DEVELOPMENT OF A ONE-DIMENSIONAL MODEL FOR THE PREDICTION OF LEAKAGE FLOWS IN REGENERATIVE PUMPS

Giulio Cantini[a], Simone Salvadori[a,1], Massimiliano Insinna[a,2], Giorgio Peroni[b], Gilles Simon[c], Duccio Griffini[b], Raffaele Squarcini[b]

[a]Department of Industrial Engineering, University of Florence, Florence, Italy
[b]Pierburg Pump Technology Italy S.p.A., Livorno, Italy
[c]Pierburg Pump Technology France S.p.A., Thionville, France

ABSTRACT

Regenerative pumps are characterized by a low specific speed that place them between rotary positive displacement pumps and purely radial centrifugal pumps. They are interesting for many industrial applications since, for a given flow rate and a specified head, they allow for a reduced size and can operate at a lower rotational speed with respect to purely radial pumps. The complexity of the flow within regenerative machines makes the theoretical performance estimation a challenging task. The prediction of the leakage flow rate between the rotating and the static disks is the one that more than others has an impact on the prediction of global performance. All the classical approaches to the disk clearance problem assume that there is no relevant circumferential pressure gradient. In the present case, the flow develops along the tangential direction and the pressure gradient is intrinsically non-zero. The aim of the present work is to develop a reliable approach for the prediction of leakage flows in regenerative pumps. The method assumes that the flow inside of the disk clearance can be decomposed into several stream-tubes. Energy balance is performed for each tube, thus generating a system that can be solved numerically. The new methodology has been tuned using data obtained from the numerical simulation of virtual prototypes of regenerative pumps where the disk clearance is part of the control volume. After that, the methodology has been integrated into an existing one-dimensional code called DART (developed at the University of Florence in cooperation with Pierburg Pump Technology Italy S.p.A.) and the new algorithm is verified using available experimental and numerical data. It is here demonstrated that an appropriate calibration of the leakage flow model allows for an improved reliability of the one-dimensional code.

KEYWORDS

Leakage Flow, Disk Clearance, Regenerative Pump, 1D Model, CFD

NOMENCLATURE

A, B	Points on the circumference that individuate a stream tube
d	Axial extension of the open channel
D_{eq}	Equivalent hydraulic diameter
f	Friction factor
h	Clearance of the disk-casing cavity

[1]Contact author: simone.salvadori@unifi.it
[2]Currently Centro Ricerche e Attività Industriali (CReAI), Pistoia, Italy

H	Impeller blade height
l	Perimeter
L	Generic distance between two points on the circumference
nd	Non-dimensional
N_{bl}	Number of impeller blades
p	Pressure
P	Point on a stream tube
Q	Flow-rate
r	Radius
Re	Reynolds number
s	Width of the disk-casing cavity
t	Impeller blade thickness
S	Passage area of the stream tube
U	Velocity contribution of relative disk-casing movement
W	Velocity
Greek	
α	Angle from stripper to point P
ϕ	Angle from stripper to point B
Φ	Flow coefficient
Ψ	Load coefficient
ρ	Density
θ	Angle from stripper to point A
ω	Rotating speed

INTRODUCTION

In the automotive field, secondary systems are often equipped with small turbomachines that elaborate small flow rates and guarantee a high pressure rise, thus positioning the machine in the low specific speed region. Considering that reduced weight and size are also necessary, regenerative pumps become competitive with respect to radial pumps. Regenerative pumps (also known as side channel pumps) are characterized by low specific speed values. If specific speed is calculated with n in $[rpm]$, Q in $\left[\frac{m^3}{s}\right]$ and H in $[m]$, typical values for regenerative pumps are between 2 and 11 as reported by Gülich (2010). These pumps combine the high pressure rise of positive displacement pumps with the flexible operation of centrifugal pumps. Although side channel pumps are characterized by a slightly lower peak efficiency with respect to centrifugal turbomachines, it must be underlined that in the optimal range of application they represent a compact solution that guarantees a stable performance close to the design point at a lower rotational speed with respect to centrifugal pumps. Regenerative pumps are characterized by the presence of an impeller equipped with plane blades and of a vaneless diffuser. The flow enters the impeller at the lower radius and is energized while passing through the blade channel, then it is purged towards the side channel at a higher radius, thus reducing its velocity along a helical trajectory and increasing the pressure level. Considering that the impeller elaborates the fluid several times through helical trajectories, the machine realizes an internal multistaging from which regenerative pumps take their name.

A one-dimensional tool called DART (Design and Analysis tool for Regenerative Turbomachinery) aimed at providing preliminary design parameters for plane blades pumps has been developed by the Turbomachinery and Combustion Research group of the Department of Industrial Engineering (DIEF) of the University of Florence (Italy) with the support of the Modeling R&D Department of Pierburg Pump Technology. The DART code is based on the momentum exchange theory described in Yoo et al. (2005) and Yoo et al. (2006) and has been verified using available experimental and numerical data in Insinna et al. (2018). Although Gülich (2010) demonstrated the high dependence of regenerative pump efficiency on the dimension of leakages in off-design conditions, the original version of the DART neglects the leakage flows' effect on the main-flow, thus providing inaccurate data when manufacturing uncertainty makes the dimension of the disk clearance relevant. The aim of this work is to develop a methodology to correctly evaluate the leakage flow through a stationary and a rotating disk in presence of tangential pressure non-uniformity and to implement it into the DART code. To comply with that aim, a theoretical study of the problem is performed and a new approach (partially based on the one originally proposed in Balje (1957)) is presented. The new method is implemented in Matlab® and is tuned using data obtained from the numerical simulation of virtual prototypes of regenerative pumps where the disk clearance is part of the control volume. In the second part of the paper the implementation of the new method in DART is described and the new version of the code is verified using the available experimental and numerical data. The obtained results show that the new methodology allows for an improved prediction of regenerative pump performance.

THE DART CODE

The use of simplified models for the preliminary analysis and design of energy systems' components is widely accepted in the turbomachinery field (see for example Griffini et al. (2015), Bontempo & Manna (2016), Bontempo & Manna (2017) and Griffini et al. (2016)). Concerning regenerative pumps, a one-dimensional tool called DART has been developed aiming at providing preliminary design parameters for the setup of detailed three-dimensional numerical simulations. The original model for performance prediction is as described in Yoo et al. (2005) and Yoo et al. (2006) and is based on the following hypotesis:

- steady, adiabatic, incompressible flow;
- pressure independent from axial and radial coordinates;
- leakage flow through disks does not have any impact on the main flow.

The second hypothesis makes the model one-dimensional, while the third hypotesis is the one that has to be overcome in order to increase DART's accuracy. The development and verification of the original version of the DART code is beyond the aims of the present paper. For a detailed description of DART refer to Insinna et al. (2018). The aim of the present work is to equip the DART code with a new methodology for the prediction of leakage flows through stationary and rotating disks in presence of a tangential pressure non-uniformity, which is a typical configuration during regenerative pump operation.

THE MODEL FOR LEAKAGE FLOWS

To allow for the relative motion between casing and impeller, a gap between these two parts is needed. This gap allows the fluid with high pressure (near the outlet section) to flows towards the lower pressure zones (see Figure 1). That leakage flow (called $Q_{l,disk}(\theta)$ in Figure

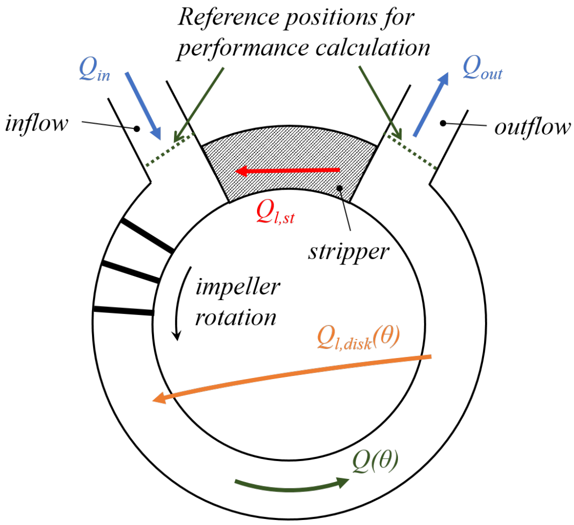


Figure 1: Scheme of the regenerative pump

1) generates a reduction of the overall efficiency (the already pumped fluid returns to lower pressure and part of the energy is wasted) and makes performances evaluation more complex.

With reference to Figure 1, it is possible to write the continuity equation as follows:

$$Q(\theta) = Q_{in} + Q_{l,st} + Q_{l,disk}(\theta) \tag{1}$$

This particular kind of leakage flow transforms the pump in a machine with tangentially-variable flow rate and then, for a correct calculation of the performance, it necessary to correctly estimate the leakage flow rate in every sector of the machine. The classical models for leakage flows are not sufficient in this case: in fact, the tangential gradient of pressure is usually not considered in those models. The models of Batchelor (1951) and of Stewartson (1953) are very important to describe the behavior of the leaked flow between the rotating and the stationary disks, but are only partially useful in the present case since they usually deal with radially inward and outward flows in side cavities (i.e. Salvadori et al. (2012) for centrifugal pumps and Gülich (2010) for centrifugal pumps), while in the present case the leakage flow moves along chords between two points along a circumferential position. That configuration has been studied by Balje (1957), but in that work only the estimation of total leaked flow was presented (no information about the distribution of the leakage flow was available). Therefore, the aim of the present work is to extend the method suggested by Balje in order to accurately evaluate the tangential distribution of leakage flow rate.

Theoretical development

The present model is based on two fundamental assumptions. The first one is that the diameter of the shaft is negligible with respect to the diameter of the hub of the blades. That hypothesis allows to develop a model where the blockage of the shaft does not modify the total leakage flow rate. The second one is that the contribution of centrifugal forces to the tangential distribution of the leakage flow is negligible with respect to the pressure ratio between two different zones of the pump. That hypothesis means that the fluid can follow a straight trajectory inside of the cavity between two points, considering negligible any deviation from a straight line between any couple of points. Given these hypothesis a stream-tube model can be developed.

The disk-casing cavity is considered as a zone delimited by a circular boundary and each couple of points P_i and P_j laying on that zone identify a stream tube along a chord C_{ij}. Therefore, the leakage flow rate in each stream tube can be determined by knowing the pressure ratio between P_i and P_j, the hydraulic resistance between P_i and P_j and the motion of the disk relative to the casing in every point of C_{ij}. In Figure 2a the circumference has been scaled to have radius equal to 1. and the radius defines the distance from the axis of the shaft to the hub of the vanes. Referring to 2a it is possible to define the angle θ that from the stripper locate the point A on the circumference and the angle ϕ that from the stripper locate the point B on the circumference. The flow rate from a point P_i to a point P_j can be written as follows:

$$q_{ij} = S_{ij}\left(W_{ij} + \frac{U_{mean,ij}}{2}\right) \tag{2}$$

where W_{ij} depends on the pressure ratio between the point P_i and the point P_j and S_{ij} is the passage area of the stream tube $P_i - P_j$. In a cavity with a relative motion between two walls, the superposition principle allows to correct the flow rate q_{ij} by a factor $\frac{U}{2}$, where U is the relative speed between the walls. Therefore, the term $U_{mean,ij}$ represents the effect of the rotation of the disk to the leakage flow and depends on the relative motion between the impeller and the casing. All of those terms have to be calculated separately. The energy equation for a generic stream tube $A - B$ can be written as follows:

$$\frac{p_A}{\rho} + \frac{W_A}{2} + R_{BA} = \frac{p_B}{\rho} + \frac{W_B}{2} \tag{3}$$

where R_{BA} are the losses in the stream-tube $A - B$ considering an uniform velocity profile. Also the section of the stream tube is considered constant, and then Equation (3) reduces to:

$$\frac{p_B - p_A}{\rho} = R_{BA} \tag{4}$$

For what concerns the sign of the velocity and the flow-rate it is considered *positive* the flow rate coming into the cavity and *negative* the flow rate exiting the cavity. The losses R_{BA} depend on the square of the velocity W, on the length L of the chord AB, on the friction factor f and on the equivalent hydraulic diameter D_{eq}:

$$R_{BA} = f\frac{L}{D_{eq}}\frac{W^2}{2} \tag{5}$$

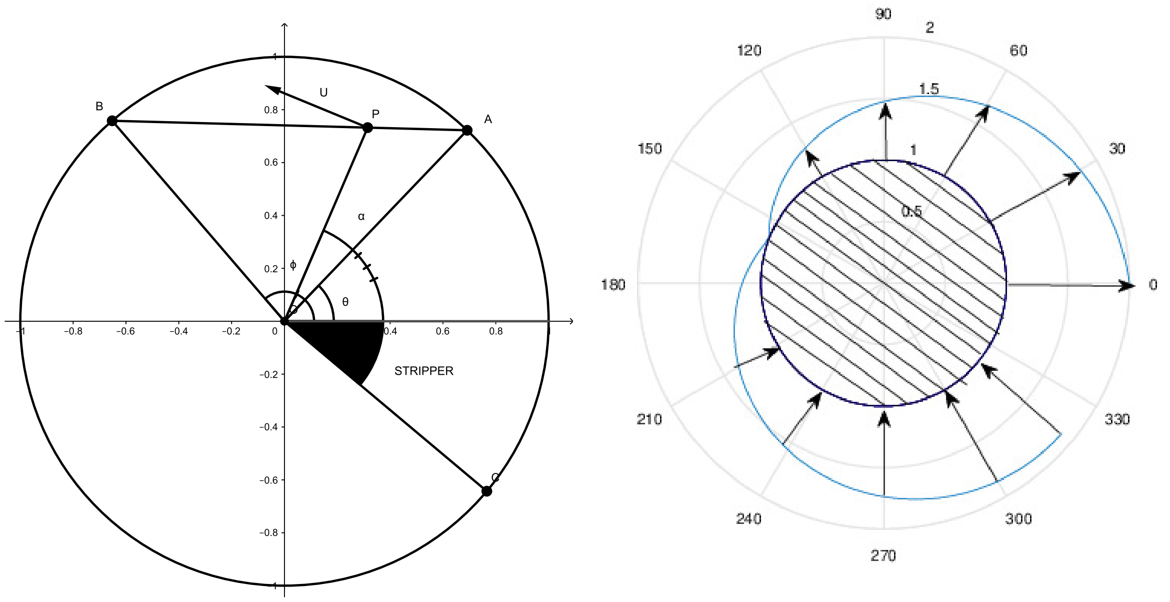
Combining Equation (5) with Equation (4) it is possible to evaluate the velocity W:

$$W_{AB} = \sqrt{\frac{2D_{eq}(p_B - p_A)}{Lf\rho}} \tag{6}$$

The equivalent hydraulic diameter is equal to the ratio between four times passage area and the wetted perimeter:

$$D_{eq} = \frac{4S}{l_{wetted}} = \frac{4hs}{2s} = 2h \tag{7}$$

where h is the height of the cavity. The length of the chord L is:



(a) (b)

Figure 2: (a) Nomenclature for angular positions
(b) Tangential $Q^{distribution}$

$$L = 2r \sin\left(\frac{\phi - \theta}{2}\right) \tag{8}$$

For what concerns the friction factor, it is a function of Reynolds number $Re = \frac{\rho W D_{eq}}{\mu}$ and of the relative roughness, which is available from a Moody's chart. Finally, an equation for the velocity W is available:

$$W_{AB} = \sqrt{\frac{2h(p_B - p_A)}{r f \rho \sin\left(\frac{\phi-\theta}{2}\right)}} \tag{9}$$

The value of the effect of the term $U_{mean,ij}$ of Equation (2) has also to be evaluated. Neither the magnitude nor the direction of U will be constant along a chord AB, and then the integral average of the parallel component of U along the chord (called U_{mean}) has to be used:

$$U_{mean} = \frac{1}{\phi - \theta} \int_{\theta}^{\phi} \frac{\mathbf{U} \bullet (A - B)}{\|A - B\|} \, \mathrm{d}\alpha. \tag{10}$$

With reference to Figure 2a it is possible to write:

$$\mathbf{U} = \rho\omega \begin{pmatrix} -\sin\alpha \\ \cos\alpha \end{pmatrix} \tag{11}$$

Combining Equation (11) with Equation (10) it is possible to evaluate U_{mean}:

$$U_{mean} = \omega \cos\frac{\phi - \theta}{2} sign\left(\sin\frac{\phi - \theta}{2}\right) \tag{12}$$

where:

$$sign(x) = \begin{cases} 1 & \text{if} x \geq 0 \\ -1 & \text{if} x < 0 \end{cases} \tag{13}$$

Once the velocities W_{ij} and $U_{mean,ij}$ are calculated it is possible to evaluate the flow rate using Equation (2) estimating the passage area. Given a discretization $d\theta = \frac{2\pi - \theta_{stripper}}{N}$, where N is the number of tangential steps, the passage area of the stream tube from $\left(\theta - \frac{d\theta}{2}, \theta + \frac{d\theta}{2}\right)$ to $\left(\phi - \frac{d\theta}{2}, \phi + \frac{d\theta}{2}\right)$ is:

$$S_{AB} = 2hr \sin\left(\frac{d\theta}{2}\right) \sin\left(\frac{\phi - \theta}{2}\right) \tag{14}$$

IMPLEMENTATION OF THE LEAKAGE MODEL

The proposed model has been initially implemented as a stand-alone tool in MATLAB® in order to test its accuracy and to tune the method using the available data obtained from a specifically performed numerical campaign using Computational Fluid Dynamics (CFD). The final version of the model has been implemented in DART and the new code has been verified using both CFD and experimental data.

In order to make the new model work in DART, the circumferential domain of the cavity is divided into n equal parts θ_i, written in a vector ϑ. The matrix Δp is also defined, where the $ij - th$ element represents the difference of pressure between ϑ_i and ϑ_j:

$$\Delta p = \mathbf{p}{\Gamma_n}^T - \left(\mathbf{p}{\Gamma_n}^T\right)^T \tag{15}$$

In Equation (15) Γ_n is a n-dimensional vector that has an unitary value in every component. With the same method the matrix containing the difference of the angular coordinate between ϑ_i and ϑ_j can be written:

$$\Delta\vartheta = \vartheta{\Gamma_n}^T - \left(\vartheta{\Gamma_n}^T\right)^T \tag{16}$$

In order to complete the implementation it is necessary to define the matrices of the passage areas, of the length of the chords and of the square root of differences of pressure between ϑ_i and ϑ_j:

$$S = \frac{h}{n-1} \left| 2r \sin\left(\frac{d\theta}{2}\right) \sin\left(\frac{\Delta\vartheta}{2}\right) \right| \tag{17}$$

$$L = \left| 2r \sin\left(\frac{\Delta\vartheta}{2}\right) \right| \tag{18}$$

$$K = \Re\left(\sqrt{\Delta P}\right) - \left[\Re\left(\sqrt{\Delta P}\right)\right]^T \tag{19}$$

In the latter equation the operator $\Re$ refers to the real part of the elements of the matrix. Since it is necessary to initialize an iterative cycle, a discharge coefficient is calculated with a methodology proposed by Balje (1957). The initial guess of the friction factor λ has therefore no physical relevance:

$$C_D = \sqrt{\frac{2h}{\lambda}\frac{1}{L}} \tag{20}$$

Then, the first attempt leakage flow rate Q^* will be:

$$Q^* = rhC_D \circ Kd\theta \tag{21}$$

It is now possible to calculate the matrix of Reynolds numbers and a fictitious velocity dividing the flow rate by the passage area:

$$Re = \frac{\rho \left| W_{old} \right| D_{eq}}{\mu} \tag{22}$$

$$W_{old,ij} = \frac{Q^*_{ij}}{S_{ij}} \tag{23}$$

The iterative cycle can start by recalculating the value of W with the following equation:

$$W_{new} = K \circ \sqrt{\frac{2\rho}{Col\left(Re; \frac{\varepsilon}{D_{eq}}\right) \circ \frac{L}{D_{eq}}}} \tag{24}$$

where $Col\left(Re; \frac{\varepsilon}{D_{eq}}\right)$ is the function that gives the friction factor using the Colebrook formula. The cycle will converge when the following convergence condition is reached:

$$\max \left| \frac{W_{new} - W_{old}}{W_{old}} \right| < \delta \tag{25}$$

In order to be coherent with the proposed physical model, it is possible to correct the obtained velocity value including the effects of rotation:

$$W_{rot} = W + \frac{\omega r \cos\frac{\Delta\vartheta}{2} \circ sign\left(\sin\frac{\Delta\vartheta}{2}\right)}{2} \tag{26}$$

where $\circ$ is the element-by-element product. According to Equation (2), the flow rate is $Q_{ij} = S_{ij} W_{rot,ij}$. From this matrix we can obtain the vector of the leakage flow rates for every discretized element $Q_i^{distribution}$ and the vector of cumulative sum of leaked flow rate Q_i^{cumul}:

$$Q_i^{distribution} = \sum_{j=1}^{N} Q_{ij} \tag{27}$$

$$Q_i^{cumul} = \sum_{j=1}^{i} Q_j^{distribution} \tag{28}$$

From these data the total flow rate leaked into the cavity can be finally calculated:

$$Q_{tot,leak} = \max_{i=1\ldots n} Q_i^{cumul} \tag{29}$$

The typical distribution of flow rate obtained from the aforementioned model is schematized in Figure 2b. The leakage flow rate is maximum close to the stripper (maximum pressure drop available) and reaches zero in the circumferential position opposite to the stripper region. In a real machine, a leakage flow rate will also be present between the stripper and the cavity. In the present model the stripper leakage flow rate is supposed to move across the stripper from the outlet to the inlet of the regenerative machine ($Q_{l,st}$ in Figure 1) without any interaction with the cavity flow and then it is treated separately from the disk leakage.

CALIBRATION OF THE LEAKAGE MODEL

The proposed leakage model has been initially calibrated using data from a numerical campaign performed considering a prototype regenerative pump whose characteristics fit the overall necessities of the automotive field of application.

Details about the three-dimensional simulations

The virtual model used for the Computational Fluid Dynamics (CFD) activity consists in a single-sided regenerative pump characterized by blades of semi-circular shape. The main parameters of the machines are reported in non-dimensional form with respect to the blade height in Table 1.

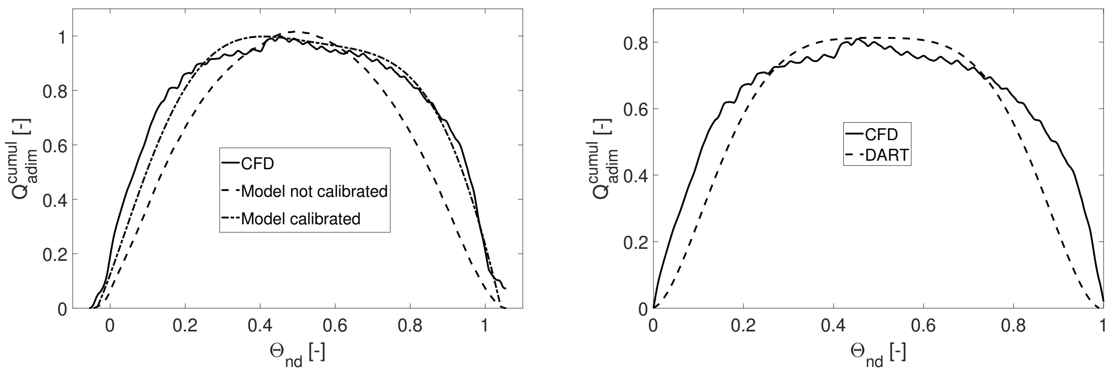
r_{hub}/H	[-]	1.23
r_{tip}/H	[-]	2.23
d/H	[-]	0.30
t/H	[-]	0.134
h/H	[-]	0.00886, 0.0177, 0.0354
N_{bl}	[-]	30
Φ	[-]	0.52
Ψ	[-]	2.5
Re	[-]	197000

Table 1: Non-dimensional parameters of the regenerative pump

The domain is composed by impeller, side channel, inlet and outlet ducts and the cavities (both stripper and disk). The outlet duct is extended about 10 diameters downstream of the pump. Such an elongation of the outlet duct is necessary for avoiding the formation of flow recirculation on the outlet section due to the presence of residual swirl under some operating conditions. Three different values for the dimension of the disk clearance has been considered. For the investigated cases, the radial clearance at the rotor tip has the same value considered for the disk clearance.

The computational grid used for the three-dimensional RANS simulations of the pump is generated with the meshing tool of the commercial software Pumplinx®. Attention is dedicated to the refinement of the critical region of the stripper leakage and of the cavity. The overall grid is composed by about 14.8M elements. Such grid resolution is appropriate to make the problem grid-independent according to the works of Quail et al. (2011) and Nejadrajabali et al. (2016), where grid sensitivity studies were performed on similar machines.

The three-dimensional numerical campaign has been carried out using the Pumplinx® solver. Air is treated as an ideal gas while the viscosity is calculated using Sutherland's law. Second-order accurate discretization is used for the continuity and the momentum equation while first-order accurate discretization is used for the energy equation and turbulence modelling. The realizable $\kappa - \epsilon$ model Shih et al. (1995) is used as turbulence closure. The selected turbulence



(a) Model and CFD data

(b) DART and CFD data

Figure 3: Q^{cumul} comparisons for $h/H = 0.00886$

closure demonstrated to be reliable for the prediction of performance of regenerative pumps, as showed by Quail et al. (2011). At the maximum flow rate simulated, the average y^+ is always below 5 (compatible with the y^+-independent approach used).

The frozen-rotor approach has been chosen. The impeller is frozen in a symmetrical position with respect to the stripper, with the maximum closure of this latter by the vanes. This position was chosen in order to compare the results with DARTs predictions, since such configuration is assumed for the estimation of the leakage flow. It has been verified that a change in the relative positioning between impeller and static parts does not modify substantially the obtained results. For all the simulations, mass-flow rate and total temperature were imposed on inlet section while static pressure is defined at the outlet. For the calculation of the pressure rise of the machine, the reference sections are in correspondence of the inlet and outlet sections of the model. Although the simulations include fluid's compressibility, it is observed that for the investigated cases density variations are negligible, thus confirming that CFD data can be compared with DART results.

Calibration of the model

The model has been calibrated comparing its results with data from the CFD simulations. For the calibration procedure the case with the smaller gap ($h/H = 0.00886$) has been considered. For the comparison between CFD data and the outcome of the 1D model, the interface region between the side channel and the cavity has been considered to extract (from the numerical simulations) the boundary conditions of the model and the reference data for the comparison. The circumference has been split into 600 sectors and for every sector the averaged values of pressure and radial velocity has been considered. The number of sectors has been chosen after a sensitivity test. It has been demonstrated that over 600 sectors there are no more differences in the obtained results, then the outcome of the model is independent from the discretization in the tangential direction. However, in the standard routine DART uses a lower number of sectors to obtain a faster convergence rate .The sum of the elements of the latter vector multiplied for the surface of one sector represents the total flow rate leaked through the cavity. The 1D model uses a discretization that is different from the one used for the numerical campaign and then the tangential pressure distribution has been interpolated at the boundary with a shape-preserving algorithm.

In Figure 3a a comparison between the model (dashed line) and the CFD data (solid line) for Q^{cumul} at $h/H = 0.00886$ is reported. In that figure $\theta_{nd} = 0$ is in correspondence of the center of inlet duct and $\theta_{nd} = 1$ corresponds to the center of the outlet duct. It can be observed that the model does not forecast the fluctuations due to the presence of the vanes, but it was an expected result since they are not considered in the hypothesis of the model. The discrepancy on the global leaked flow rate can be estimated around 1,66%. The trend of the curve for the model is quite similar to the curve obtained from CFD data, particularly for what concern the tangential position of the maximum of the curve. The limits of the zones with a positive derivative (the flow rate that leaks from the side channel into the cavity) and the zones with a negative derivative (the flow rate leaks from the cavity to the side channel) are also well reproduced.

The model seems to be able to reproduce the overall flow rate passing from side to side of the stator/rotor cavity, but also shows some discrepancies in the reproduction of the flow rate repartition around the disk. In order to overcome such limitation, a non-dimensional distribution of the flow rate around the stator/rotor cavity obtained from the CFD campaign is used to tune the model. Since the leakage flow through the stripper zone is not considered in DART as a contribution to the total leaking flow, that flow rate has been deducted and the CFD data distribution visible in Figure 3b (solid line) is obtained. As it is possible to observe the non-dimensional distribution does not reach the unitary value due to the exclusion of the stripper flow rate from the computation of the cumulative distribution (the θ_{nd} variable is therefore limited between 0 and 1). The CFD data distribution has been fitted with a polynomial function and used in a non-dimensional form to correct the theoretical flow rate (Equation (2)) obtained from the proposed model. The comparison between the CFD data and the stand-alone model after the tuning procedure is visible in Figure 3b (dash-dotted line).

In Figure 3b a comparison between the results calculated by a version of DART equipped with the tuned leakage model (dashed line) and the results from CFD campaign without stripper leakage is reported. The two curves showed in Figure 3b are obtained for the same pump but with two completely different approaches. In fact, the dashed line in Figure 3b is obtained by calculating iteratively the pressure distribution and the leakage flow rate in a coupled way with DART. As can be observed, the prediction of the leakage flow distribution along the tangential direction is quite close to the CFD data and to the one obtained with the calibrated stand-alone model (dash-dotted line in Figure 3b), thus demonstrating that the tuned model implemented in DART is able to capture the quantitative and qualitative behaviour of the leakage flow.

VALIDATION OF THE MODEL

It has been showed that the DART code equipped with a tuned disk leakage model is able to reproduce correctly the tangential distribution of leakage flow rate if compared with CFD data. It is now necessary to check its accuracy when the performance of the regenerative pump is of interest. The code verification procedure is therefore completed comparing the data obtained using the DART code with the CFD data from the already described campaign and with experimental data from literature.

Comparison with CFD data

A comparison between the performance obtained with DART and with the CFD campaign that has been introduced in the previous section is here reported. In Table 2 the data obtained from DART with the leakage option switched on and off are compared with CFD data. Variable were reduced to non-dimensional values with reference to the non-dimensional Δp value obtained using CFD for the case with a non-dimensional axial clearance equal to 0.0177.

As can be observed, both the Δp and the η values are greatly overestimated with respect to CFD data if the leakage model is switched off. The accuracy of the DART code increases when the leakage flow model is used, especially in terms of Δp for both the $h/H = 0.00886$ and the $h/H = 0.0354$ cases. Also the trend of variation of the performance is well captured, with some notable exception. In fact, both the Δp and the η values for the $h/H = 0.0177$ case does not diminish as expected and in general the η values obtained with DART are higher than the respective ones obtained from the CFD campaign. The latter behaviour can be explained considering that a one-dimensional model cannot correctly evaluate the impact of three-dimensional phenomena (like secondary flows) on the performance of the machine, unless they are specifically modelled with correlations that has to be tuned for the specific range of application.

h/H	0.0		0.00886		0.0177		0.0354	
	Δp	η	Δp	η	Δp	η	Δp	η
DART w/o leakage	2.40	50%	-	-	-	-	-	-
DART with leakage	-	-	2.07	44%	1.43	45%	0.52	30%
CFD	-	-	2.05	34%	1.00	26%	0.58	21%

Table 2: Comparison between performances of CFD and DART

In order to better understand the differences in the Δp prediction, the pressure variation along the circumferential coordinate for the three investigated cases has been analyzed. Concerning the CFD data, the same procedure used to extract the pressure boundary condition for the stand-alone model is used. Results are reported in Figure 4 in a non-dimensional form with respect to the Δp value obtained from the CFD campaign excluding the stripper region. Data showed in Figure 4 refer to a case with $h/H = 0.0177$ and a halved radial clearance at the rotor tip. As can be observed, the inclination of the pressure variation along the circumferential direction obtained without the leakage model (dash-dotted line) is higher than the one obtained from the CFD campaign (solid line), thus leading to a higher Δp value of the pump. That overestimation is corrected by activating the leakage model (dashed line in Figure 4), which iteratively corrects the local flow rate of the pump and reduce the overall performance in terms of Δp. Current data demonstrate that the implementation of a calibrated leakage flow model in the original one-dimensional code improves its accuracy by a non-negligible factor.

Comparison with experimental data

There is a limited amount of experimental data in literature about regenerative pumps' performance. In fact, those machines are not extensively studied if compared with centrifugal pumps. Anyway, the dimension of the disk clearance can be extrapolated from a study made by Yoo et al. (2006) about a regenerative pump used as an heart pump. It can be observed that for the cases studied in Yoo et al. (2006), leakage flows' impact on the performance of the heart pump is more relevant as the ratio h/H increases. That outcome is coherent with the numerical data showed in Table 2, where it is demonstrated that increasing the clearance value has a detrimental effect on the regenerative pump performance.

Since the heart pump is fully described by Yoo et al. (2006), their experimental results can be compared with data obtained with DART for the same geometrical configuration (including clearance dimension) and working conditions. Only flow rates close to the design point at $\omega = 2400$ RPM are considered, to study the clearance effects limiting the impact of off-design

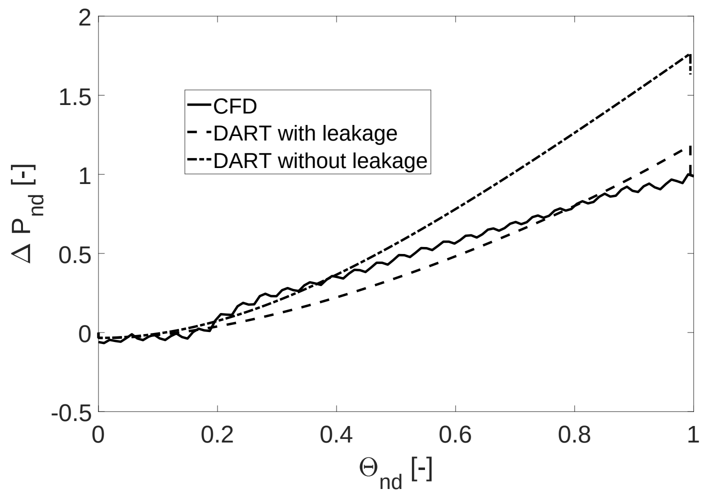


Figure 4: Comparison of pressure rise between DART (with and w/o leakage option) and CFD data

conditions on the overall performance. The comparison between the experimental data and the numerical results is reported in Table 3 for the case $h/H = 0.00282$. The percentage of overestimation of the pressure rise made by DART with respect to the experimental value is tabulated.

Q [lpm]	DART without leakage [%]	DART with leakage [%]	Experimental data [mbar]
3.45	+30	+11	216
3.95	+42	+9	182
4.40	+40	+3	149
4.85	+41	-6	115
5.25	+42	-20	82

Table 3: Comparison between pressure rise of experimental data and DART

The entity of the misestimation for the case without leakage model is quite high (around +40%), while the leakage model allows to limit the difference with respect to the experiments to the (-20%;+11%) interval. Further than the reduction of the error, it is interesting to observe that switching on the leakage flow model makes the variation of performance with respect to the experimental data dependent on the flow rate of the pump.

Although it is not possible to state that the model is fully validated, it can be concluded that the implemented method increases the accuracy of the DART code by introducing a physical feature that has a fundamental impact on the regenerative pump performance.

CONCLUSIONS

A novel methodology for the evaluation of leakage flow tangential distribution through a stationary and a rotating disk subject to a non-uniform pressure field is proposed. The aim of the activity is to implement the methodology in the DART code, which is able to predict the performance of a regenerative pump using a one-dimensional approach. The theory underlying the methodology is explained and the implementation procedure is detailed. The outcome of the stand-alone methodology is initially compared with data obtained from a three-dimensional CFD campaign of a virtual prototype used as reference case. A tuning procedure is defined in order to correctly reproduce the shape of the tangential flow rate distribution neglecting the stripper flow rate contribution.

The tuned model is coupled with the original algorithm in DART and results are compared with CFD data, thus demonstrating that the new version of the code is able to capture the main-flow/leakage-flow interaction phenomenon. Then, DART is used to analyse the performance of the reference case varying the clearance dimension and results are compared with CFD data. The obtained results show that if the clearance dimension increases, the performance of the pump decreases. Furthermore, data demonstrate that the implementation of the calibrated leakage-flow model improves DART accuracy by a non-negligible factor. Finally, DART is used to analyse a heart pump geometry whose characteristics are available from literature. Based on the data comparison, it can be concluded that the implemented method greatly increases the accuracy of the DART code by considering a real-machine effect that has a fundamental impact on regenerative pump performance.

ACKNOWLEDGEMENTS

The authors want to thank sincerely Giacomo Armenio (R & D Director Global, BU Pierburg Pump Technology), Frank Maurer (CEO, Pierburg Pump Technology Italy), Prof. Marco Pierini and Prof. Francesco Martelli (University of Florence) for without them, this paper would not have been possible.

References

Balje, O. (1957), 'Drag Turbines Performances', *Transactions of ASME* **79**(6), 1291–1302.

Batchelor, G. (1951), 'Note on a class of solutions of the navier−stokes equations representing steady rotationally-symmetric flow', *Quart. J. Mech. Appl. Math.* **4**, 29–41. DOI:10.1093/qjmam/4.1.29.

Bontempo, R. & Manna, M. (2016), 'Analysis and evaluation of the momentum theory errors as applied to propellers', *AIAA Journal* **54**(12), 3840–3848. DOI:10.2514/1-J055131.

Bontempo, R. & Manna, M. (2017), 'Highly accurate error estimate of the momentum theory as applied to wind turbines', *Wind Energy* **20**(8), 1405–1419. DOI:10.1002/we.2100.

Griffini, D., Salvadori, S., Carnevale, M., Cappelletti, A., Ottanelli, L. & Martelli, F. (2015), 'On the development of an efficient regenerative compressor', *Energy Procedia* **82**, 252–257. DOI:10.1016/j.egypro.2015.12.030.

Griffini, D., Salvadori, S. & Martelli, F. (2016), 'Thermo-hydrodynamic analysis of plain and tilting pad bearings', *Energy procedia* **101**, 2–9. DOI:10.1016/j.egypro.2016.11.001.

Gülich, J. F. (2010), *Centrifugal Pumps*, second edn, Springer. DOI:10.1007/978-3-642-12824-0.

Insinna, M., Salvadori, S., Martelli, F., Peroni, G., Simon, G., Dipace, A. & Squarcini, R. (2018), 'One-dimensional prediction and three dimensional cfd simulation of the fluid dynamics of regenerative pumps', *ASME Turbo Expo 2018: Turbomachinery Technical Conference and Exposition* **2C: Turbomachinery**, V02CT42A036. Paper No. GT2018-76416, DOI:10.1115/GT2018-76416.

Nejadrajabali, J., Riasi, A. & Nourbakhsh, S. (2016), 'Flow pattern analysis and performance improvement of regenerative flow pump using blade geometry modification', *International journal of rotating machinery* **2016**(77), 1–16. DOI:10.1155/2016/8628467.

Quail, F., Scanlon, T. & Stickland, M. (2011), 'Design optimisation of a regenerative pump using numerical and experimental techniques', *International Journal of Numerical Methods for Heat & Fluid Flow* **21**(1), 95–111. DOI:10.1108/09615531111095094.

Salvadori, S., Marini, A. & Martelli, F. (2012), 'Methodology for the residual axial thrust evaluation in multistage centrifugal pumps', *Engineering Applications of Computational Fluid Mechanics* **6**(2), 271–284. DOI:10.1080/19942060.2012.11015420.

Shih, T., Liou, W., Shabbir, A., Zhang, Z. & Zhu, J. (1995), 'A new $\kappa - \epsilon$ eddy-viscosity model for high reynolds number turbolent flows - model development and validation', *Computers fluid* **24**(3), 227–238. DOI:10.1016/0045-7930(94)00032-T.

Stewartson, K. (1953), 'On the flow between two rotating coaxial discs', *Proc. Camb. Phil.Soc.* **49**, 333–341. DOI:10.1017/S0305004100028437.

Yoo, I. S., Park, M. R. & Chung, M. K. (2005), 'Improved Momentum Exchange Theory For Incompressible Regenerative Turbomachines', *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* **219**(7), 567–581. DOI:10.1243/095765005X31252.

Yoo, I. S., Park, M. R. & Chung, M. K. (2006), 'Hydraulic design of a regenerative flow pump for an artificial heart pump', *Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy* **220**(7), 699–706. DOI:10.1243/09576509JPE211.