inseparable part of the design of hydrodynamic machines on many levels of the process and as computer hardware progresses more and more complex calculations are possible.

Acknowledgement

This work has been supported by The Ministry of Education, Youth and Sports of the Czech Republic from the Specific Research Project SP2019/123 (Numerical Modeling of Dynamic Phenomena in Fluid Mechanics with Support of Experimental Research).

The authors thank to the project CZ.02.1.01/0.0/0.0/17_049/0008408 "Hydrodynamic Design of Pumps", provided by the Czech Ministry of Education, Youth and Sports

References

1. A. Ramadhan Al-Obaidi. Monitoring the Performance of Centrifugal Pump under Single-Phase and Cavitation Condition: A CFD Analysis of the Number of Impeller Blades. 2019. Journal of Applied Fluid Mechanics, Vol. 12, No. 2, pp. 445-459.. ISSN 1735-3572, EISSN 1735-3645. DOI: 10.29252/jafm.12.02.29303
2. Gülich, J. F. Centrifugal Pumps 3rd edition 2014. ISBN 978-3-642-40114-5
3. Marcalík, P. Design of Inducer with Variable Pitch: Diploma Thesis. Ostrava: VŠB –Technical University of Ostrava, Department of Hydrodynamics and Hydraulic Equipments, 2019, 65 p. Thesis head: Kozubková, M. (in Czech)
4. Csuka, Z., Olšiak, R., & Fuszko, Z. Research of cavitation at high shear stress. 2016. Strojnicky Casopis, 66(1), 7-16. DOI:10.1515/scjme-2016-0006
5. Brennen C.E. Cavitation and Bubble Dynamics. 270p. 2013 ISBN 13: 978-1-107-64476-2
6. Kyuichi Yasui Acoustic Cavitation and Bubble Dynamics. 2018. 131p. ISBN 13: 978-3-319-68237-2.
7. Knížat, B.; Olšiak, R.; Mlkvik, M. Observation of development of cavitation damage. 2014. In EPJ Web of Conferences (Vol. 67, p. 02053). EDP Sciences, DOI: https://doi.org/10.1051/epjconf/20146702053.
8. Luca d'Agostino, Maria Vittoria Salvetti (eds.) Cavitation Instabilities and Rotordynamic Effects in Turbopumps and Hydroturbines: Turbopump and Inducer Cavitation. 2017. Experiments and Design. 288p. ISBN 13: 978-3-319-49719-8
9. Poullikkas, Andreas. Two phase flow and cavitation in centrifugal pump: a theoretical and experimental investigation. 2019. figshare. Thesis.. https://hdl.handle.net/2134/11344
10. Noskievič, J. and collective. Cavitation in hydraulic machinery and equipment. 1990. Praha: SNTL. (in Czech)
11. Kratky, Tomas & Zavadil, Lukas & Doubrava, Vit. CFD and surrogates-based inducer optimization. 2016. International Journal of Fluid Machinery and Systems. 9. 213-221. 10.5293/IJFMS.2016.9.3.213.
12. Kozubková, M. Matematické modely kavitace a hydraulického rázu. 2009. Ostrava. 130 s. ISBN 978-80-248-2043-9. (in Czech)