Numerical Evaluation of the Effects of the Rotating Cavities on the Axial Thrust Evaluation in Centrifugal Pumps

NUMERICAL EVALUATION OF THE EFFECTS OF THE ROTATING CAVITIES ON THE AXIAL THRUST EVALUATION IN CENTRIFUGAL PUMPS

A. Marini[†], *C. Bernardini*[†], *S. Salvadori*[†], *A. Piva*[‡], *A. Nicchio*[‡]

[†]University of Florence, Energetic Department "Sergio Stecco"
via di S. Marta, 3 50139 Florence, ITALY
Contact author: Alessandro.Marini@unifi.it

[‡]Weir Gabbioneta S.p.a.
v.le Casiraghi, 68 20099 Sesto S. Giovanni (MI), ITALY

ABSTRACT

From an experimental point of view, the effects of the leakage flows in centrifugal pumps have been widely investigated. The leakage mass-flow and the losses have been correlated to the geometrical parameters that characterize the cavities and to the working conditions. Nevertheless, specific studies on particular geometries are sometimes necessary. To accomplish this necessity a numerical approach based on the CFD simulation of the leakage flows is presented. The leakage flow rate, the thrust acting on the walls and the losses are evaluated and correlated with the working conditions through a 2D/3D CFD study. Several simulations have been performed to account for the possible shift in the shaft positioning, the rotational velocity, the wear of the ring and the three dimensional effects. The front and back shroud of a centrifugal impeller and the central balancing drum of a multistage pump in opposite configuration have been considered. In order to validate the used CFD methodology, the results are compared with the data given by the experimental correlations used by the industry. The results obtained for the leakage flows are used to evaluate the variations in the residual axial thrust in multistage pumps using an in-house developed tool. The obtained thrust curves are compared with each other and show a good agreement with the real thrust supported by the bearing rings.

NOMENCLATURE

A	[m^2]	area
C	[m]	radial clearance of wear ring
C_d		discharge coefficient
d	[m]	diameter
F	[N]	force
g	[m/s^2]	gravity acceleration
H	[m]	head
k	[m^2/s^2]	turbulent kinetic energy
L	[m]	length of wear ring gap
M	[Nm]	moment of pressure force
n		normal vector
p	[Pa]	pressure
Q	[m^3/s]	flow rate
T	[N]	thrust
u, v	[m/s]	velocity
V	[m^3]	cell volume
W	[W]	power

y^+		wall y plus

Greek

δ_L	[m^3/h]	leakage flow rate
ε	[m^2/s]	dissipation rate
ρ	[kg/m^3]	density
ω	[rad/s]	rotational velocity speed

Subscripts and Superscripts

ax	axial
bs	back shroud
bus	bushing
fs	front shroud
inlet	inlet section
mom	momentum
r	radial
t	tangential

INTRODUCTION

The evaluation of the residual axial thrust in a multistage centrifugal pump is a task that has interested several researchers and industries over the years. During the pump working period the thrust is balanced by bearings which guarantee the mechanical reliability of whole pump. To accomplish this objective a good estimation of the forces acting on the pump surfaces over a wide range of operating conditions is necessary. Then, the leakage flow inside the gaps between impellers and pump stationary walls must be studied. The pressure distribution on the back faces of the impellers is a function of the cavity mass flow, which depends on the local working conditions (e.g. impeller or diffuser head). Since the changes of the leakage flow rate affect the operating point of the impeller, an iterative procedure should be used to couple the study of the components and obtain the actual behavior of both impeller and cavity system.

The problem of the experimental (EXP) evaluation of the loss coefficient and the mass-flow in the cavities has been studied by several authors. Amongst them, the works of Gülich (1999, 2003) should be underlined. Starting from the initial work of Nece and Daily (1960) and referring to the contribution of many other authors, Gülich proposed a detailed table of correlations for the evaluation of the power loss coefficient, of the pressure coefficient and of the leakage flow. All these parameters have been correlated with the geometrical data (cavity width, casing diameter, surface roughness), the working conditions (pressure rise across the cavity and rotational velocity) and the Reynolds number. Other relevant contribution to the leakage flow study can be found in Baskharone and Wyman (1999), Debuchy *et al.* (1998), Storteig (1999), Tamm and Stoffel (2002) and Utz (1972).

A general starting point for the axial thrust evaluation in centrifugal pumps is represented by the approach proposed by Lobanoff and Ross (1985). Even though the proposed equations can provide reliable information on the thrust acting on a single stage, the complexity of the real machines introduces changes in the expected results, especially when considering multistage pumps with facing impeller backwards. More detailed analysis on the axial thrust evaluation and on the effects of the cavity flow conditions have been performed by Gantar *et al.* (2002) and Della Gatta *et al.* (2006). In the study of Gantar, the pressure and velocity profiles within the cavities have been experimentally and numerically investigated considering several realistic working conditions. In the work of Della Gatta the authors describe a CFD based approach for the evaluation of the residual axial thrust of a generic pump, focusing on the results obtained 2D/3D simulation of the actual cavity geometries. Both the authors stated that the Computational Fluid Dynamic (CFD) can be an effective tool for axial thrust prediction.

The present paper is concerned with the investigation of a multistage centrifugal pump designed and produced by Weir Gabbioneta Srl. The residual axial thrust has been evaluated for different flow rates and a detailed analysis of the leakages has been conducted. The positioning effects, the radial gap increase and the off-design conditions have been investigated and the obtained data have been compared with the correlations used by the industry. In order to govern the residual axial thrust direction and intensity, possible geometrical changes to the cavity have been investigated and the global effects on the axial thrust have been compared and commented.

AXIAL THRUST EVALUATION

The multistage centrifugal pump reported in Figure 1a has been designed by Weir Gabbioneta Srl and has been investigated by the University of Florence. The numerical methodology used for the evaluation of the residual axial thrust have already been described in Della Gatta *et al*. (2006) and in Salvadori *et al.* (2007), then just a brief description will be given here. The evaluation of the forces acting on each single stage is the initial point to determine the residual axial thrust. This balance can be expressed as follows:

$$\int_A p_{fs}\,\vec{n}\,dA + \int_A p_{bs}\,\vec{n}\,dA + p_{inlet}A_{inlet}\vec{n} + \rho\frac{Q^2}{A_{inlet}}\vec{n} - p_{bus}A_{bus}\vec{n} = \vec{T}_{ax} \tag{1}$$

Equation 1 is referred to the control volume reported in Figure 1b and contains all the rotating walls. The contribution to the thrust T_{ax} are respectively the global force acting on the front and back shroud walls, the pressure field at the impeller inlet section, the momentum contribution in axial direction and the pressure integral on the bushing walls. For a defined single stage geometry at a chosen flow rate the axial thrust can be calculated once the pressure field inside its components, including leakage cavities, is known. Therefore the multistage pump axial load can be obtained algebraically adding the single stage contributions which have been calculated by applying the above equation.

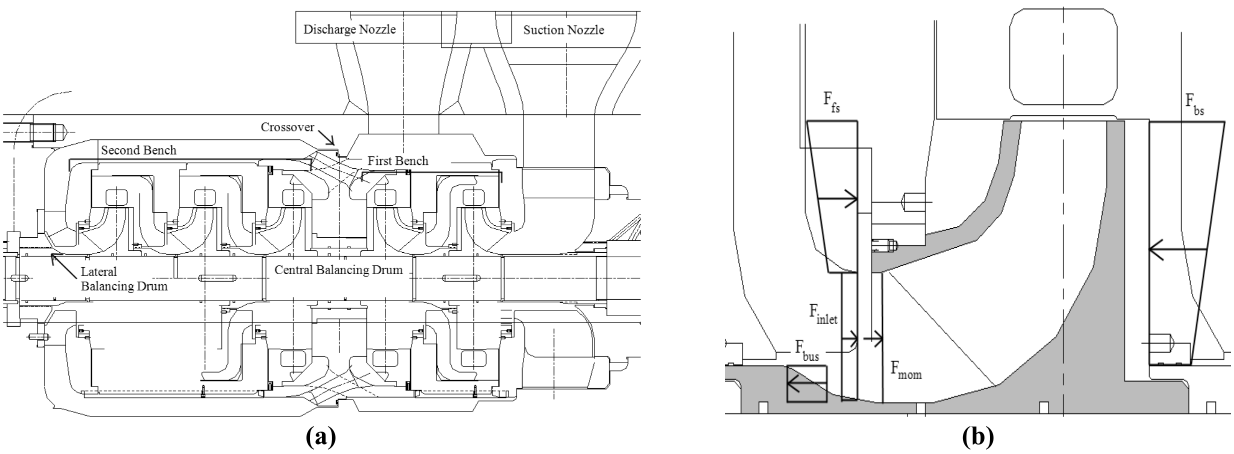


Figure 1 Multistage centrifugal pump by Weir Gabbioneta Srl (a) and forces description (b)

NUMERICAL METHODOLOGY

The used numerical approach has been described in the already cited works by Della Gatta *et al.* (2006) and Adami *et al.* (2005) and only a brief description will be given here. Several 2D/3D steady simulations solving Reynolds Averaged Navier Stokes equations have been carried out in the fluid domain. Standard techniques have been employed for the numerical computation. A finite-volume pressure-correction procedure for incompressible flow has been used. An analysis on the turbulence models' effect has been conducted on the 3D central drum only. The reason for this choice was that this component is fundamental for the evaluation of the direction and the intensity of the thrust in multistage pumps with opposite benches. The standard *k-ε* model with wall functions and enhanced wall treatment and the *k-ω* model have been evaluated. Considering the thrust, the differences between the obtained results are negligible while a variation on the leakage mass-flow and on the losses can be found. The *k-ε* model with wall functions has been used, since it performed well on previous studies on a similar centrifugal pump. This choice was also supported by a comparison between the losses and the leakage mass-flow estimated by CFD with the results obtained by correlations presented in the paper.

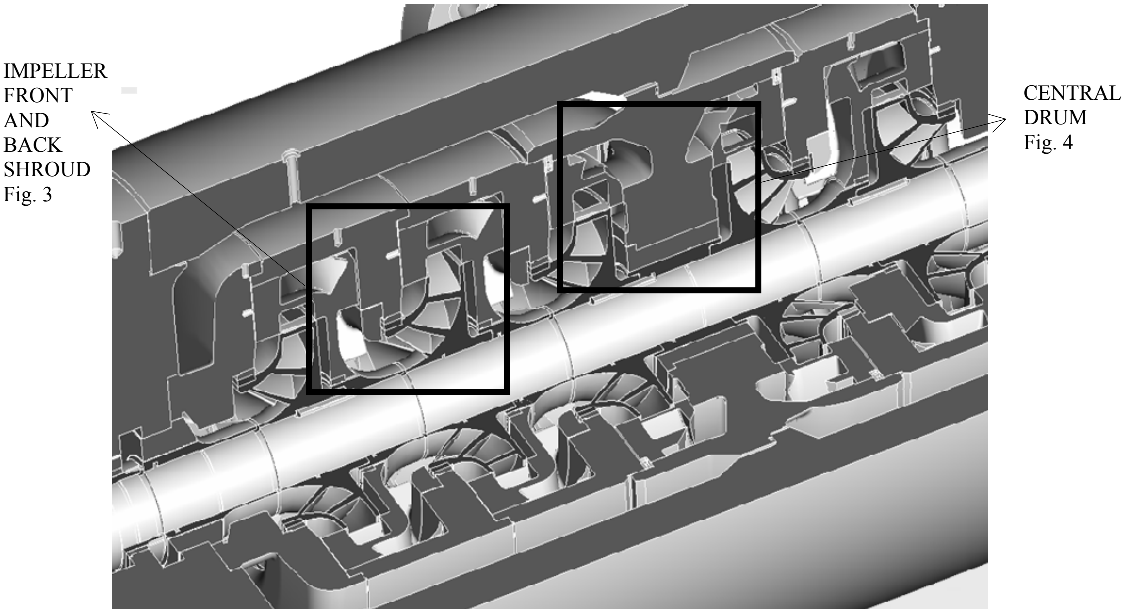


Figure 2 Cross-section of the multistage centrifugal pump

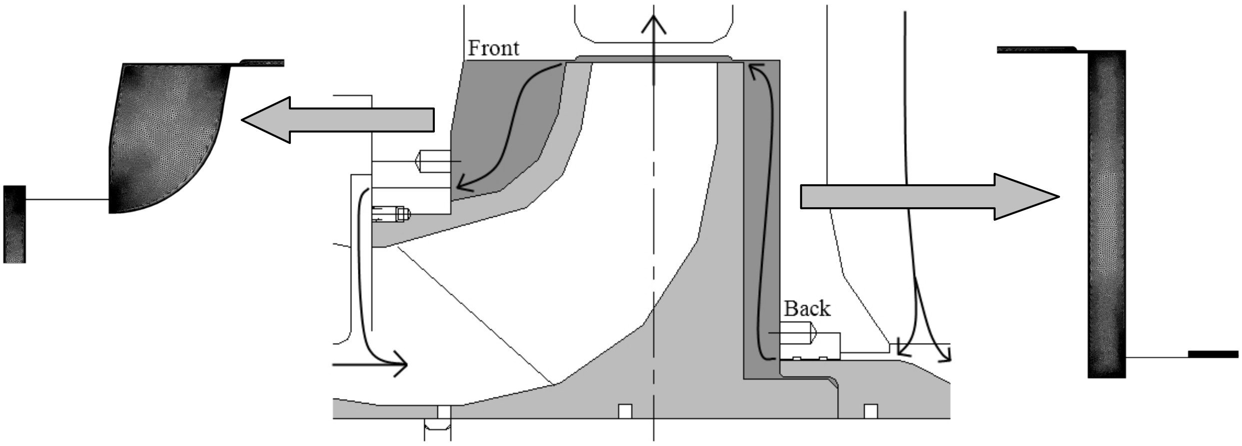


Figure 3 Impeller front and back shroud geometry and grids

Pump main components have been analyzed separately in order to reduce computational costs. Impeller side chambers have been studied by means of 2D and 3D CFD approaches. Three components have been taken into account: the front shroud on the impeller side, the back shroud on the diffuser side (Figure 3)and the central balancing drum (Figure 4). The position of these elements in the context of the global machines is shown in Figure 2. Leakage flow behavior has been characterized by several differential head across the shroud, ranging from 0.3 to 1.2 times the head at the Best Efficiency Point (BEP) of the impeller or the diffuser. Two rotational speeds and three different shaft positions have been studied. The hypothesis of axial symmetry has been reasonably assumed for these problems except for the central balancing drum and the leakages equipped with winglets. Results from the front and back shroud chamber analysis are discussed here below and details of the corresponding computational grids are shown. These results can be employed to create correlations tuned on the actual geometrical and operating properties of the selected cavities. By such a way, cavity flow is solved at different operating points in a preliminary phase, but does not require to be calculated at every update of the stage coupling boundary conditions.

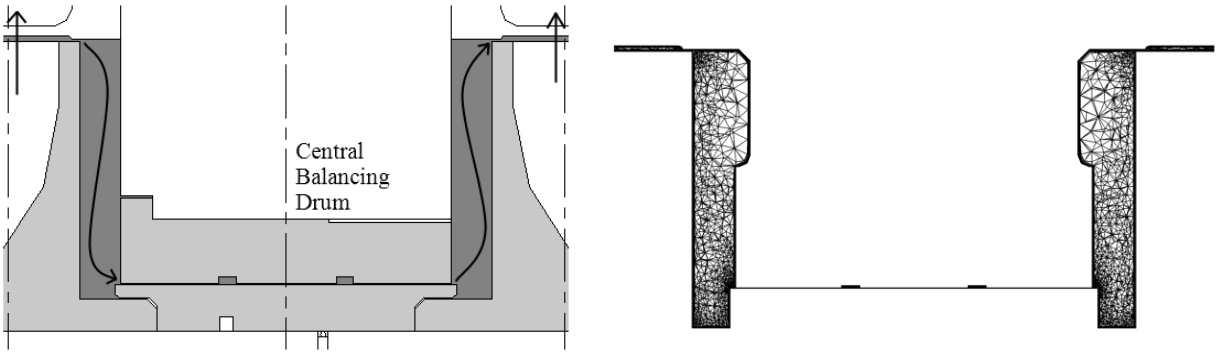
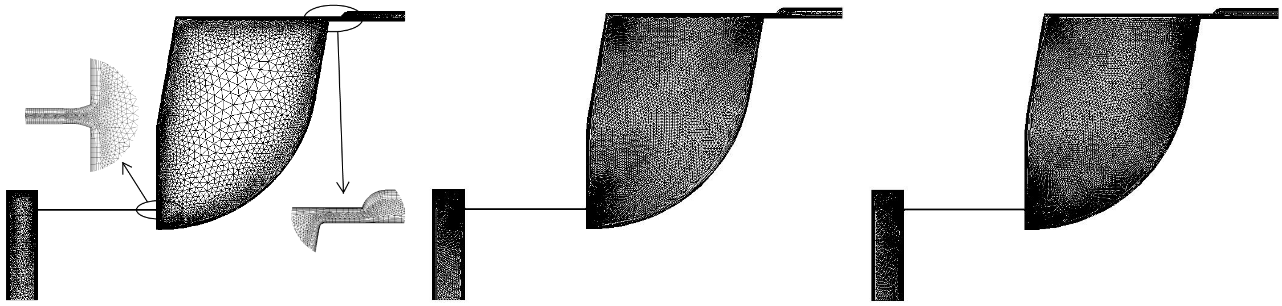


Figure 4 Central balancing drum geometry and grid

DISCUSSION ON THE OBTAINED RESULTS

The results of the simulation campaign that has been carried out are shown hereafter. The results of the grid independence study are presented, and then the influence of shaft installation and the machine detriment on leakage flows and axial thrust are discussed. Furthermore, the behavior of wear rings at different rotational speed and some enhanced configurations, where protrusions inside the front and back shroud cavities have been introduced, have been studied. Some numerical results have been compared with experimental correlations and the differences between the two approaches are shown. All the results have been conveyed in non-dimensional form by normalization in respect to the values at the best efficiency point of the same configuration.



(a) Grid 1 **(b) Grid 2** **(c) Grid 3**

Figure 5 Grids analyzed for the front shroud

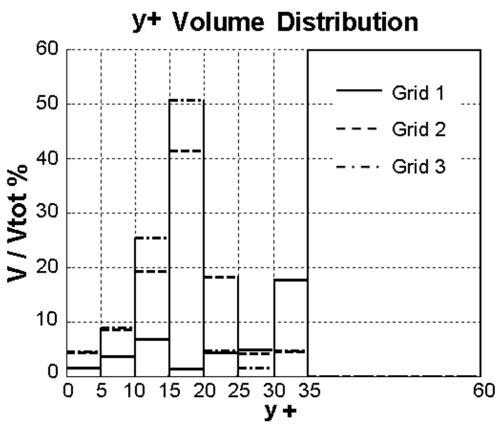


Figure 6 Volume y^+ distribution for front shroud, impeller head/impeller head@BEP=0.95

Grid Independence Evaluation

The studied geometries have been discretized by means of an unstructured approach. Three grid levels have been considered for the 2D geometries of front and back shroud. The number of elements for the three grids of the front shroud is 27300, 70400 and 89800, and are shown in Figure 5. The three grids have a different distribution of y^+ over the volumes as shown in Figure 6. There, V is the cell volume and V_{tot} is the volume of all cells adjacent to the boundary walls. While Grid 2 and Grid 3 do not have cells with y^+ values higher than 35, Grid 1 has 60 % of the wall adjacent cells volume where y^+ is between 35 and 60. Besides Grid 1 has less volume than Grid 2 and Grid 3 with y^+ values less than 10.

Simulations at four different impeller head across the shroud for the three grids have been performed. Figure 7 shows the results of thrust and leakage flow rate against the impeller head. It must be pointed out that the a good discretization of the fluid domain is fundamental for the study of cavity flows as proved by the high difference (≈ 10%) between the axial thrust and the leakage flow values obtained using Grid 1 or Grid 2. Results for Grid 2 and Grid 3 do not vary significantly, and then Grid 2 has been chosen for the cavity characterization. The same analysis has been carried out for the back shroud performance evaluation.

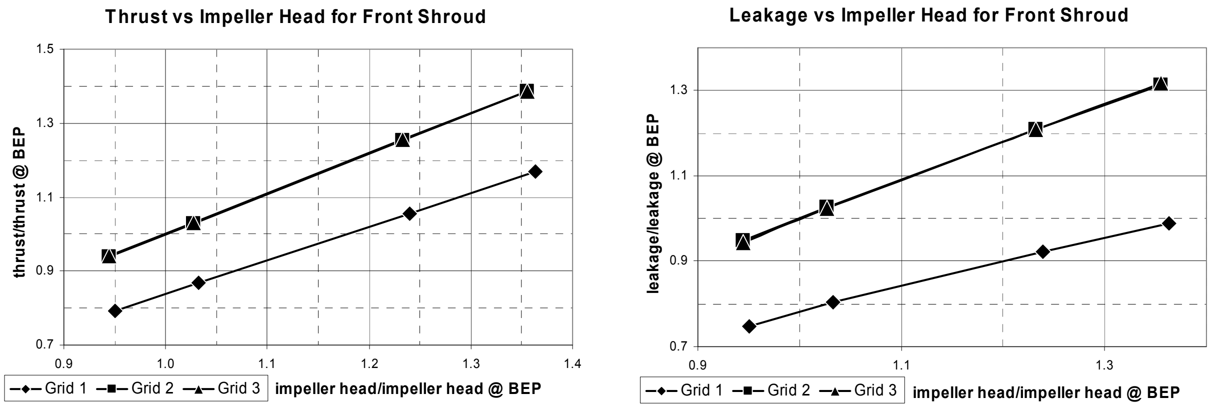
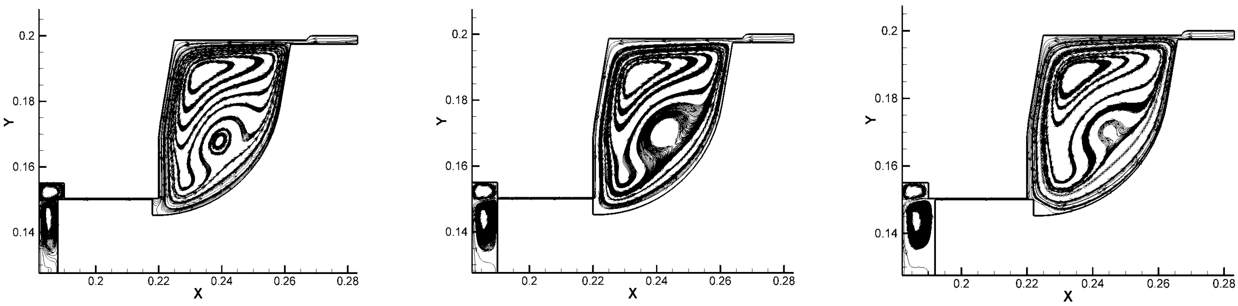


Figure 7 Thrust and leakage of the front shroud for three grid levels



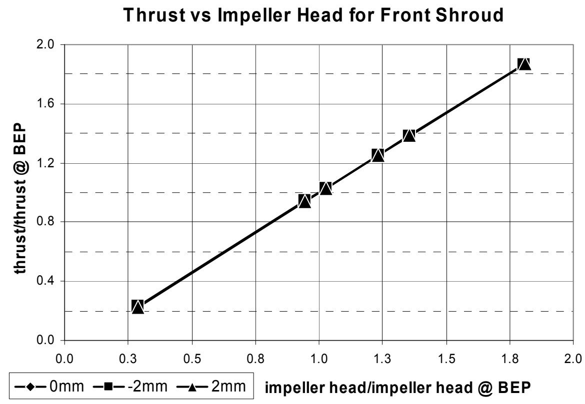
(a) (b) (c)

Figure 8 Streamlines in the front shroud for different shaft displacements

Effects of the Shaft Positioning

Our attention has been directed to some geometrical configurations that differ from the nominal design conditions. These differences could have effects on the global axial thrust thus causing unexpected problems on selected bearings. The first aspect we considered is the possibility of axial shifts during the shaft installation. The relative movement between rotational and fixed components leads up to a change in the wear ring length whose effects should be foreseen. Two opposite displacements have been considered: as suggested by Weir Gabbioneta Srl, a difference of ±2 mm (relative to the design position) has been taken into account.

The streamlines in the front shroud for the three shaft positions are shown in Figure 8. As visible, the main flow structures in the front cavity do not differ very much. The main part of the chambers of the clearance is in stagnation condition. In the center of the cavity, two wide counter-rotating vortices defined by their core position remains the same for the three configurations. These stalled central regions inhibit the fluid that flows from the entrance toward the clearance following the stationary wall. Despite of the expected behavior, the effects of the shaft displacement on the leakage flow is negligible, as shown in Figure 9. The leakage flow rate does not change at all as well as the thrust evaluated on the rotating walls. Similar results have been found for the back shroud and for the drums, too.



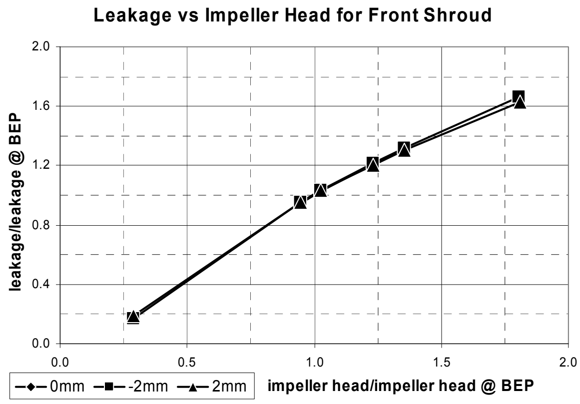
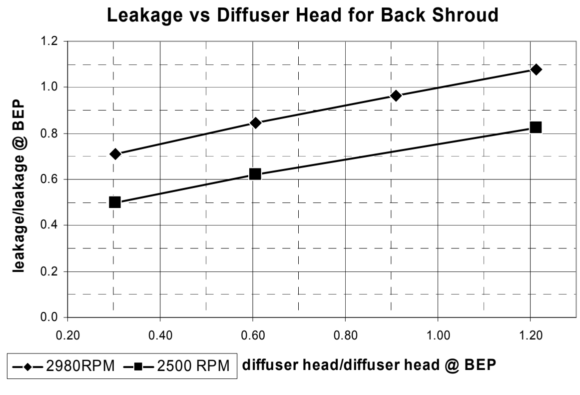


Figure 9 Leakage flow and axial thrust for the front shroud at different shaft displacements

Evaluation of the Rotational Velocity Effect

Rotational speed affects the behavior of cavity flows because of the pumping effects that moves radially outward the fluid in the shroud chambers. The increase in rotational speed generates an improvement of the leakage flow rate for the back shroud. In Figure 10 this increase for the back shroud can be appreciated through normalization relative to the head across the cavity relative to the BEP at 2980 RPM.



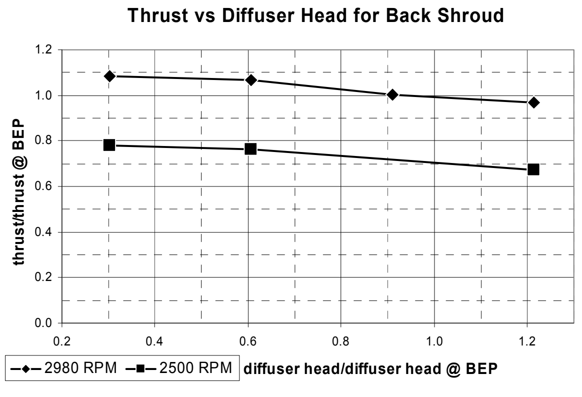


Figure 10 Comparison of leakage flow and thrust for 2500 and 2980 RPM for back shroud

The increased rotational speed yields greater pumping effects thus enhancing the recirculation vortex in the chambers with an increase of leakage flow rate. This behavior can be seen in the radial and tangential velocity profiles in the impeller side chamber (Figure 11) taken at 63% of the radial height and referred to the tangential velocity at the impeller tip. The flow inside the cavity is radially outward at the rotating wall and radially inward at the casing wall (Figure 11a). The internal flow of the impeller side chamber presents a core zone between the diffuser wall and the impeller's one where the rotating speed is much lower than the impeller one and can be individuated for a relative width between 0.2 and 0.8 in Figure 11a. The normalized radial velocity differs not very much in the center of the chamber between the two operating conditions. As a matter of fact this zone is filled up with the vortex core where the flow is almost still. In the zone adjacent to the wall instead, the radial velocity component is different for the two rotational speeds, showing the stronger recirculation for the higher one. The effect of differential head across the shroud is also evident on recirculation, and is consistent with leakage flow increase. The tangential velocity components show the same profiles and are somehow shifted when compared with each other. Then, the flow structures are not affected by the rotational velocity change but its variation generates a stronger recirculation on the traverse plane and a higher pumping effect.

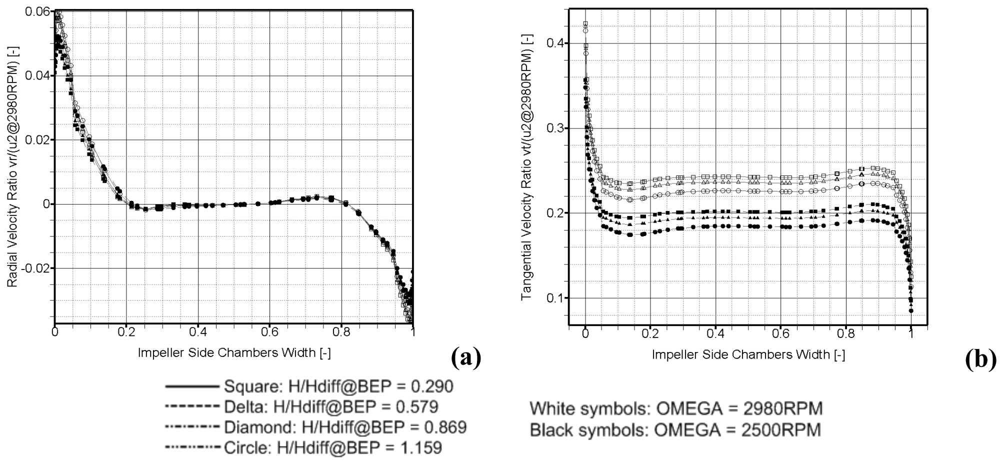


Figure 11 Radial and tangential velocity profiles at different rotational speeds

Comparison between Numerical Data and Correlations

One of the objectives of our study is the evaluation of the reliability of CFD simulations when considering leakage flows. The idea is to support generic correlations obtained for a large number of geometries with "numerical" correlations that take into account the actual geometry of the cavity. The CFD results are then compared with the data obtained by correlations that are currently in use in the industry during the design processes (Nece and Daily (1960), Stirling (1982), Gülich (2003)). The study on the accuracy of the experimental correlations is demanded to the cited papers.

The experimental correlation proposed by Stirling (1982 has been developed treating the wear ring gap as an annular orifice and is commonly used by the industries for a preliminary leakage flow evaluation. The leakage flow rate is thus expressed using a discharge coefficient for the orifice:

$$\delta_L = \frac{C_d A}{2gH_L} = \left(\frac{\lambda L}{2C} + (k_i + k_o)\right)^{-1} \frac{A}{2gH_L} \tag{3}$$

where A is the area of wear ring gap, H_L is the differential head across wear ring gap, and C_d is the discharge coefficient. The values k_i and k_o represent the loss coefficients for a sudden contraction and sudden expansion, then their sum is 1.5. Comparing the results of this correlation with the CFD solution, an underestimation of CFD on the leakage can be noticed in Figure 12. A substantial point of uncertainty on the application of this correlation is the wear ring differential head. As the CFD flow is not one-dimensional, the pressure gap to use for the correlation can not be univocally determined due to the complexity of pressure field pattern at the annular gap inlet and outlet. The same underestimation of CFD can be found varying the shaft installation position at fixed shroud differential head. Then, other aspects like inlet velocity swirl profile, surface actual roughness or turbulence modelling are mainly suspected to be the responsible of the CFD underprediction. Nevertheless, the trend of the CFD curves is the same as the correlations, confirming the reliability of the CFD in the general flow physics modelling.

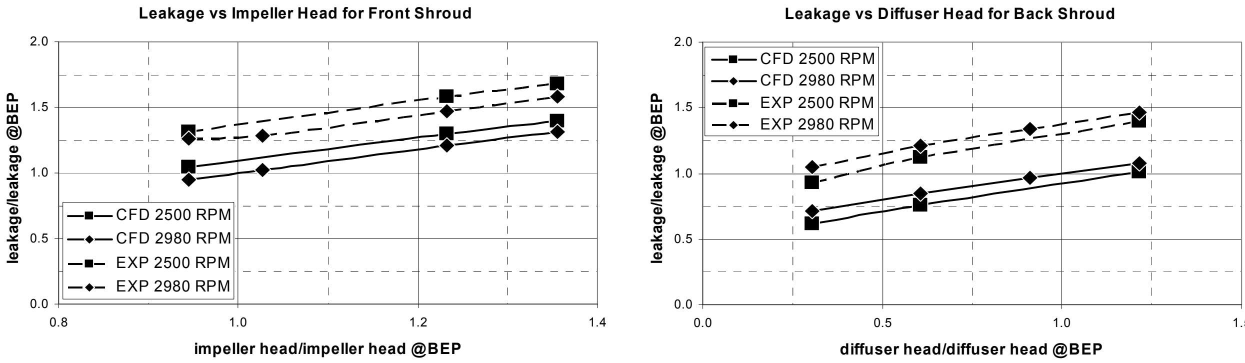


Figure 12 Comparison between CFD and experimental leakage flow values

The power losses on the shaft due to pressure work in the impeller and diffuser side chambers are calculated from CFD and compared with experimental correlations taken from literature (Nece and Daily (1960) and Gülich (2003)). The power losses from CFD are estimated as follows:

$$W_{LOSS} = M \cdot \omega \tag{3}$$

where M is the moment of pressure forces on the impeller side wall. The correlation by Nece and Daily is written for turbulent, separated boundary layers and considers some shroud geometrical parameters but zero leakage flow. The two correlations by Gülich enhance the Nece and Daily one, introducing the effect of leakage flow and also of other geometric details of the impeller shroud. In Figure 13 the data have been compared. The numerically evaluated power loss value is almost two times that of the correlations. Besides, as the correlation becomes more comprehensive taking into accounts more geometric details and flow characteristics, it moves toward CFD solution. Then, as underlined for the leakage flow evaluation, it is possible that the numerical simulation reproduces particular features of the actual geometry and then the correlative and numerical data are not really comparable. An experimental and a numerical campaign performed on the same geometry could provide the necessary information to understand whether the numerical estimation of power losses in cavity flows is more precise than general correlations.

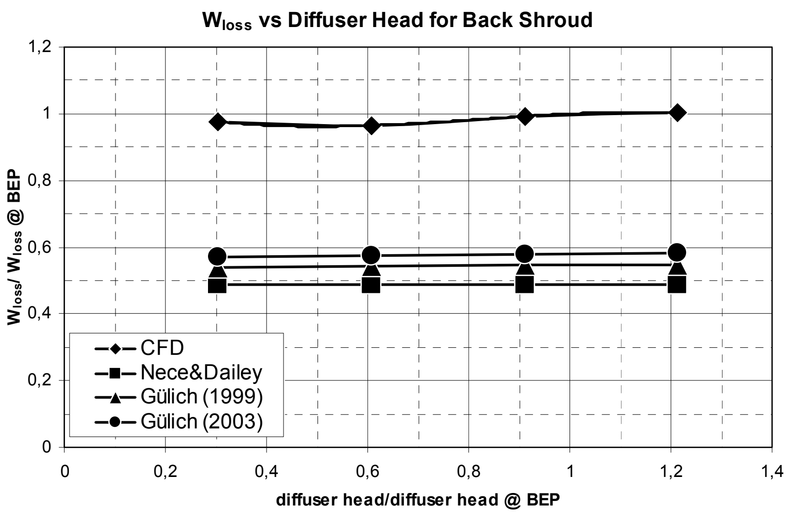
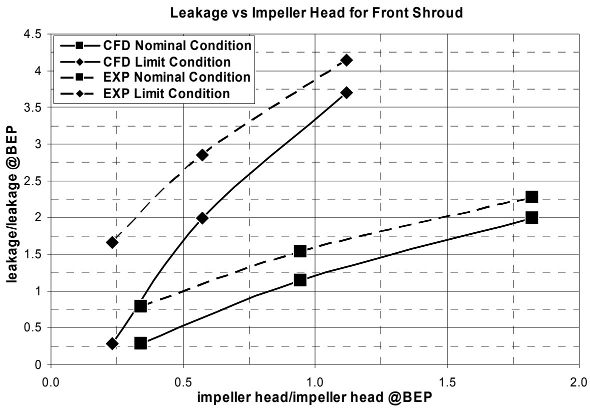


Figure 13 Power losses of front shroud: CFD vs correlations

Evaluation of the Limit Conditions

The effect of the machine detriment due to its utilization is hereafter analyzed. During the pump lifetime the materials can wear, thus inducing an increase of the radial clearance of the wear rings that deteriorates their performances. According to API 610/8 the pump elements as the thrust bearings should be designed to balance axial thrust at the design clearance and at two times as much design internal clearance. Thus it has been assumed worn wear rings with a double clearance size of the design one. With the worn wear ring the leakage flow increases compared to the design condition and it grows more rapidly with head as shown in Figure 14. No appreciable changes have been evidenced for the thrust, and then the pressure distribution in the cavity does not seem to be influenced by the wear. The leakage flow calculated by CFD is compared to the data obtained using correlations proposed by Stirling (1982). It can be underlined that the trend of the curves is the same and shows the effective worsening of wear rings performances. As a general observation, the CFD simulations seem to reproduce correctly the main physics and then the curve shape. Nevertheless, the differences between the leakage flow rates obtained with experimental correlations and CFD suggest that some aspects must be investigated. It must be pointed out that correlations are used as a general law for several cavity geometries while the CFD simulation provides results for the actual shape of the secondary channels and should be more accurate than the correlations, at least from a geometrical point of view. The gap between the CFD and correlation values will be further analyzed in the following.



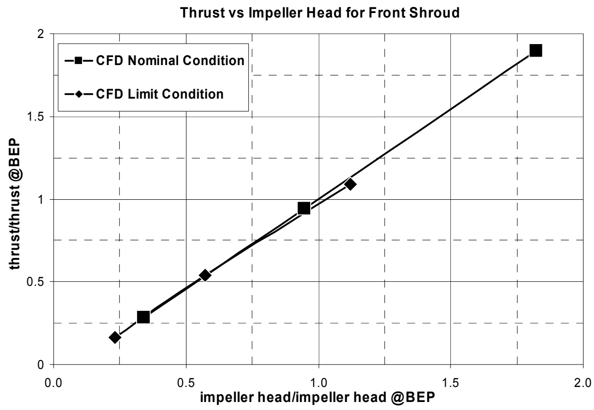
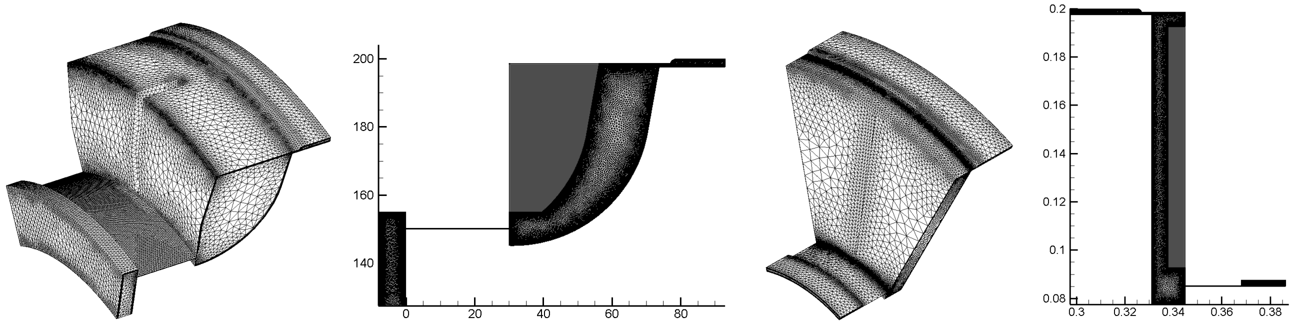


Figure 14 Leakage flow and axial thrust for the front shroud for new and worn wear ring

Possible Improvements to the Baseline Configurations

Some changes in the cavity design have been investigated, aiming to a possible improvement in the wear ring performances with a positive effect on axial thrust. Based on the previous analyses on standard configurations, two modifications have been proposed and tested for both impeller side chambers. The first modification consists on the installation of radial winglets inside the impeller side chambers as shown in Figure 15a and Figure 15c. The winglets number has been chosen as to avoid mechanical instabilities. In the second configurations, called "plenum", the protrusion has the same shape that have been used for the winglets (Figure 15b and Figure 15d) but is extended all along the circumference to reduce the chamber volume and the axial gap. Such modifications lead up to different behavior in the front and in the back shroud, thus bringing to different effects relative to their improvement on wear ring performances. The aim is to reduce the leakage flow rate and the losses, and to guide the residual axial thrust toward a defined condition. Considering the front shroud (Figure 16), the geometry with plenum shows a reduction of the leakage flow. This could be due to the constriction of the stagnation chamber that closes the passage to the flow. Instead, the modification with winglets does not affect significantly the leakage flow. The difference in axial thrust between these three configurations of the front shroud is lower than expected if compared with the effect of the plenum on the leakage flow.



(a) (b) (c) (d)

Figure 15 Geometry and mesh of the front and back shroud with winglets installation

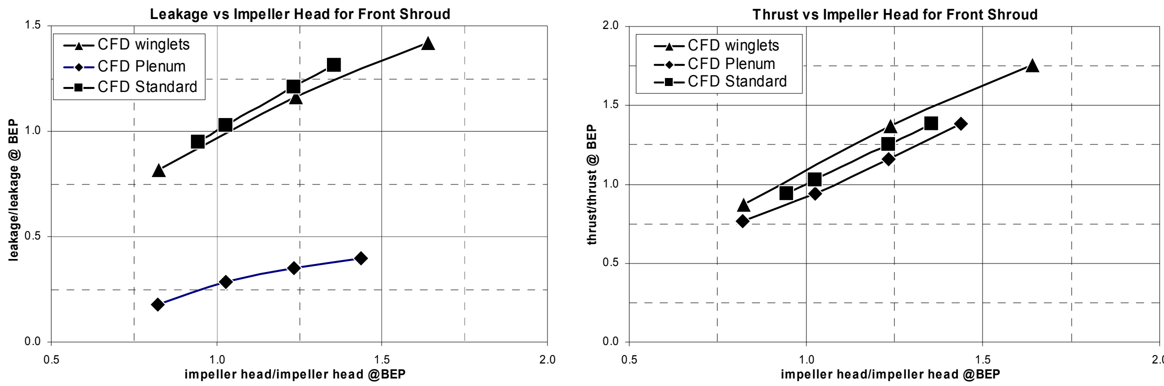


Figure 16 Leakage and Thrust for standard front shroud, winglets and plenum modification

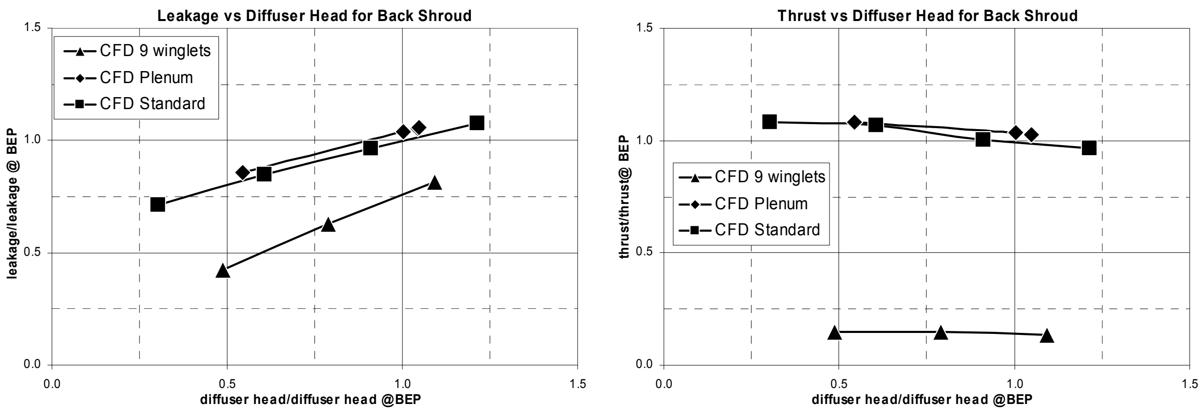
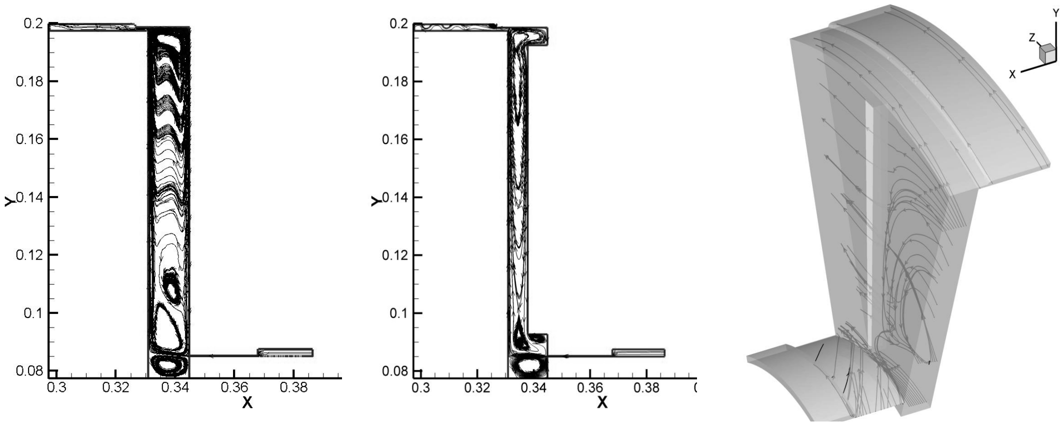


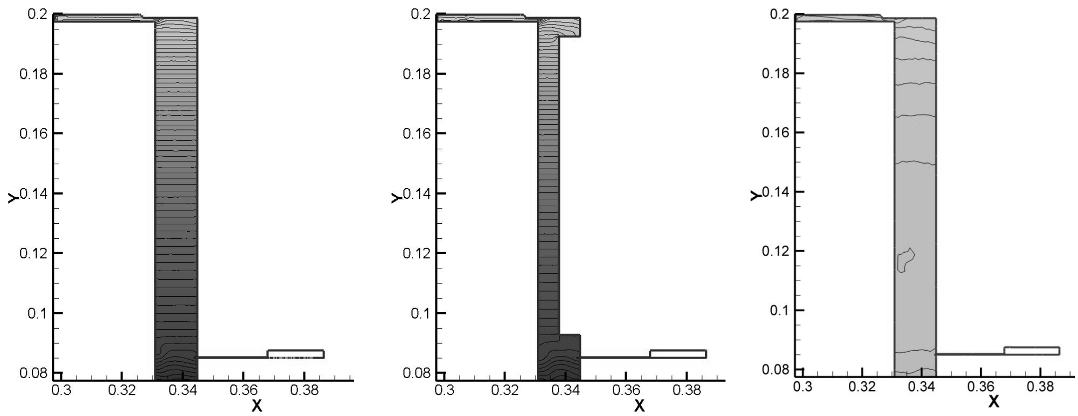
Figure 17 Leakage and Thrust for standard back shroud, winglets and plenum modification

The addition of these elements in the back shroud brings to different effects than in the front shroud. As shown in Figure 17, the configuration with winglets has a lower flow rate and a much lower load if compared with the other geometries. The presence of radial winglets inhibits the tangential flow and causes circumferential recirculation with the flow going also opposite to the rotation speed, as shown in Figure 18. Then, the pumping effect is reduced as well as the radial component and the flow rate. This effect is not so strong in the front shroud configuration because in that case the protrusion leaves a bigger free zone in the chamber and most of the fluid is not interested to this vorticity normal to axial direction. Therefore, the pressure field in the front shroud cavity is not affected significantly as well as the thrust value. For the back shroud the flow is more affected by the winglets and the pressure field results to be uniformly at a lower intensity (Figure 19). The plenum configuration instead does not change very much the flow pattern, since there is no circumferential opposition to the flow, but only a restricted axial gap. The better modified design should thus be chosen relative to each component, bringing positive effects on leakage flow and axial thrust.



(a) Standard **(b) With Plenum** **(c) With Winglets**

Figure 18 Streamlines in the back shroud



(a) Standard **(b) With Plenum** **(c) With Winglets**

Figure 19 Pressure field in the back shroud

Global Effects on the Axial Thrust

The residual axial thrust on the pump shaft has been evaluated. The “Standard” (STD) configuration has standard cavities; the “Limit” case show the detrimental effects on the global thrust and the “Plenum” case investigates the results obtained when using the cavity equipped with plenums. The “Winglet” configuration has winglet on the back shroud of the first stage and on the front shroud of the third, fourth and fifth bench. The positioning effects have not been considered according to the negligible effects shown in Figure 9. All the cases are evaluated at the nominal rotational velocity and the same central balancing drum. The obtained curves are non-dimensional with respect to the results obtained for the standard configuration at the BEP condition. The load is considered to be positive when directed from the second to the first bench (Figure 1a).

In the standard configuration the shaft is subject to negative thrust whose intensity increases during the pump working period, as proved by the “limit” curve. The introduction of plenums in the cavities contributes to weaken the load entity and can improve the bearings working conditions. It can be observed that, even though the evaluation of the residual thrust is clearly a non linear problem, the curves maintain the same shape and are simply shifted to different intensity levels. This result could suggest that the behaviour of the residual thrust is mainly governed by the pump design and the stage number and positioning, while the cavity flows affect the intensity only. This is partially confirmed by the results obtained considering the “winglet” configuration. This particular combination of cavities allowed changing the thrust direction and intensity, leading the shaft toward a traction condition. This result is quite important since the use of a numerical approach allowed to predict and govern the thrust intensity and direction and then to chose the best solution for cavity geometries and bearings type.

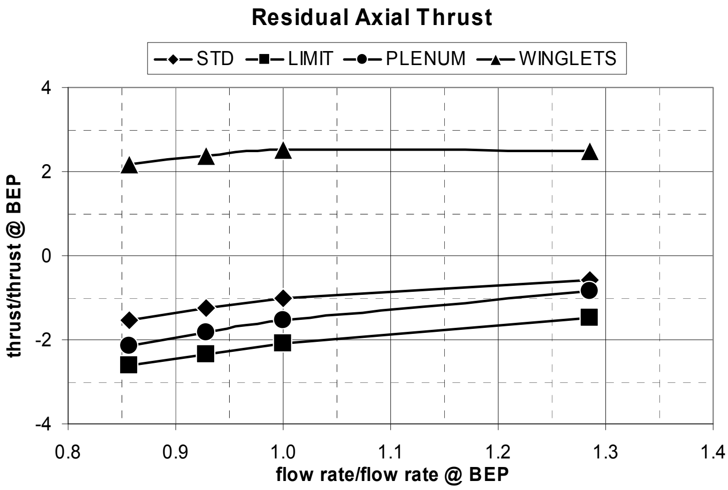


Figure 20 Residual axial thrust on the pump shaft

CONCLUSIONS

A numerical campaign for the study of cavity flows has been performed. In the following list the main results are reported:

- A good discretization of the fluid domain is necessary to obtain the correct evaluation of flow rates and pressure distribution.
- The shaft positioning does not seem to affect the cavity flow performances.
- The CFD simulation is able to reproduce the shaft positioning, the rotational velocity and the detrimental effects.
- The comparison between CFD and correlations suggest that the main physics are well captured by the numerical simulations, while a direct comparison of the flow rate is difficult for the modeling approximations and the general nature of the correlations.
- The numerical simulation of the pump components can be used to study the residual axial thrust and design solutions for better pump working conditions.

ACKNOWLEDGEMENTS

The authors are grateful to Prof. Eng. F. Martelli, Eng. Paolo Adami and Eng. Stefania Della Gatta of University of Florence, Eng. G. Marenco and Eng. D. Maestri of Weir Gabbioneta Srl for their valuable suggestions during the development of this work.

REFERENCES

[1] Adami, P., Della Gatta, S., Martelli, F., Bertolazzi, L., Maestri, D., Marenco, G., Piva, A. (2005), *Multistage Turbo-Pumps: Assessment of a Mixing Plane Method for CFD Analysis*, Proc. 60° ATI Conference, Roma, Italy, ATI60-12-1.

[2] Baskharone, E. A., Wyman, N. J. (1999), *Primary/Leakage Flow Interaction in a Pump Stage*, ASME Journal of Fluids Engineering, **121**, pp. 133-138.

[3] Daily, J.W., Nece, R.E., (1960), *Chamber dimension effects on induced flow and frictional resistance of enclosed rotating discs*, J. Basic.Eng, **82**, pp. 217-232.

[4] Debuchy, R., Dyment, A., Muhe, H., Micheau, P., (1998), *Radial inflow between a rotating and a stationary disc*, Eur. J. Mech. B/Fluids, **17**, N°6, pp. 791-810.

[5] Della Gatta, S., Salvadori, S., Adami, P., Bertolazzi, L., (2006), *CFD Study for Assessment of Axial Thrust Balance in Centrifugal Multistage Pumps*, Proceedings of 13th International Conference on Fluid Flow Technologies CMFF'06, September 6-9, Budapest, Hungary

[6] Gantar, M., Florjancic, D., Sirok, B., (2002), *Hydraulic Axial Thrust in multistage Pumps-Origins and Solutions*, ASME J Fluids Engineering, **124**, pp. 336-341.

[7] Gülich, J. F., (1999), *Kreiselpumpen. Ein Handbuch für Entwicklung, Anlagenplanung und Betrieb*, Springer Berlin

[8] Gülich, J. F., (2003), *Disc friction losses of closed turbomachine impellers*, Forschung im Ingenieurwesen 68

[9] Gülich, J.F., (2003), *Effect of Reynolds Number and Surface Roughness on the Efficiency of Centrifugal Pumps*, ASME J. Fluids Engineering, **125**, pp. 670-679

[10] Lobanoff, V.S., Ross, R.R., (1985), *Centrifugal Pumps Design and Application*, Gulf Publishing Company, Houston, Texas

[11] Nece, R. E., Daily, J. W., (1960), *Roughness effects on frictional resistance of enclosed rotating discs*, ASME J. Basic Engineering, **82**, pp. 553–562

[12] Salvadori, S., Della Gatta, S., Adami, P., Bertolazzi, L., (2007), *Development of a CFD Procedure for the Axial Thrust Evaluation in Multistage Centrifugal Pumps*, Proceedings of ETC7, European Turbomachinery Conference, March 5-9, Athens, Greece

[13] Stirling, T.E., (1982), *Analysis of the Design of two Pumps using NEL Methods*, Centrifugal Pumps Hydraulic Design, I Mech E Conference publications 1982 – 11

[14] Storteig, E., (1999), *Dynamic Characteristics and Leakage Performance of Liquid Annular Seals in Centrifugal Pumps*, Thesis submitted at the Department of Marine Engineering, NTNU, Norway

[15] Tamm, A., Stoffel, B., (2002), *The Influences of Gap Clearance and Surface Roughness on Leakage Loss and Disc Friction of Centrifugal Pumps*, Proceedings of ASME FEDSM´02, ASME Fluids Engineering Division Summer Meeting. Montreal, Canada, on July 14 – 18, 2002

[16] Utz, C., (1972), *Experimentelle Untersuchung der Stromungveluste in einer mehrstufigen Axialturbine*, Mitt.Ins.Therm.Turbomasch. ETH Zurich, Nr.19.