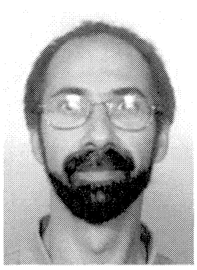
NUMERICAL PREDICTION OF CAVITATION IN PUMPS

by
Philippe Dupont
Head of the CFD Group
Sulzer Pumps Ltd.
Winterthur, Switzerland



Philippe Dupont is Head of the Computational Fluid Dynamics Group of the Sulzer's Pump Division, in Winterthur, Switzerland. Before joining Sulzer in 1998, he headed the Cavitation Research Group of the Laboratory of Hydraulic Machines at the Swiss Federal Institute of Technology, in Lausanne, Switzerland.

Dr. Dupont received his M.S. degree (Mechanical Engineering, 1984) from the Swiss Federal Institute of Technology of Lausanne (EPFL), and his Ph.D. degree (1991) from the same technical school.

ABSTRACT

With the increase in power per unit of volume for modern pumps, driven by manufacturing costs reduction, cavitation becomes more and more the main limiting factor in pump design. The classical one-dimensional design rules applied for decades are no longer sufficient, and for this reason, there is a strong need for more accurate numerical tools to predict the cavitation behavior of pumps.

An original cavitation prediction method is presented and its results are compared to experimental data for five pumps of specific speeds from 15 to 125 (775 to 6500 US units). The numerical method introduced in this paper for the prediction of the cavitation development in pumps is a simplified version of a previously developed program successfully used in similar cases. Motivation for this simplification was to allow the program usage during the design process of suction impellers. The accuracy of this simplified method as well as its usage as a design tool is discussed.

INTRODUCTION

The usage of numerical tools to design and optimize pump hydraulics is nowadays a standard in the industry (Casartelli, et al., 1995; Zangeneh, et al., 1996, 1998; Goto, 1997). Up-to-now, this optimization has mainly focused on the efficiency and on the stability of the head curve through a better control of the recirculation and a reduction of secondary flows. As a matter of fact, the cavitation is also a major limiting factor in pump design. Nevertheless, only few attempts to improve cavitation in pumps using numerical approaches have been done. Most of the time, this optimization is limited to a finer adaptation of vane inlet angles based on the predicted flow angles obtained from cavitation free flow computational fluid dynamics (CFD) analysis and to an improvement of the profiling of the vanes in order to limit the minimum pressure (Spring, 1992). If this approach gives usually good results for the improvement of the cavitation inception, it does not help the designer to improve the suction capabilities of the pump. These strongly depend on the way the cavitation developed along the vane as a function of the pressure level and how its presence affects the performance of the pump. An optimization of the suction capabilities is then only possible if one is able to control the effect of a modification of the vane shape on the cavitation development and to predict when this cavitation development will impair the head developed by the pump. For that reason, there is a strong need for an accurate and rapid method to predict the three-dimensional cavities' development as well as the associated performance drop. This is especially true for the design of high suction specific speed pumps for which a smooth operation at partload can only be obtained by a strong optimization of the vane shape taking into account the three-dimensional behavior of the flow in front of the impeller.

In the last decades, models for the prediction of cavitation development have been refined and successfully applied on isolated 2-D profiles for steady (Ukon, 1980; Uhlman, 1983; Favre, et al., 1987; Lemonnier and Rowe, 1988; Dupont and Avellan, 1991) and unsteady flows (Delonnay, 1989; Delonnay and Kueny, 1990; Kubota, et al., 1989, 1992). In the same period, few 3-D models, based on potential (Kinnas and Fine, 1993), on S1/S2, or on Euler (Maître and Kueny, 1990; Kaenel, et al., 1995) methods have been developed and applied, but no one is taking into account the 3-D turbulent and viscous effects of the cavity on the mean flow.

More recently, attempts to use the volume-of-fluid method to predict the cavitation have been performed (Dieval, et al., 1998) and some commercial CFD codes already proposed such capabilities.

The different methods presented above can be divided into two main groups:

- The cavity interface tracking method, which iteratively adapts the cavity shape in order to reach a given condition (velocity or pressure) at its interface,
- The two-phase model approach, using a phase change model in the flow calculation code.

While the second approach is certainly able to model much more realistically the phenomena involved in the cavitation development, its application needs an unsteady approach. This means a large computation time, which is not compatible with a design process.

A program to predict the cavitation in pumps of the first kind has been developed at the laboratory of Hydraulic Machinery of the Swiss Federal Institute of Technology with financial support of the Swiss government and several Swiss industries including Sulzer Pumps. The goals of this program development were to:

- Calculate the evolution of the length of an attached cavity with the NPSH value taking into account the viscous, turbulent, and three-dimensional nature of the flow in a pump,
- Predict the head impairment due to the cavitation development,
- Minimize the computational effort in order to use the program in a design process,
- Allow a stand-alone usage of the code with a coupling to any available CFD code.

The method used, introduced by Dupont and Avellan (1991) and developed by Hirschi, et al. (1997), is an iterative determination of the interface shape based on a constant pressure constraint. This iterative approach is needed as the cavity is supposed to have an influence on the mean flow. A deformation algorithm is used to modify the interface shape in order to reach a constant pressure equal to the vapor pressure along it. The reader can refer to Hirschi, et al. (1998b), for a detailed description of the cavity surface tracking method. The iterative process is coupled with the 3-D flow calculation but independent from it to give the user the ability to choose any available CFD code. In order to increase the speed of the adaptation process of the interface location, an initial shape of the cavity is determined using an original method.

Thanks to the initial cavity shape used, the iterative process is quite fast. Only small adaptation of the cavity surface is needed between each step, and the CFD solution of the previous step can be used as an initial solution for the next one.

The proposed approach has successfully predicted cavitation development as well as performance losses on pumps (Hirschi, et al., 1998a) and on largely twisted hydrofoils (Hirschi, et al., 1998b).

Nevertheless, the number of steps needed by the iterative process is too large to allow the method to be used in a design process. For this reason, a simplification of the calculation method is proposed.

MODELIZATION OF CAVITATION

The comparison of the results obtained with the developed method to the experimental data (Hirschi, et al., 1997, 1998a, 1998b) has shown the initial cavity shape to be very close to the measured one for moderate cavity length. This has been observed to be true as long as the cavity development does not affect the main flow. In general, this holds as long as the cavity does not reach the throat of the blade-to-blade channel. In order to have a fast estimation of the cavity length, it was decided to use the initial cavity shape developed in the original method without applying the surface tracking method.

Cavitation Free Flow

For the results presented in this article, the cavitation free flow is calculated using the Reynolds-averaged Navier-Stokes code TASCflow™. This code has been intensively used for turbomachinery flow calculation and validated (Guelich, et al., 1997; Muggli, et al., 1997). The calculations presented here are done for isolated impellers, not taking into account the upstream and downstream components of the pump. In order to calculate the cavitation coefficient σ_{U1} based on the condition at the pump entry from the one based on the reference values for the CFD calculation, χ_e as given in Equation (2), a correction formula has to be applied. This correction, developed in Equation (1), takes into account the losses coefficient $\zeta_{s\div 0}$ between the pump and the impeller entries, as well as the prerotation flow angle α_0, the volumetric efficiency η_v, and the flow coefficient φ_1.

$$\sigma_{U1} = \chi_e \frac{R_2^2}{R_1^2} + \varphi_1^2\left(1 + \frac{1}{tg^2\alpha_0} + \zeta_{S\div 0}\right) \quad (1)$$

$$with\ \sigma_{U1} = \frac{p_s - p_v}{\frac{1}{2}\rho U_1^2}\ and\ \chi_e = \frac{p_0 - p_v}{\frac{1}{2}\rho U_2^2} \quad (2)$$

Cavity Length

The initial cavity shape is calculated solving a normalized form of the well-known Rayleigh-Plesset equation, as given in Equation (3). It is supposed that the envelope of bubbles in evolution over the profile can approach the cavity shape, as shown in Figure 1.

The radius instead of the diameter of the bubble is chosen to define this envelope. This is based on the experimental observation of a hemispherical shape of the bubbles in evolution along a solid

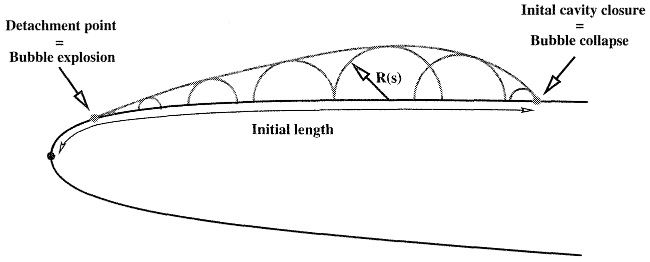


Figure 1. Bubble Envelope Taken as Initial Cavity Shape.

wall and on a measured height of these bubbles corresponding roughly to their planform radius (Arn, et al., 1998). A nondimensional form of the Rayleigh-Plesset equation is solved to improve the stability of the solution of the differential equation.

$$2r\ddot{r} + 3\dot{r}^2 + \frac{8}{Re}\bullet\frac{\dot{r}}{r} + \frac{4}{We^2}\bullet\frac{1}{r} - G\bullet\frac{1}{r^{3\Gamma}} = -\sigma - C_p(t^*) \quad (3)$$

Where:

$$r = \frac{R_b}{R_2}$$

$$C_p = \frac{p(t*) - p_v}{½\rho U_2^2}$$

$$Re = \frac{U_2 R_2}{\nu}$$

$$G = r_0^{3\Gamma}\cdot\left(\sigma + \frac{4}{We^2}\frac{1}{r_0}\right)$$

$$t^* = \frac{t \cdot U_2}{R_2}$$

$$We = U_2\sqrt{\frac{\rho R_2}{\gamma}}$$

$$\sigma = \frac{p_0 - p_v}{½\rho U_2^2}$$

A typical nucleus size is chosen to initiate the calculation. A comparison is done of the nucleus size with the critical radius according to the minimum pressure along the considered mesh line. If the critical size is too small to ensure the explosive development of the nuclei, the calculation is not performed and the point is considered free of cavitation or corresponding to development of isolated bubbles.

This calculation of bubble growth and collapse gives a rapid estimation of the detachment and the closure locations of the attached cavity, as shown in Figure 1. The cavity length is then defined as the collapse location of the bubbles. For each operating point to analyze, this Rayleigh-Plesset is solved on five streamlines from hub to shroud along the impeller vane in the main flow direction using the pressure distribution obtained for the noncavitating condition. An example of the predicted cavity length, divided by the impeller mean outlet radius, is given in Figure 2 as a function of the sigma value. The nonhomogeneous cavity development along the span of the vane is very well illustrated in this example. The cavity develops much sooner at shroud compared to hub, as expected because of the higher relative velocity.

The incipient cavitation coefficient is defined as the first nonzero cavity length along the vane span, as shown in Figure 2. Due to surface tension and bubble dynamic effects, this value does not correspond to the minimum pressure coefficient along the vane. For this reason, the prediction of the incipient cavitation coefficient

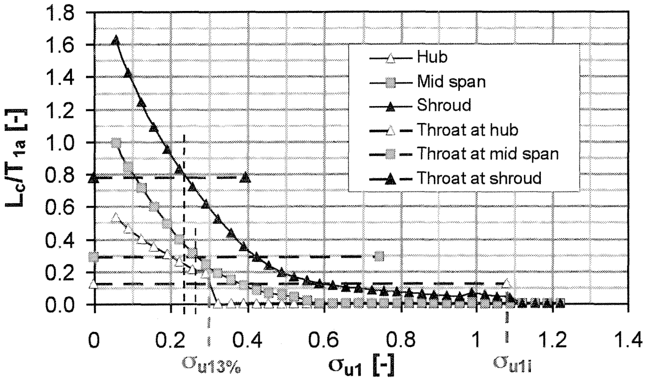


Figure 2. Predicted Cavity Length as a Function of the Sigma Value Along Three Streamlines.

based only on the minimum pressure calculated from the cavitation free condition is wrong. This remark will be further developed for one calculation example given below.

As this incipient condition is quite difficult to determine on the test bed and depends on the quality of the water, a cavity length of 1 percent of the impeller mean outlet diameter is used in practice as a criterion for the cavitation inception. It is this criterion that will be compared to the measurement in the application examples presented hereafter.

Performance Impairment

In the original method, the performance impairment was directly calculated based on the calculated modification of the flow due to the cavity development. In this simplified approach, an attempt to predict the beginning of the performance impairment due to cavitation is done based on the cavity length. The impairment is supposed to take place when the cavity induces a blockage in the flow. When the cavity is attached to the suction side of the vane, the sigma $\sigma_{3\%}$ is predicted when the cavity length is equal to, or greater than, the blade-to-blade channel throat location for one streamline along the blade span. When the cavity develops on the pressure side of the vane, the sigma $\sigma_{3\%}$ condition is supposed to be reached for a relative cavity length of 1 percent.

This method is known to be very rough, but it has already been used with some success in the past. The comparison with the experimental result done in this paper also shows the method to give quite accurate results as long as the throat of the blade-to-blade channel is correctly calculated.

These blade-to-blade throat positions are given for the example in Figure 2 along the different streamlines by the horizontal dotted lines. For the example given in Figure 2, a sigma $\sigma_{3\%}$ of 0.3 is predicted. It has to be noted that the head impairment is due to the cavity reaching the blade-to-blade throat at hub, even if the cavity starts to develop at shroud much sooner than at hub. This is due to the fact that the throat area is positioned more downstream at shroud than at hub, allowing a larger cavity to develop before reaching the blade-to-blade throat. It shows the necessity to calculate accurately the cavity length as a function of the cavitation coefficient to be able to predict the associated head impairment.

A possible physical explanation of the correlation between the beginning of the head impairment and the cavity length reaching the blade-to-blade throat is the influence of the wake of the cavity on the velocity field at the impeller outlet. As shown by Hirschi, et al. (1998a), from Navier-Stokes calculations including the attached cavity, the wake of the cavity is drastically reduced when reaching the blade-to-blade throat due to the relative flow acceleration. No modification of the velocity profiles at the impeller outlet is then observed. On the contrary, when cavities are longer than the throat location, there is no more reduction of the wake, resulting in a significant change in the velocity distribution at the impeller outlet that can lead to the head drop.

APPLICATION

The simplified version of the cavitation prediction program is applied in this article to five radial and semi-axial pumps ranging from specific speed n_q of 15 to 125 (775 to 6500 in US units) (Figure 3), with suction specific speed ranging from 170 to 280. The types, specific speeds, and flow coefficients of these pumps are summarized in Table 1.

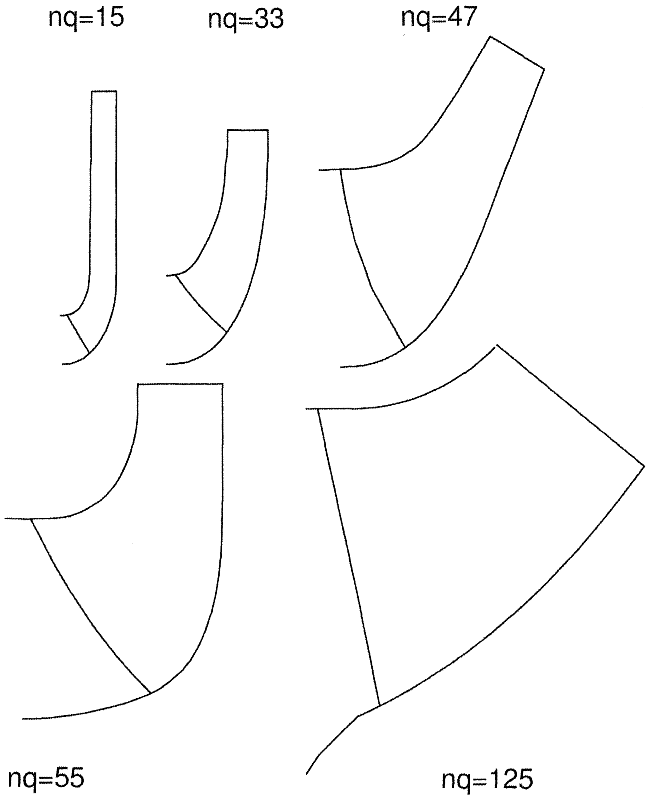


Figure 3. Meridian Contours of the Pumps Used in the Study.

Table 1: Pumps Used in the Study.

	Pump type	Specific speed n_q	Suction specif speed n_{ss}
1	Radial, diffuser	15* (17**)	173
2	Radial, diffuser	33	178
3	Semi-axial, diffuser	47	209
4	Radial, volute	55	278
5	Semi-axial, diffuser	125	283

*a_3= 0.017; **a_3= 0.024

Radial Diffuser Pump N_q = 15

The results obtained for a radial diffuser pump of a specific speed of n_q = 15 (Pump #1 in Table 1) are presented on Figure 4. The predicted cavitation coefficients corresponding to relative cavity lengths L_c/D_2 of 1 percent (circles), 4 percent (diamonds), and 10 percent (squares), represented with hollow points, are compared to the measurement, marked with plain points. On the same figure, the measured sigma 3 percent values (triangles) are compared to the predicted one.

This comparison is very good taking into account the unavoidable scatter of the test data for cavity length measurements. In the observed flow rates (0.15 to 0.35), one can then suppose the cavity to develop on the suction side of the blade. Nevertheless, a steep rise of the measured sigma 3 percent curve (gray triangles), characteristic of a cavity development on the pressure side of the blade, is observed for flow coefficients above 0.27. This behavior

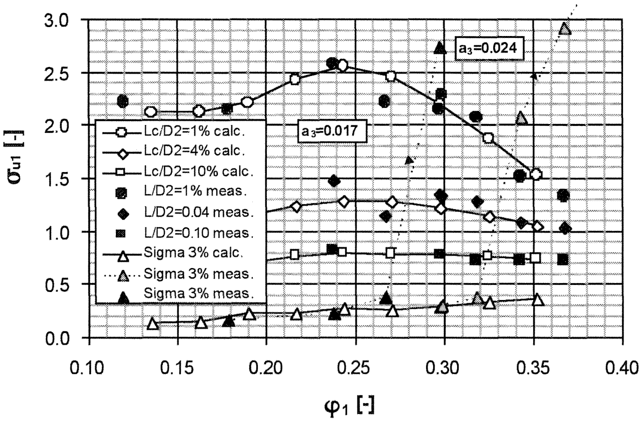


Figure 4. Predicted and Measured Cavity Lengths for a Radial Diffuser Pump of N_Q = 15 (775 US Units).

is not predicted by the calculations, which gives a slowly rising sigma 3 percent curve for the entire flow regime.

In order to explain this difference, two measured sigma 3 percent curves are given in Figure 4, corresponding to two diffusers having different throat areas. It can be observed that the sigma 3 percent rise occurs for a much smaller flow coefficient for the diffuser with a relative throat area of 0.017 (φ_1 = 0.27) compared to the diffuser with the relative throat area of 0.024 (φ_1 = 0.32).

This phenomenon is most probably due to cavitation taking place at the diffuser inlet due to excessive velocities. Steady-stage calculations performed by Hirschi, et al. (1998a), were not able to detect pressure below the vapor pressure in the diffuser. Therefore an unsteady effect due the rotor-stator interaction is supposed to induce that cavitation.

The modification of the velocity field at the outlet of the impeller due to cavitation attached to the leading edge of the impeller vanes can be a second possible explanation for the head impairment depending on the diffuser geometry. As shown by Filipenco, et al. (2000), and by Deniz, et al. (2000), the diffuser performance is very sensitive to small variations of the inlet flow field. Nevertheless, if this second explanation may be the reason for the very beginning of performance drop due to the diffuser, it fails to explain the impairment observed for sigma values above the incipient cavitation coefficient of the impeller.

Radial Diffuser Pump N_q = 33

The results obtained for a radial diffuser pump of a specific speed of n_q = 33 (Pump #2 in Table 1) are presented in Figure 5. Additionally to the calculated and measured sigma values corresponding to cavity lengths of 1 percent, 4 percent, and 10 percent as well as the sigma 3 percent curves, the cavitation inception curve is plotted in the figure.

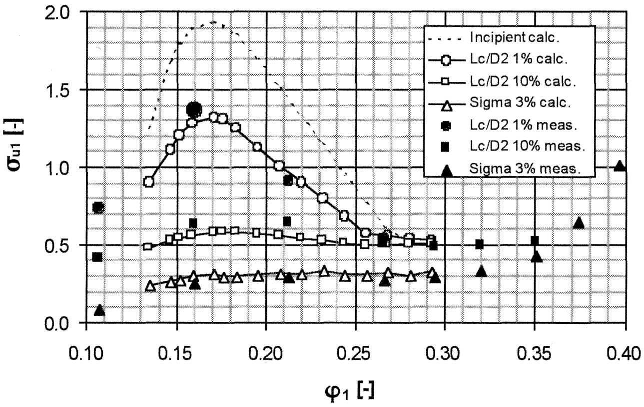


Figure 5. Predicted and Measured Cavity Lengths for a Radial Diffuser Pump of N_Q = 33 (1700 US Units).

One can observe this incipient cavitation coefficient, corresponding to the minimum calculated pressure coefficient on the impeller vanes $\chi_{ei} = -Cp_{min}$ and referenced to the pump inlet using Equation (1), is far from the first observable developed cavity, corresponding to a relative cavity length L_c/D_2 of 1 percent. This proves, if necessary, the utility of such a tool to predict the effective visible incipient cavitation limit, taking into account the delay due the dynamic effects on the cavitation development.

It is interesting to follow, in Figure 6, the evolution from hub to shroud of the streamwise minimum pressure coefficient along the pressure and the suction side of the vane. This minimum pressure is fairly constant along the vane span at BEP. For a relative flow rate of 55 percent, this well-defined minimum occurs on the suction side close to the shroud.

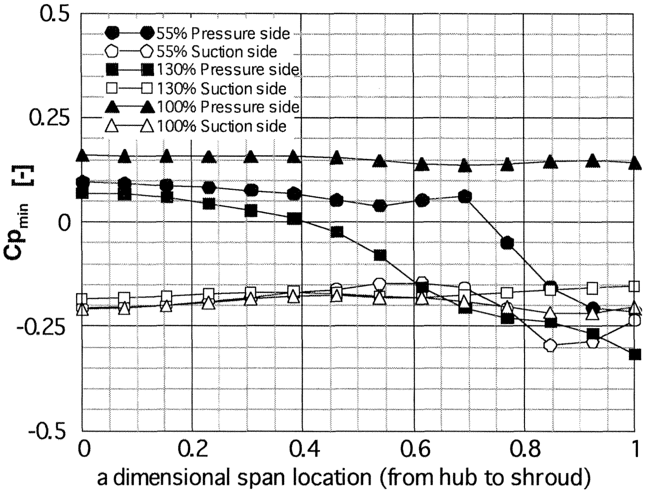


Figure 6. Evolution of the Minimum Pressure Coefficient Along the Vane Span for Different Operating Points (Pump N_Q = 33 (1700 US Units)).

This is due to the recirculation that takes place at partload close to the impeller shroud, locally reducing the inlet flow angle. For a relative flow rate of 130 percent, the minimum pressure coefficient is still at shroud, but it occurs on the pressure side of the blade for nondimensional span values above 0.6. This example illustrates quite well the dilemma of the pump designer. On one side, he is tempted to slightly increase the vane inlet angle at shroud in order to improve the sigma 3 percent at overload. On the other side, he would have to reduce it to prevent an increase of the cavitation inception, or even the sigma 3 percent, at partload. The prediction method presented here is a good solution in order to find an optimum between these two contradictory trends.

Semi-Axial Pump N_q = 47

The results obtained for a semi-axial pump of a specific speed of n_q = 47 (Pump #3 in Table 1) are presented in Figure 7. For this pump, only the incipient and 3 percent sigma values test results were available. For this reason, only these predicted values are reported in Figure 7.

The cavitation predictions are in good agreement with the measurements. It especially well reproduces the change from suction to pressure side of the cavitation development. The flow coefficient corresponding to the rise of the sigma 3 percent at overload is very accurately predicted. This is very important for a design point of view. In order to reach the design goals, this limit needs to be very well predicted, because it will strongly influence the maximum run out condition.

Radial Volute Pump N_q = 55

The results obtained for a radial volute pump of a specific speed of n_q = 55 (Pump #4 in Table 1) are presented in Figure 8. Once

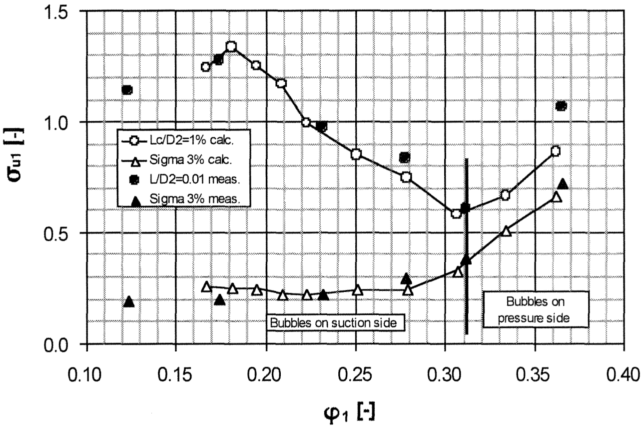


Figure 7. Predicted and Measured Cavity Lengths for a Semi-Axial Pump of N_Q = 47 (2425 US units).

again, the numerical results fit the measured ones very well. The example shows quite well one of the advantages of using such a cavitation prediction tool. No measurement is available for flow coefficient around the local calculated maximum in the visible cavitation corresponding to a relative cavity length of 1 percent, at φ_1 equal 0.19. For this reason, the required NPSH for cavitation-free operation on the entire flow range based on the measurement data is probably underestimated for this pump. Additional measurement points would have been done if these numerical data were available prior to the tests.

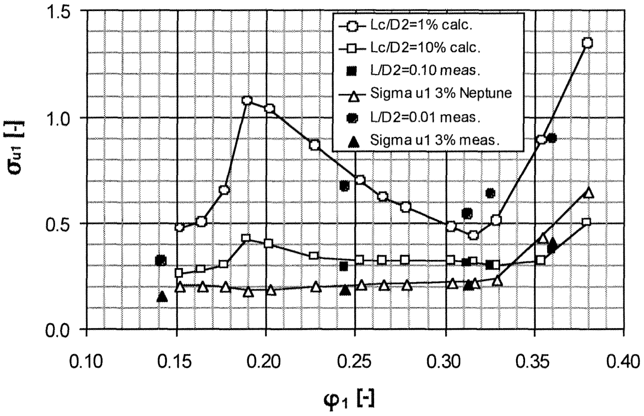


Figure 8. Predicted and Measured Cavity Lengths for a Radial Volute Pump of N_Q = 55 (2835 US units).

These complementary tests were performed for the n_q = 33 pump (Pump #2 in Table 1) on the basis of the numerical cavitation prediction. This is the larger full circle point at a flow coefficient of φ_1 equal 0.17 given in Figure 5. Thanks to the numerical prediction, the information regarding the cavitation-free condition has been improved.

Semi-Axial Pump N_q = 125

The results obtained for a semi-axial pump of a specific speed of n_q = 125 (Pump #5 in Table 1) are presented in Figure 9. For this pump, the measured sigma 0 percent is also available and is presented by gray triangle points. The sigma 0 percent corresponds to the very beginning of the performance impairment due to cavitation. The local rise of the measured sigma 0 percent at partload around φ_1 = 0.26, is very similar to the predicted shape of the sigma values corresponding to a 10 percent relative cavity length.

This confirms the accuracy of the performance impairment predictions, even if the calculated sigma 3 percent values at flow coefficients below 0.26 are underestimated for this pump.

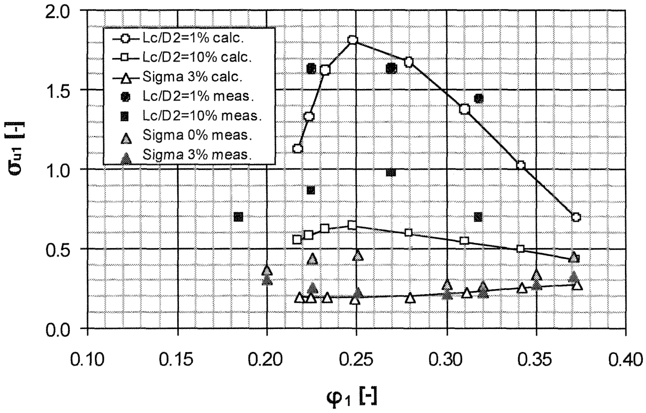


Figure 9. Predicted and Measured Cavity Lengths for a Semi-Axial Pump of N_Q = 125 (6445 US units).

CONCLUSIONS

A new tool for the prediction of the cavitating flows in pumps has been presented. It consists in a simplified version of a cavity interface tracking method coupled with a Navier-Stokes flow solver. This simplified approach allows this tool to be used in the design process of runners. Examples of characteristic results are given for five pumps ranging from specific speeds of 15 to 125.

The total calculation time needed in order to obtain the cavitation-free flow is about eight hours for a typical number of operating points equal to 10 using a standard workstation. To this time, an additional two hours are needed to calculate the attached cavity lengths for all operating points for about 50 cavitation coefficients ranging from the minimum pressure coefficient over the blade down to zero. This means an overnight calculation in order to obtain cavitation free as well as cavitation performance curves.

The comparison of the obtained results with test bed measurements shows this method to be very accurate for both the prediction of the cavitation inception and the performance impairment.

For the pump user, the ability of the presented method to predict the increase of the incipient cavitation, as well as the possible increase of the $NPSH_{3\%}$ at partload will help the definition of the necessary flow regimes to be tested on the test bed in order to obtain more accurate information in this operating range.

This accurate cavity length prediction can also be used in the future to better estimate the erosion rate for different rated operating points and, by this means, to more precisely predict the requested NPSH for safe operation.

Such a method is also expected to significantly improve the reliability of high suction specific speed pumps, allowing to better forecast the effect of possible inlet recirculation at partload on cavitation behavior.

For the pump designer, this method will help to improve the suction capabilities of the impellers by an accurate prediction of the effects of geometry modifications.

In the future, the very rough method used to predict the head impairment needs to be improved using a more physical criterion as the blockage factor in the throat of the blade-to-blade channel. It also needs to be improved to be able to deal with the effect of the diffuser on the head impairment due to cavitation.

NOMENCLATURE

A	=	Area
a	=	Nondimensional area
C	=	Velocity
C_p	=	Pressure coefficient
G	=	Coefficient depending on nature and size of gaz nuclei
Q	=	Flow rate

L_c = Cavity length
p = Static pressure
p_v = Vapor pressure of pumped liquid
n = Rotational speed in rad • s^{-1}
n_q = Specific speed of pump stage in ISO units
N_s = Specific speed of pump stage in US units
n_{ss} = Suction specific speed in ISO units

$$n \frac{\sqrt{Q}}{NPSH_{3\%}^{0.75}}$$

NPSH = Net positive suction head
r = Relative bubble or nuclei radius
R_b = Bubble or nuclei radius
R_1 = Impeller eye radius
R_2 = Impeller mean outlet diameter
Re = Reynolds number
T_{1a} = Inlet pitch at shroud
t* = Nondimensional time
U = Blade peripheral speed
We = Weber number

Greek

α_0 = Prerotation angle (0 = axial flow)
χ_e = Cavitation coefficient referred to the impeller inlet
γ = Gaz-liquid surface tension
Γ = Specific heat ratio
φ = Flow coefficient
υ = Cinematic viscosity
ρ = Density of the pumped liquid
σ = Thoma's coefficient
ζ = Loss coefficient

Subscript

BEP = Best efficiency point
d = Pump discharge
i = Incipient condition
m = Meridional component
s = Pump suction
0 = Calculation domain inlet
1 = Impeller eye
2 = Impeller outlet
3 = Diffuser inlet

REFERENCES

Arn, C., Avellan, F., and Dupont, P., 1998, "Prediction of Francis Turbines Efficiency Alteration by Traveling Bubble Cavitation," *Proceedings of the Third International Symposium on Cavitation*, Grenoble, France, *1*, pp. 81-86.

Casartelli, E., Evertz, H., Gyarmathy, G., and Saxer, A., 1995, "Secondary Flows in a Centrifugal Compressor Stage as Predicted by Steady Viscous Flow Computations," The Second International Conference on Heat Engines and Environmental Protection, Balatonfuered, Hungary.

Delonnay, Y., 1989, "Modélisation d'écoulements instationnaires et cavitants, thèse," Institut National Polytechnique de Grenoble, France.

Delonnay, Y. and Kueny, J., 1990, "Two Phase Flow Approach in Unsteady Cavitation Modeling," Cavitation and Multiphase Flow Forum, *98*, Toronto, Canada, ASME.

Deniz, S., Greitzer, E. M., and Cumpsty, N. A., January 2000, "Effects of Inlet Flow Field Conditions on the Performance of Centrifugal Compressor Diffusers: Part 2—Straight-Channel Diffuser," *Journal of Turbomachinery*, 122, pp. 11-21.

Dupont, P. and Avellan, F., 1991, "Numerical Computation of a Leading Edge Cavity," Cavitation '91 Symposium, First ASME-JSME Fluids Engineering Conference, Portland, Oregon.

Dieval, L., Marcer, R., and Arnaud, M., 1998, "Modélisation de poches de cavitation par une méthode de suivi d'interfaces de type VOF," *Proceedings of the Third International Symposium on Cavitation*, Grenoble, France.

Favre, J. N., Avellan, F., and Ryhming, I. L., 1987, "Cavitation Performance Improvement Using a 2-D Inverse Method of Hydraulic Runner Design," *Proceedings of International Conference on Inverse Design Concepts and Optimization in Engineering Science-II (ICIDES)*, Pennsylvania State University.

Filipenco, V. G., Deniz, S., Johnston, J. M., Greitzer, E. M., and Cumpsty, N. A., January 2000, "Effects of Inlet Flow Field Conditions on the Performance of Centrifugal Compressor Diffusers: Part 1—Discrete-Passage Diffuser," *Journal of Turbomachinery*, 122, pp. 1-10.

Goto, A., 1997, "Prediction of Diffuser Pump Performance Using 3-D Viscous Stage Calculation," *Proceedings of 1997 ASME Fluids Engineering Division Summer Meeting*, FEDSM'97, Paper FEDSM97-3340.

Guelich, J., Favre, J. N., and Denus, K., 1997, "An Assessment of Pump Impeller Performance Predictions by 3D-Navier-Stokes Calculations," *Proceedings of the ASME Fluids Engineering Division Summer Meeting*.

Hirschi, R., Dupont, P., Avellan, F., Favre, J. N., Guelich, J. F., Handloser, W., and Parkinson, E., 1997, "Centrifugal Pump Performance Drop Due to Leading Edge Cavitation: Numerical Prediction Compared with Model Tests," *Proceedings of the 1997 ASME Fluids Engineering Division Summer Meeting*, Vancouver, Canada.

Hirschi, R., Dupont, P., Avellan, F., Favre, J. N., Guelich, J. F., Handloser, W., and Parkinson, E., 1998a, "Centrifugal Pump Performance Drop Due to Leading Edge Cavitation: Numerical Predictions Compared with Model Tests," *Journal of Fluids Engineering*, *120*, pp. 705-711.

Hirschi, R., Dupont, P., and Avellan, F., 1998b, "Partial Sheet Cavities Prediction on a Twisted Elliptical Planform Hydrofoil Using a Fully 3-D Approach," *Proceedings of the Third International Symposium on Cavitation*, Grenoble, France.

Kaenel, A., Morel, P., Maître, T., Rebattet, C., and Kueny, J. L., 1995, "3-D Partial Cavitating Flow in a Rocket Turbopump Inducer: Numerical Predictions Compared with Laser Velocimetry Measurements," *Proceedings of the International Symposium on Cavitation*, CAV'95, Deauville, France, pp. 57.

Kinnas, S. A. and Fine, N. E., 1993, "A Numerical Nonlinear Analysis of the Flow Around Two- and Three-Dimensional Partially Cavitating Hydrofoils," *Journal of Fluid Mechanics*, *254*, pp. 151-181.

Kubota, A., Kato, H., and Yamaguchi, H., 1989, "Finite Difference Analysis of Unsteady Cavitation on a Two-Dimensional Hydrofoil," *Proceedings of the Fifth International Conference Numerical Ship Hydrodynamics*, Hiroshima, Japan.

Kubota, A., Kato, H., and Yamaguchi, H., 1992, "A New Modeling of Cavitating Flows: A Numerical Study of Unsteady Cavitation on a Hydrofoil Section," *Journal of Fluid Mechanics*, *240*, pp. 56-96.

Lemonnier, H. and Rowe, A., 1988, "Another Approach in Modeling Cavitation Flows," *Journal of Fluid Mechanics*, *195*, pp. 557-580.

Maître, T. and Kueny, J. L., 1990, "Three-Dimensional Models of Cavitation in Rocket Engine Inducers," ASME Fluid Machinery Forum.

Muggli, F. A., Wiss, D., Eisele, K., Casey, M. V., et al., December 1997, "Flow Analysis in a Pump Diffuser—Part 2: Validation and Limitations of CFD for Diffusers Flow," *Journal of Fluids Engineering*, *119*.

Spring, H., 1992, "Affordable Quasi Three-Dimensional Inverse Design Method for Pump Impellers," *Proceedings of the Ninth International Pump Users Symposium*, Turbomachinery Laboratory, Texas A&M University, College Station, Texas, pp. 97-110.

Uhlman, J. S., 1983, "The Surface Singularity Method Applied to Partially Cavitating Hydrofoils," Ph.D. Thesis, Massachusetts Institute of Technology, Department of Ocean Engineering, Cambridge, Massachusetts.

Ukon, Y., 1980, "Partial Cavitation on Two- and Three-Dimensional Hydrofoils and Marine Propellers," *Proceedings of the Tenth AIRH Symposium*, Tokyo, Japan, pp. 195-206.

Zangeneh, M., Goto, A., and Takemura, T., 1996, "Suppression of Secondary Flows in a Mixed-Flow Pump Impeller by Application of Three-Dimensional Inverse Design Method: Part 1—Design and Numerical Validation," Transactions of ASME, *118*, pp. 536-551.

Zangeneh, M., Goto, A., and Harada, H., October 1998, "On the Design Criteria for Suppression of Secondary Flows in Centrifugal and Mixed Flow Impellers," *Journal of Turbomachinery*, *120*, pp. 723-735.

BIBLIOGRAPHY

Geurst, J. A., August 1959, "Linearized Theory for Partially Cavitated Hydrofoils," *International Shipbuilding Progress*, *6*, (60).

Kinnas, S. A., March 1991, "Leading-Edge Corrections to the Linear Theory of Partially Cavitating Hydrofoils," *Journal of Ship Research*, *35*, pp. 15-27.

ACKNOWLEDGEMENT

The author would like to thank the management of Sulzer Pumps Ltd., Winterthur, for permission to publish this paper. He also extends thanks to the hydraulic development staff of Sulzer Pumps HQ for providing test bed measurements and to the Swiss Federal "Commission pour la Technologie et l'Innovation" for his financial support.