Development of a CFD Procedure for the Axial Thrust Evaluation in Multistage Centrifugal Pumps

DEVELOPMENT OF A CFD PROCEDURE FOR THE AXIAL THRUST EVALUATION IN MULTISTAGE CENTRIFUGAL PUMPS

S. Salvadori[*] – *S. Della Gatta*[*] – *P. Adami*[*] – *L. Bertolazzi*[+]

[*]Department of Energetic "S. Stecco", University of Firenze,
via di S. Marta, 3 50139 Firenze, Italy
Simone.Salvadori@unifi.it, Stefania.DellaGatta@unifi.it, Paolo.Adami@unifi.it

[+]WEIR Gabbioneta Srl., v.le Casiraghi, 68 20099 Sesto S. Giovanni (Mi), Italy
Laura.Bertolazzi@weirgabbioneta.com

ABSTRACT

One of the most challenging aspects in horizontal pumps design is represented by the evaluation of the axial thrust acting on the rotating shaft. The thrust is affected by pump characteristics, working conditions and internal pressure fields. Solving this problem is simple for single stage pumps while several complications arise for multistage pumps even in partially self-balancing opposite impeller configuration. Therefore a systematic approach to the axial thrust evaluation for a multistage horizontal centrifugal pump has been assessed and validated. The method consists in CFD simulation of each single pump component to obtain correlations which express the axial thrust as a function of the working conditions. The global axial thrust is finally calculated as balance of the forces acting on each stage. The numerical procedure will be explained and its main results shown and discussed in the present paper.

NOMENCLATURE

A	[m^2]	Passage area
BEP		Best Efficiency Point (pump design point)
CFD		Computational Fluid Dynamic
F	[N]	Force
H	[m]	Head
$\vec{n}$	[-]	Normal vector (directed axially)
p	[Pa]	Pressure
Q	[m^3/s]	Flow rate
r	[m]	Radial coordinate
T	[N]	Thrust

Subscripts and Superscripts

ax	Axial
bs	Back shroud
bus	Bushing
cav	Cavity
fs	Front shroud
inlet	Impeller inlet section
mom	Momentum
ref	Reference

INTRODUCTION

In a multistage horizontal centrifugal pump, the residual axial load should be balanced by thrust bearings which can guarantee the mechanical reliability of whole pump if properly chosen. Over the whole pump operating range the main contribution to the axial thrust is due to the impellers flow fields, but also to the presence of leakage flows through wear rings, balancing drums and inside the gaps between impeller shrouds and pump stationary walls.

The numerical simulation of each single pump component has been chosen to solve the so complex problem of axial thrust evaluation. The single components contribution, which have to be calculated taking into account the resulting pressure distribution on its rotating walls, have then to be matched together. Furthermore a deep study of the fluid dynamic fields inside the single pump components can help to reach a higher knowledge of the pump characteristics already during its design. As a consequence, several geometrical modifications could be suggested to improve head or efficiency values and to minimize the residual axial thrust**.** Present paper describes the CFD investigation of a multistage centrifugal pump in opposite impeller configuration designed and produced by Weir Gabbioneta Srl. and whose section is reported in Figure 1.

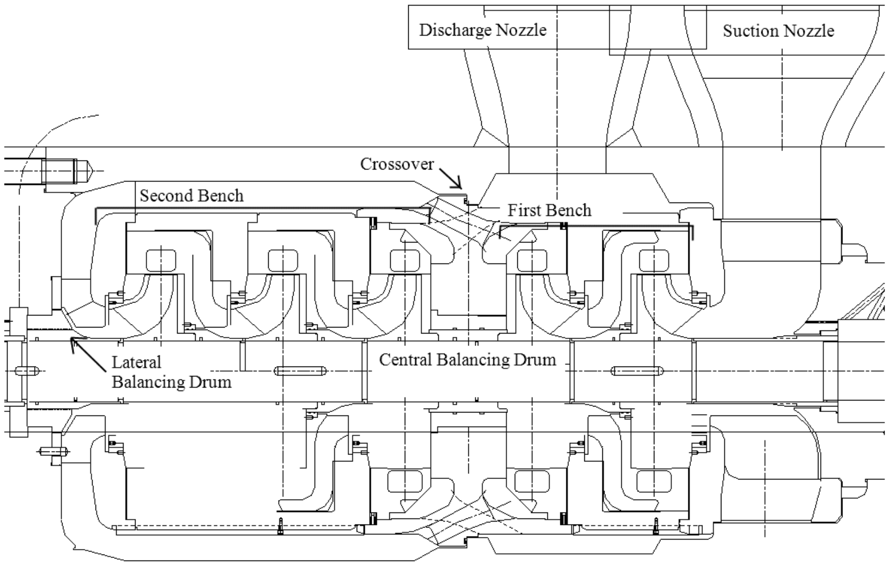


Figure 1: Multistage centrifugal pump by Weir Gabbioneta Srl.

THRUST EVALUATION IN MULTISTAGE PUMPS

The evaluation of the resultant of the forces acting on each single stage is the starting point to determine the residual pump axial thrust. With reference to the schematic distribution of forces and the control volume shown in Figure 2, this balance can be expressed as follows:

$$\vec{F}_{fs} + \vec{F}_{bs} + \vec{F}_{inlet} + \vec{F}_{mom} + \vec{F}_{bus} = \vec{T}_{ax} \qquad (1)$$

Equation 1 is referred to a control volume containing all the rotating walls; F_{fs} is the global force acting on the front shroud walls while F_{bs} is relative to the back shroud, F_{inlet} is due to the pressure field at the impeller inlet section, F_{mom} is the momentum contribution in axial direction and F_{bus} is the pressure integral on the bushing walls. All these terms can be elaborated considering pressure values, pump geometry and mass flow conditions:

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

The possibility of evaluating internal runner forces directly from the CFD program has not been taken into account because it requires the integration of viscous terms, while the calculation of the momentum flux on plane surfaces is easier and less prone to inaccuracies.

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.

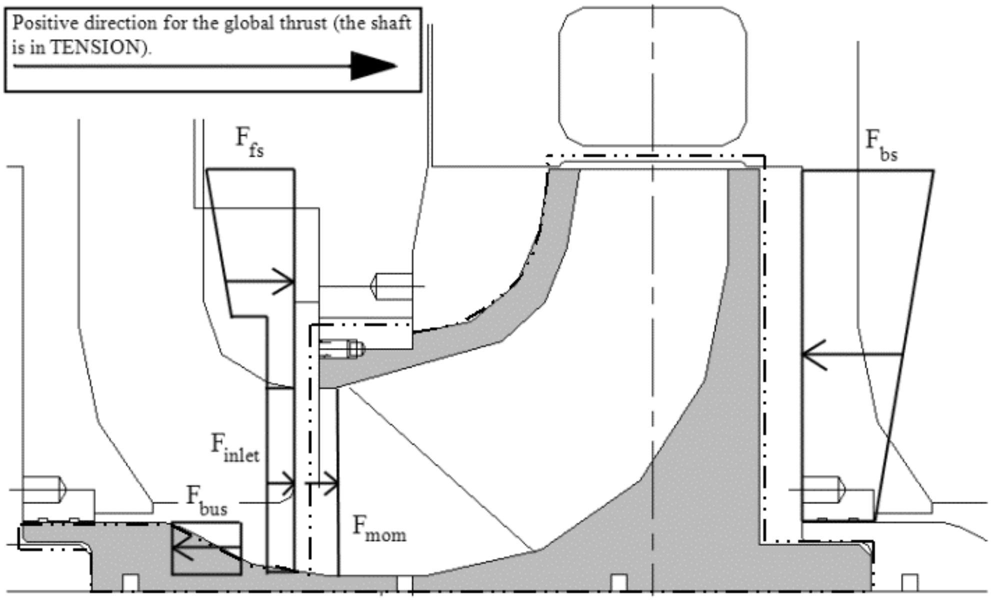


Figure 2: Forces balance for a single stage. The dashed line indicates the control volume

In the pump shown in Figure 1, the flow enters the suction nozzle and passes through the two stages of the first bench. Then a crossover device leads it to the third stage inlet through an annular chamber, thus being responsible for its turning of direction. Then the fluid evolves in the three stages of the second bench and finally reaches the discharge nozzle. All the diffusers are characterized by the same geometry except for the ones facing the crossover while all the impellers are geometrically identical, but have different inlet conditions due to the presence of different static upstream components. The opposite impeller configuration helps to balance the axial thrust, but does not guarantee it also in pumps with an even impellers number because of the effect of leakage flows and shroud-stationary walls gaps.

To properly choose the thrust bearings, the residual axial load intensity and direction must be correctly evaluated taking into account the contribution of the impeller shroud chambers as well. The leakage flows modify the pumped flow rate (and consequently the total head) and, together with the gaps, affect the pressure distribution on the rotating walls. Figure 3 shows the main and leakage flows in the front and back shroud chambers and in the central balancing drum. While the flow inside the front shroud chamber always turns inward in radial direction, the one inside the back shroud cavity turns according to the local pressure gradient. The second and the fifth impeller back shroud cavities correspond to the central drum ones in which the leakage flow goes from the second to the first group of impellers. A lateral balancing drum is located before the first impeller of the second bench and is connected to the suction volute by means of a balancing duct thus keeping the stuffing box pressure very close to the suction pressure.

To evaluate the residual axial thrust for this geometry a specific but flexible procedure has been developed. It consists in the following main steps:

- the single stage analysis (both impeller and diffuser hydraulic channels);

- the simulation of other stationary components (volutes, crossover);
- the study of the flow conditions inside the front and back shroud impeller side chambers;
- the study of the flow conditions inside the central and lateral balancing drums;
- the final data collection for the residual hydraulic axial thrust calculation.

The computational strategy which has been developed for each single issue will be described and the results of its application to the investigated pump will be presented and discussed here below.

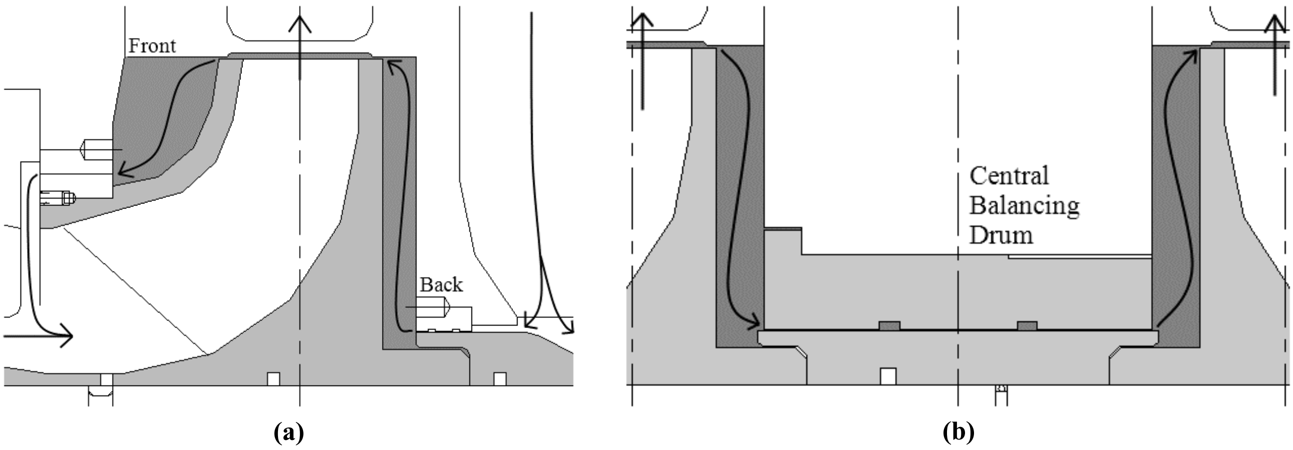


Figure 3: Impeller side chambers (a) and central balancing drum (b) main and leakage flows

CFD PROCEDURE DESCRIPTION

Several 2D and 3D steady simulations solving Reynolds Averaged Navier-Stokes (RANS) equations have been carried out. A finite-volume pressure-correction procedure for incompressible flows has been employed with a two equation k-ε turbulence model and standard wall functions. The pump has been divided in its main components which have been analyzed separately to reduce computational costs. The following elements or group of elements have been simulated:

- suction volute;
- first impeller;
- each diffuser not facing the crossover device coupled with its downstream stage impeller;
- second diffuser together to crossover device and annular chamber;
- third impeller;
- fifth diffuser and discharge volute;
- impeller side chambers (front and back shroud), central and lateral balancing drums.

The impeller shroud chambers and the balancing drums have been analyzed with 2D axial symmetric CFD simulations apart from the main flow. This choice is supported by the results shown by Gantar et al. (2002) for a geometry similar to the present one. Usually a flow recirculation in the impeller exit area does not interest leakage flow rate probably due to the small clearance that separates the main impeller passages from its own shroud chambers.

The sequence of simulations has been chosen considering each result as a boundary condition for the following CFD analysis. For instance, the single impeller boundary conditions have been provided by its upstream stationary component simulation (both diffuser or annular chamber). Furthermore, to study the shroud chambers behaviour, the relation between the head across the impeller and the leakage flows should be known.

Stage Coupling Approach

The axial thrust on pump shaft is due to the forces balance on the impeller walls, however the correct pressure field evaluation requires also the stationary components simulation to provide the right boundary conditions for the rotating parts analysis. The presence of both stationary and rotating parts in each stage suggests a coupled approach for the diffuser-impeller interaction evaluation. A mixing plane technique has been applied in present study. Diffuser inlet and impeller outlet have been extended in the computational models to avoid errors induced by imposing the boundary conditions on an interface plane exactly corresponding to the real components inlet and outlet sections.

The mixing plane approach allows to solve separately the rotating and the stationary domains in steady conditions. Data from adjacent zones are tangentially averaged and then imposed as "mixed" boundary conditions at the interface. This approach removes any unsteadiness deriving from the circumferential variations in the interface plane, thus yielding a steady state result. However this method allows maintaining a non-uniform radial distribution of the conservative variables and then a swirled inlet velocity distribution. Since the impeller eye has a large inlet area this expedient allows a more realistic evaluation of the impeller performance. Furthermore, despite the simplifications, several applications demonstrate that the approximation of the time-averaged results is quite reasonable especially for the performance parameters evaluation (Adami et al. (2005)).

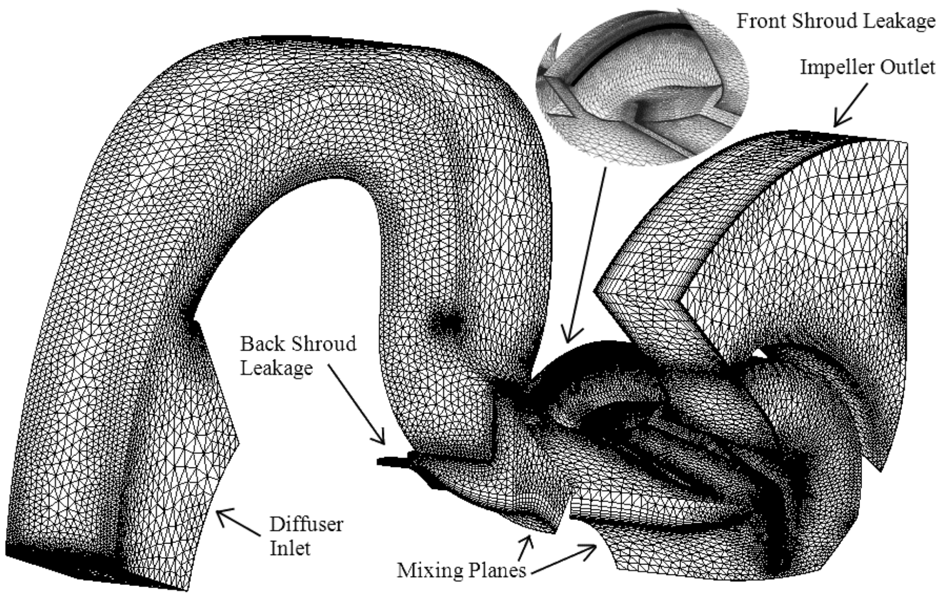


Figure 4: Computational grid for the steady stage simulation with mixing plane approach

As a further hypothesis the stage kinematics repeatability has been assumed. Therefore, the velocity profile at the impeller exit section has been employed to update the inlet boundary conditions of the diffuser model. The whole process is iterative and can be synthesized in the following main steps:

[1] imposition of the initial boundary conditions on the diffuser inlet (flow rate, velocity direction), on the impeller inlet (flow rate) and front shroud leakage inlet (flow rate);
[2] solution of the RANS equations in both domains;
[3] during the simulation, the CFD solver updates the conditions on the mixing plane averaging the flow field and the static pressure values in tangential direction;
[4] once the convergence has been reached, the boundary conditions are updated at the interfaces;
[5] the steps from [1] to [4] have to be repeated until convergence is achieved both for velocity profiles and pressure values.

The stage computational hybrid grid is reported in Figure 4. Inlet and outlet sections of the model are evidenced as well as the mixing planes. The mean values for the updating of the boundary conditions on the diffuser inlet are obtained on the real interface. Then the velocity vectors are scaled assuming a free vortex distribution and finally imposed at the diffuser inlet.

The leakage flow passing through the back shroud chamber is negligible with respect to the main flow rate (about 0.5% in the range of interest), while the flow passing in the front shroud cavities must be considered. Therefore the impeller geometric domain has been modeled including the stage interface with front shroud chamber, while the leakage flow in the back shroud has been neglected. The numerical study of the impeller shroud chambers has been carried on before the stage analysis, assessing a correlation between the impeller head and the front shroud leakage flow to update the boundary conditions.

Furthermore, an iterative cycle on the leakage flow rate has to be performed because the impeller head decreases when the main flow rate increase, while the leakage flow increases with the impeller head. The CFD procedure has also been repeated for different capacities. Stage efficiency and characteristic curve have been obtained and their dimensionless version is shown in Figure 5. The reference value corresponds to the result obtained at the design point.

The head has been defined as:

$$H = \frac{p_{outlet} - p_{inlet}}{\rho g} \tag{3}$$

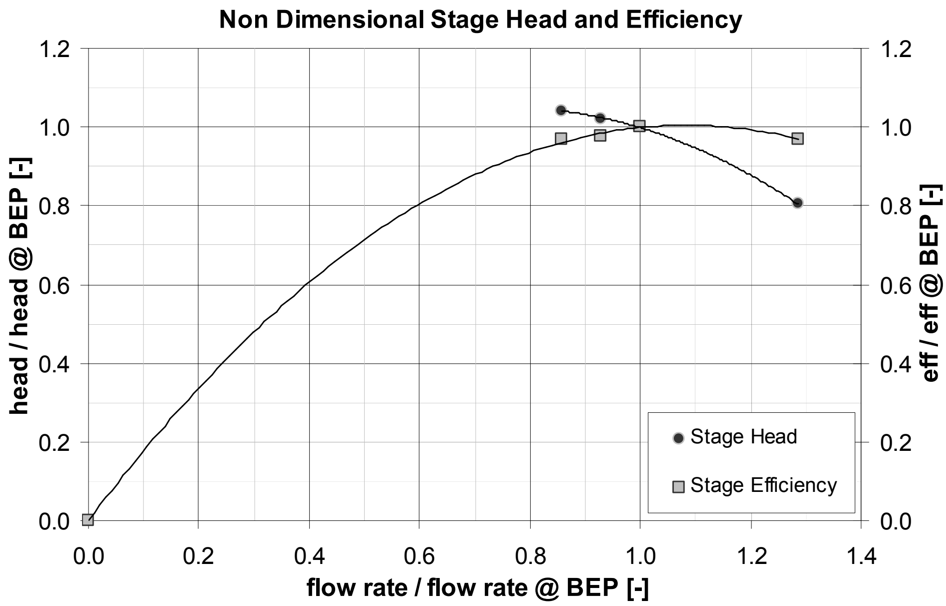
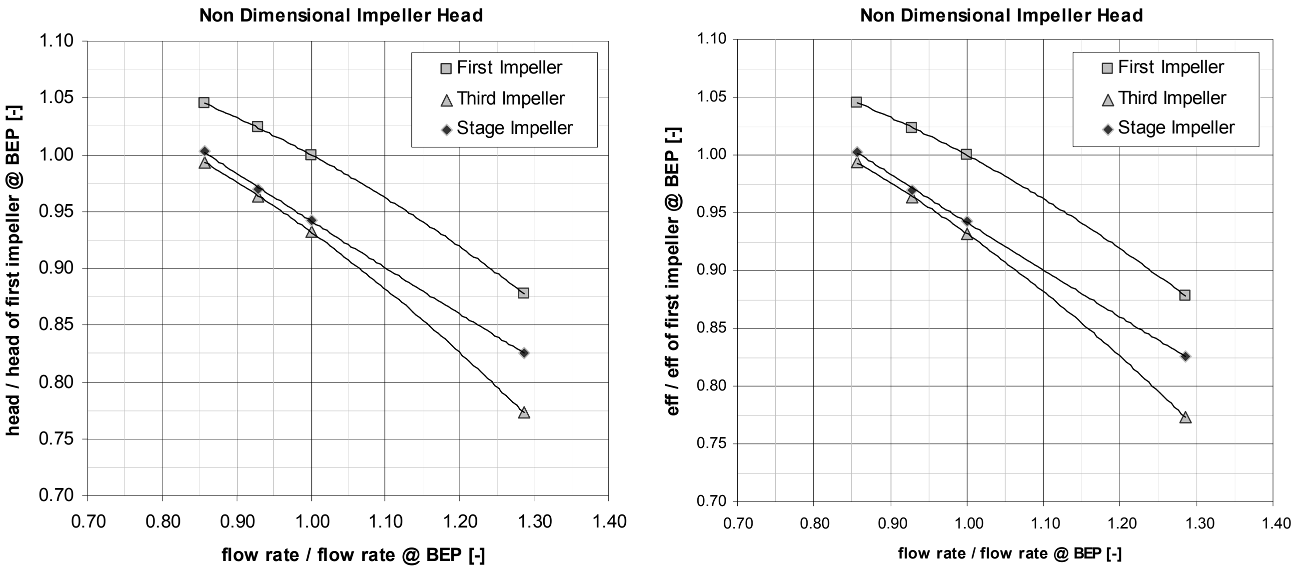


Figure 5: Non dimensional stage head and efficiency

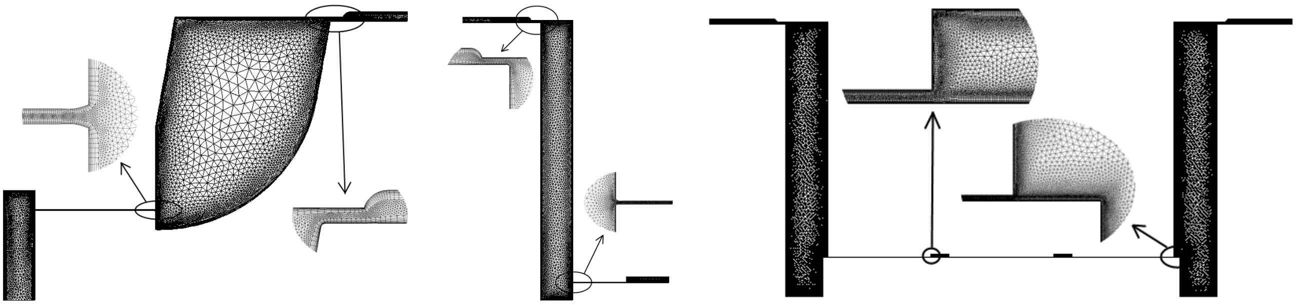
As observed for the stage simulation, the impeller performances strongly depend on the inlet conditions. To simulate the first and the third impeller, the suction volute and the crossover device have been studied by a stand alone approach as well. For these components, correlations between the evolving flow rate and the pressure variation have been obtained. Furthermore, velocity distributions of the suction volute and crossover device have been tangentially averaged at the interface plane and then imposed on the downstream impeller inlet. This method is similar to mixing plane one, but neglects both the steady and unsteady interactions.

Impellers performance and efficiency curves have been plotted (Figure 6). The shown results are non dimensional with respect to the value obtained for the first impeller at the pump design point. The first impeller shows the highest head for every flow rate probably due to the fact that the flow enters the first impeller axially, that is at the design conditions. Instead, the third impeller has lower values of head which get further and further from the first impeller ones since the flow rate grows. In fact, the flow entering the third impeller has not been straightened by the diffuser but directly comes from the annular chamber where only structural elements are present. Finally the other impellers head is a linear function of the flow rate and has halfway values between the first and the third ones. The efficiency curves confirm good performances for the first impeller and less good values for the third one. However the third impeller efficiency is higher than expected at low flow rates probably due to the leakage flow effect. The results confirm an high dependence of impeller performance on its inlet flow field distribution.



(a) (b)

Figure 6: Dimensionless head (a) and efficiency (b) for the simulated impellers



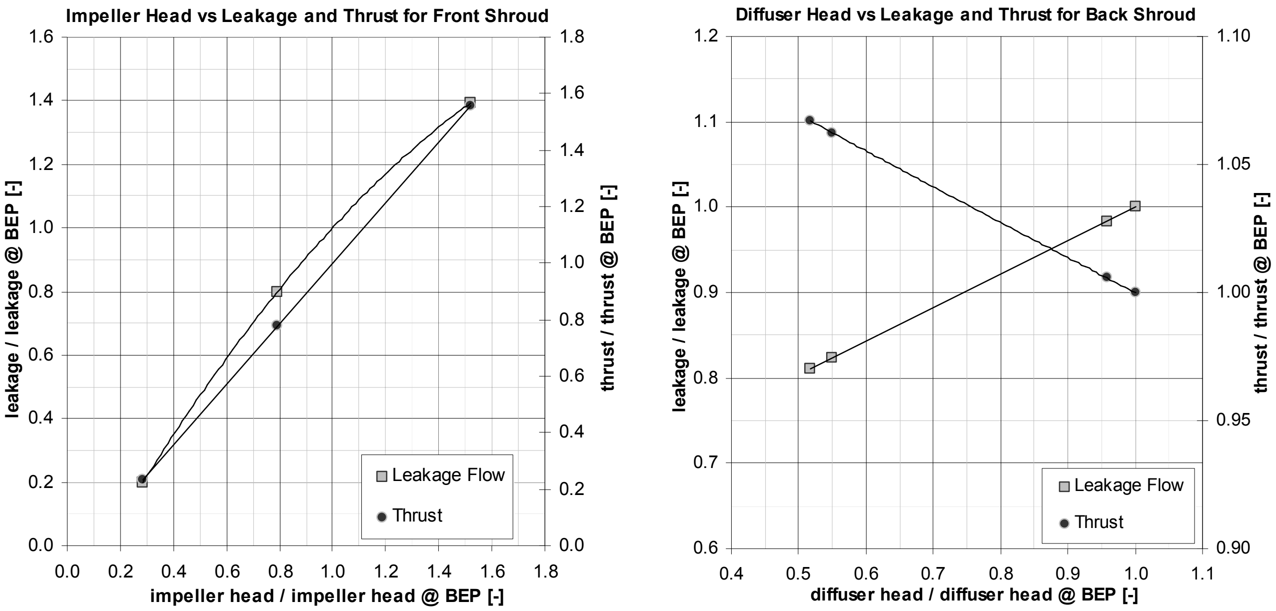
(a) (b) (c)

Figure 7: Front shroud (a), back shroud (b) and central drum (c) computational hybrid grids

Impeller Shroud Chambers and Balancing Drums CFD Analysis

Impeller shroud chambers have been studied by means of a 2D CFD approach. The hypothesis of axial symmetry has been reasonably assumed for this problem. The flow inside the cavity moves outward in radial direction at the rotating wall and inwards at the casing wall. To get an exhaustive leakage characterization, the CFD analysis has been performed for different heads across the cavity calculating the corresponding leakage mass flow and the contribution to the axial thrust. The results have been employed to assess some correlations between the head across the chambers, the leakage flow rate and the axial thrust on the rotating walls (Figure 8 and Figure 9). The obtained equations are consistent with the well known correlations from the literature (Denny (1954), Traupel (1958), Wortster and Thorne (1959), Utz (1972) and Della Gatta et al. (2006)). Cavity flows are therefore solved at different operating points in a preliminary phase and do not require to be calculated at every update of the stage coupling boundary conditions.

The rotating velocity inside the chambers depends on peripheral conditions, namely on the impeller rotating speed and on the head across the cavity itself. Due to pump geometry (Figure 2 and Figure 3), head across the front shroud cavity corresponds to impeller head, while the leakage flow in the back shroud chamber depends on the pressure rise in the diffuser. Therefore, leakage mass flow and its contribution to the axial thrust depend on the pump working conditions.



(a) **(b)**

Figure 8: Leakage flow and thrust as a function of impeller head for front and back shroud

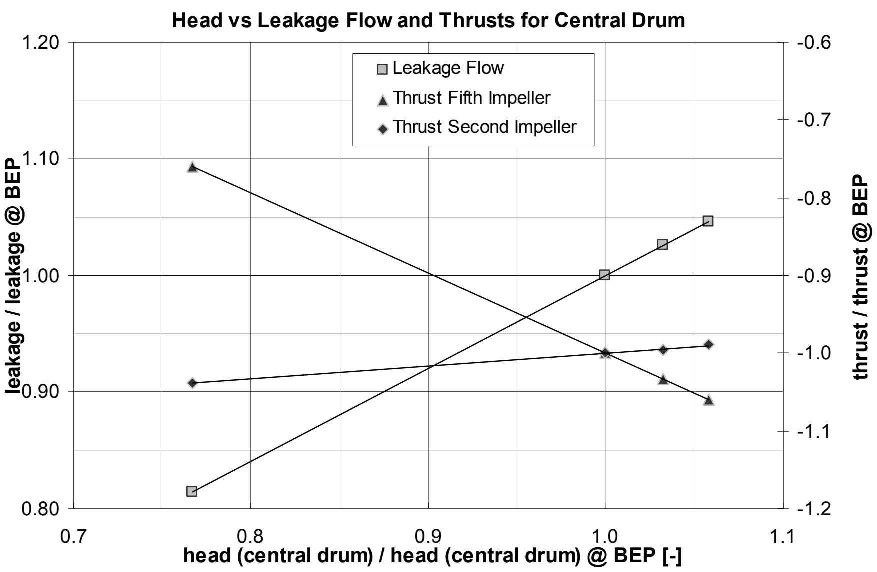


Figure 9: Leakage flow and thrust as a function of the head across the central drum

Results from the front shroud chamber analysis are discussed here below and details of the computational grid are shown in Figure 7a. Head across the front cavity results in a centripetal flow. The leakage mass flow and the contribution to the residual thrust value are obtained as a function of the head across the cavity (Figure 8a). The shown results are non dimensional with respect to the BEP values. It should be noticed that, despite of the complex geometry of the front shroud cavity, the axial thrust acting on the rotating wall is a linear function of the impeller head. In Figure 7b the back shroud impeller chamber grid is shown. Its leakage flow rate and its contribution to the residual axial thrust as a function of the diffuser head are visible in Figure 8b. Both leakage flow rate and axial load are a strictly linear function of the diffuser head.

As far as the central balancing drum is concerned, the head across the cavity depends on the working conditions in the second group of impellers and in the crossover device. The computational grid is reported in Figure 7c, while the obtained results are visible in Figure 9. All the correlations can be expressed as linear function of the head across the balancing drums. The two contributions to the thrust have opposite behaviours. The thrust on the second impeller is nearly constant in the range of interest while high variations on the fifth impeller can be observed. In fact, for a variation of the head of 40% the axial thrust almost decrease of the same percentage.

AXIAL THRUST: RESULTS AND DISCUSSION

To calculate the axial thrust acting on the whole pump, the contribution of every stage has been separately evaluated according to Equation 2. As regards the cavities, every contribution to the thrust obtained by the 2D CFD simulations must be added to a term due to the real corresponding reference pressure. Finally the contribution to the thrust due to the cavities (the F_{fs} and F_{bs} terms of Equation (1)) can be calculated as follows:

$$\int_A p_{cav}\,\vec{n}\,dA = \int_A p_{ref,cav}\,\vec{n}\,dA + \int_A \Delta p_{cav}\,\vec{n}\,dA = p_{ref,cav}\int_A \vec{n}\,dA + \int_A \Delta p_{cav}\,\vec{n}\,dA \qquad (4)$$

If directed from the second to the first bench (Figure 1 and Figure 2) a force is considered positive. Each stage contribution to the axial thrust for the design flow rate is reported in Figure 10, highlighting the opposite effects of impeller shroud chambers. The values are non dimensional with respect to the modulus of the result at BEP.

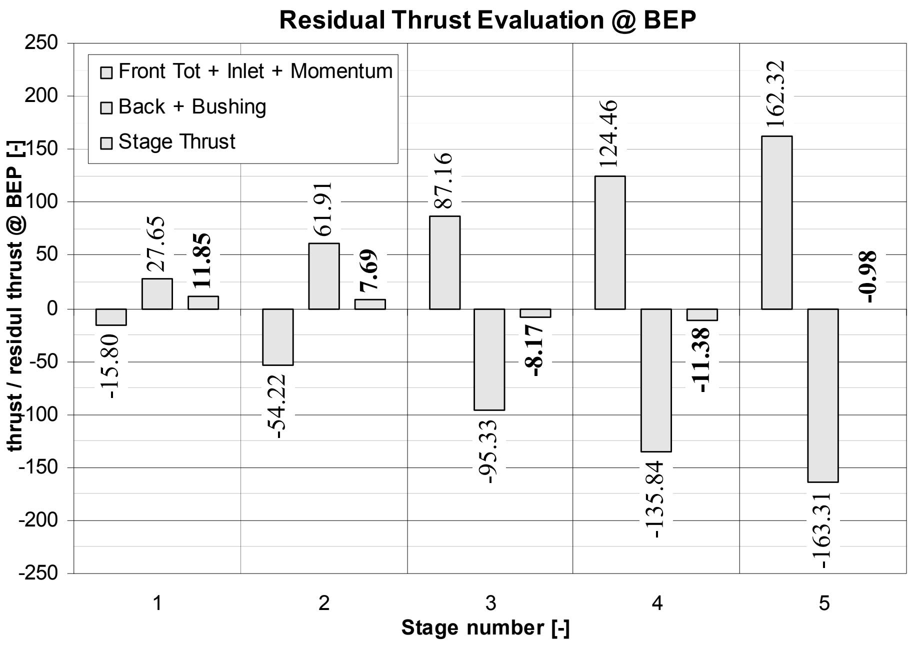


Figure 10: Discrete residual thrust evaluation at BEP flow rate

Some remarks on these results can be done. First of all, the impeller back shroud contributions are higher than the corresponding one from the impeller front shroud. Furthermore, the opposite configuration of two groups of stages is not enough to balance the global thrust. The residual thrust is negative in the chosen referring system: this means that the shaft is in compression.

For a proper design of the thrust bearings, it is still required to know the maximum thrust in pump operating range. Analysis have been carried out between the 85% and the 130% of the BEP flow rate as reported in Figure 11. The maximum thrust for the investigated pump in the considered range is obtained at the lowest studied mass flow corresponding to the highest value of pump head.

Presented results have been carried out with fix design values for clearances. However, during its life, the real pump will undergo a process of wear resulting in clearance enlargement and consequently increase of leakage flow, global performance worsening and residual axial thrust modifications as far as module and sometimes direction is concerned. As a consequence the axial thrust trend can be obtained even for the limit enlarged clearances of worn rings and drums.

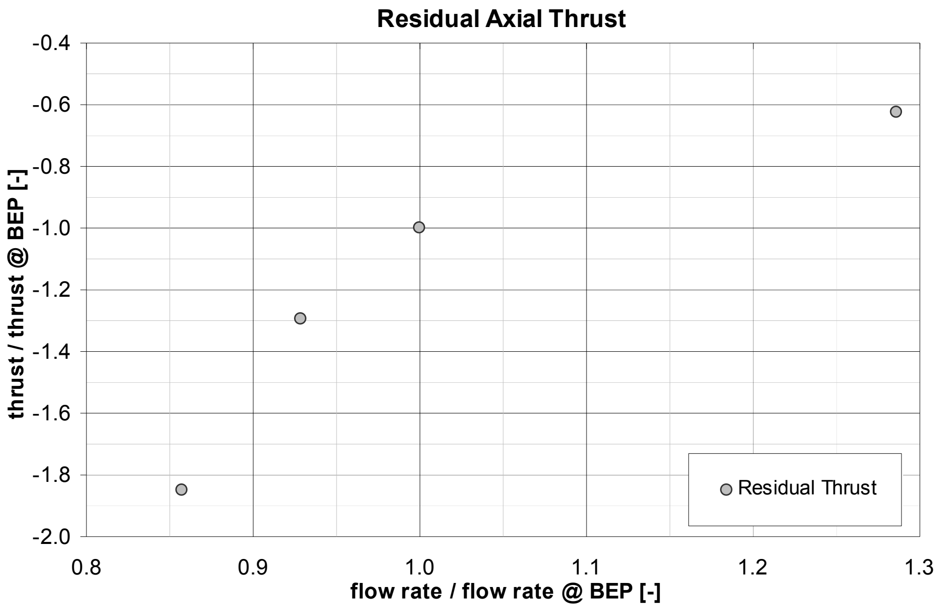


Figure 11: Residual axial thrust between 85% and 130% of BEP flow rates

In fact flexibility is the greatest advantage of the present method. Once the components have been characterized as a function of the flow rate (impeller and static components) or of the differential pressure (leakage flows), the resulting correlations coupled with the presented algorithm brings easily to the residual thrust value for any flow rate in the investigated range. Furthermore, the effects which can be obtained on the residual axial thrust by modification of a single kind of pump component, can be evaluated by means of the correlation which characterize the component itself.

However, since the residual thrust results from the difference of the pump components thrusts which are one order of magnitude larger than the residual one, the use of the procedure for bearing dimensioning requires a careful evaluation of the accuracy of the CFD simulation or an experimental validation of the procedure. In this case, some validations about the pump characteristics can be found in Adami et al. (2005), while an experimental campaign for pump seal pressure drop is being carried out.

CONCLUSIONS

A computational study of a horizontal multistage pump has been carried out, with the residual axial thrust evaluation as main target. The contribution of each single pump component to the axial load has been estimated by a CFD investigation of its internal flow and pressure field. Impellers and diffusers have been coupled and analyzed by applying a mixing plane approach, while the leakage flows through wear rings of shroud chambers and balancing drums have been simulated separately with 2D axial symmetric models. Then, all the computational results have been collected and a methodology to get the residual axial thrust of the whole pump has been developed.

The assessed method suggests a proper dimensioning of multistage pump thrust bearings which can guarantee the mechanical pump reliability. Moreover it helps to deeply understand pump sensitivity to each single component and parameter as far as axial thrust is concerned. Finally, the obtained results highlight the great importance of leakage flows in shroud chambers and balancing drums for a multistage pump axial balancing. Axial thrust values also depends on pump operating and wear conditions which can be taken into account by applying the described method.

ACKNOWLEDGEMENTS

The authors are grateful to Prof. Ing. F. Martelli of University of Firenze, Ing. G. Marenco, Ing. D. Maestri and Ing. A. Piva of Weir Gabbioneta Srl. for their valuable suggestions during the development of this work.

REFERENCES

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, ATI60-12-01

Della Gatta, S., Salvadori, S., Adami, P., Bertolazzi, L. (2006), *CFD Study for Assessment of Axial Thrust Balance in Centrifugal Multistage Pumps*, Proc. Conference on Modeling Fluid Flow CMFF'06, Budapest, Hungary

Denny, D.F. (1954), *Leakage Flow Through Centrifugal Pump Wear Rings*, TN460, Beds: BHRA, Cranfield

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

Traupel, W. (1958), *Thermische Turbomaschinen*, Springer-Verlag, Berlin

Utz, C. (1972), *Experimentelle Untersuchung der Strömungsverluste in einer mehrstufigen Axialturbine*, Mitt. Ins. Therm. Turbomasch. ETH Zurich, n° 19

Worster, R.C., Thorne, E.W. (1959), *Measurement of Leakage Flow Tough the Wearing Rings Of a Centrifugal Pump and its Effect on Overall Performance*, Beds: RR 619, Cranfield