Turbopump Design: Comparison of Numerical Simulations to an Already Validated Reduced-Order Model

A Apollonio¹*, A Anderlini¹, D Valentini², G Pace¹, A Pasini¹, M V Salvetti¹, and L d’Agostino¹

¹ Department of Civil and Industrial Engineering, University of Pisa, Pisa, Italy
² Researcher, Italy

*Corresponding author’s e-mail: alessandro.apollonio@ing.unipi.it

Abstract. The article expands on the ongoing assessment of the reduced order model proposed by some of the authors for the geometric definition and noncavitating performance evaluation in the preliminary design and parametric optimization of mixed-flow centrifugal turbopumps. Some of the dynamically most significant predictions of the model are compared with the experimentally validated URANS (Unsteady Reynolds-Averaged Navier-Stokes) simulations of the non-cavitating flow through a typical six-bladed unshrouded mixed-flow turbopump for liquid propellant rocket engines operating at both design and off-design flow conditions and different values of the impeller clearance. The observed discrepancies can be explained in terms of the simplifying assumptions introduced for the development of the model and their relative magnitude (< ±10%) does not adversely interfere with the accurate prediction of the turbopump performance over a wide range of operating conditions above and below design flow rate. Together with earlier experimental validations, the results dramatically confirm the capability of the proposed model to generate useful engineering solutions of the turbopump preliminary design problem at a negligible fraction of the computational cost required by 3D numerical simulations.

1. Introduction

Nowadays, the competitiveness of the global space transportation market is a main driver in the design of liquid propellant rocket engine turbopumps. Unshrouded impellers can reach higher power densities at lower manufacturing costs compared to shrouded impellers [1]. For this reason, they have been introduced by ESA and JAXA in the turbopump design of the European Vulcain X demonstration program [2] and the new first-stage Japanese LE-9 rocket engine [3]. Avoiding rotor-stator contacts and meeting tip speed and rotordynamic limitations represent some of the most critical aspects of successful design of unshrouded impellers [4]. The choice of the blade tip clearance is especially crucial in these respects since its excessive increase leads to separated and possibly cavitating leakage and secondary flows, which adversely affect the pump performance [4–6]. Because of its importance in applications, a significant body of information on impeller clearance effects in turbopumps and compressors is therefore available in the literature (Negishi et al. [6]).

The first step in the conceptual design of liquid propellant rocket engine turbopumps consists in the development of quasi-1D models [7] as a first step towards more complex 2D or quasi-3D-dimensional analyses. However, there is a lack of examples in the literature of application of fully-3D analytical models to the design of space propulsion turbopumps. As reported by Arnone [7] and Williams [1], 3D
analyses of these machines typically consist instead in steady-state incompressible CFD simulations validated by comparison against the results of water tests (e.g. [8]). Next, more complex numerical simulations, usually based on compressible – possibly unsteady – RANS (Reynolds-Averaged Navier-Stokes) models, can be adopted to account for cavitation, fluid compressibility, flow instabilities and the effects of geometrical details, such as clearance flows and leakage losses [6,9,10]. However, especially when used for iterative optimization of the machine geometry, these simulations require prohibitive computational resources and extremely expensive validations in complex full-scale experiments under real-life operating conditions with actual (possibly cryogenic) propellants [6]. Therefore the design development of high-performance turbopumps for space propulsion applications can greatly benefit from the preliminary definition of a realistic geometry of the machine satisfying given performance requirements. To this purpose, the availability of an analytical model capable of rapidly providing information on both the pump geometry, performance and flow field at an affordable computational cost is highly desirable.

Closed-form direct (e.g. [11,12]) and inverse (e.g. [13,14]) methods are ideally suited for geometric definition and performance prediction of turbopumps, but they are limited in their capability of accurately capturing the influence of backflow phenomena, clearance effects and secondary flows. The degradation of the pumping performance induced by blade tip leakage effects can be empirically modeled and estimated in terms of a linear relation between the head coefficient and the relative clearance of the impeller. Negishi et al. performed compressible and incompressible URANS (Unsteady Reynolds-Averaged Navier-Stokes) [10] simulations as well as compressible RANS [6] simulations of the relative flow (frozen rotor) in the prototype of the LE-9 turbopump. They compared their numerical results with experimental data and confirmed the linear dependence of the head drop on the impeller clearance, indicating simulations as a rapid and cost-effective way to account for this source of flow losses. Fu et al. [15] carried out a series of frozen-rotor steady RANS simulations of an unshrouded centrifugal pump with a relative blade tip clearance of 11.2% and compared the results obtained using four different turbulence models with those of experimental tests in water, concluding that their choice has little influence on the prediction of the pump performance. Kergourlay et al. [16] and Yan et al. [17] investigated the internal flow in a centrifugal impeller by means URANS method with a sliding mesh technique to simulate the rotating components. In both cases the numerical predictions closely matched the experimental results over a relatively wide range of operating conditions. A similar numerical approach has also been adopted by Kim et al. [9] to study an oxidizer pump. They used an unstructured grid and viscous no slip boundary conditions on all wetted surfaces for four different inducer tip clearances, including the configuration with zero tip clearance.

In conclusion, the available evidence confirms that the effective development of turbopump design should progress from the use of analytical models and relatively inexpensive incompressible URANS simulations, capable of analyzing the response of the machine performance to geometrical changes and optimizing its shape, to the experimental validation of scaled models of the machine in laboratory tests under fluid dynamic similarity and, ultimately, to the final demonstration of the prototype in full-scale tests under real-life operating conditions.

In recent years d’Agostino and his collaborators developed and experimentally validated two closed-form models based on a direct approach for the preliminary geometrical definition and performance prediction of inducers [11] and centrifugal turbopumps [12]. The models describe the velocity field in the impeller as the superposition of a 2D axial vorticity correction (slip flow) to the fully-guided forced-vortex flow, with viscous effects (flow blockage, head losses, deviation, etc.) evaluated from the integration of the turbulent boundary layers along the blade channels. Other major sources of losses (incidence, deviation, mixing and clearance) are included using, when necessary, accepted semi-empirical correlations of experimental or numerical results. Both of these models have been successfully validated for the prediction of integral properties of the machine, namely its head performance characteristic and hydraulic efficiency under non-cavitating conditions. In this context, the present work specifically aims at investigating to what extent the centrifugal turbopump flow model proposed by d’Agostino et al. [12] is also capable of describing the flow field in the machine. To this purpose the
predictions of the model are compared with results of URANS simulations of the velocity field and flow deviation at zero tip blade clearance.

2. Method

2.1. Analytical Method

The main aspects of the analytical model proposed by d’Agostino et al. [5,11,12,18] for the prediction of the head characteristic and hydraulic efficiency of non-cavitating centrifugal turbopumps are here summarized herein. For more details the reader is referred to the cited publications.

The blade channels are defined as the intersection of \( N \) infinitely thin impeller blades with the hub and tip surfaces of radii \( r_h \) and \( r_t \) respectively, both coinciding with the streamlines of a 2D axisymmetric stagnation flow in the meridional plane. The shape of the blades is described in terms of its helical angle \( \gamma \) w.r.t. to the axial direction and radial back sweep angle \( \chi \). The resulting blade pitch \( P \) is assumed to be a cubic function of the axial \( z \) coordinate, with suitable initial and final conditions for gradual transition from the axial inlet inducer to the radial impeller outlet.

The inlet velocity \( \bar{w}_0 \) and pressure \( p_0 \), as well as the density \( \rho \), the impeller rotational speed \( \Omega \) and the tip radius \( r_2 \), are constant. Moreover, the axial velocity \( w(z) \) through the impeller is assumed radially uniform at any axial station and the 3D flow in the blade channels is described as the superposition of a fully-guided axisymmetric (rotational) flow \( \bar{u} \) and a displacement (slip) flow \( \bar{u}_d \), necessary to satisfy the irrotationality condition (Kelvin’s theorem) in the absolute frame. Neglecting axial derivatives of the blade pitch and back sweep angle (therefore treated as slowly-varying in \( z \)), the displacement flow \( \bar{u}_d \) is planar on any axial cross-section of the blade channels and can therefore be expressed in terms of a 2D stream function \( \bar{\Psi}(z) \), which is obtained in closed form as the spectral solution of a Poisson’s boundary value problem with homogeneous boundary conditions \( \bar{\Psi} = 0 \) [12].

The average circumferential velocity at the impeller exit surface (subscript 2) is then evaluated as:

\[
\bar{v}_2 = \bar{u}_2 \tan \bar{\phi}_2 + \Omega r_2 + \int_{S_2} \bar{v}_2 \ dS / \int_{S_2} dS, \tag{1}
\]

where

\[
\bar{\phi}_2 = a \tan \left( -\left( \int_{S_2} \bar{v}_2 \ dS / \int_{S_2} dS - \Omega r_2 \right) / \int_{S_2} \bar{u}_2 \ dS / \int_{S_2} dS + \delta_2 \right) \tag{2}
\]

is the mean flow angle w.r.t. the radial direction and \( \delta_2 \), estimated as in [19], is the flow deviation generated by the blade shape, loading and boundary layer displacement effects. The impact of the deviation angle, blockage and slip velocity at the impeller exit section is illustrated in figure 1, where \( \bar{\phi}_2 \) is mass averaged flow angle generated by the fully guided flow. Hence, for a fully guided flow \( \bar{\phi}_2 = \bar{\phi}_2 \). Assuming perfect mixing of the impeller discharge flow at the inlet, the geometry of the radial diffuser is designed for generating log-spiral streamlines of the flow throughout the exit section, [12,18–20]. The single volute is characterized by elliptical meridional cross-sections transitioning from a straight segment at the tongue to a circle at the exit. No pressure recovery is assumed for the (concentrated) mixing losses at the inlet of the diffuser and volute. For more details, incompatible with the space limitations of the present article, interested readers are referred to the cited publications.

![Figure 1. Velocity triangles of flow at the exit section of the turbopump impeller. The slip flow, deviation and blockage effects reduce the angle \( \bar{\phi}_2 \) w.r.t. its ideal value \( \bar{\phi}_2 \).](image-url)
The main sources of performance degradation are included in the model in order to more accurately evaluate the actual pumping characteristic and hydraulic efficiency of the machine under non-cavitating conditions [19]. Frictional losses, flow blockage and flow deviation effects in the impeller are evaluated from the thicknesses of the boundary layers at the exit of the blade channels. Incidence losses at the inlet of the blades and of the volute tongue, as well as frictional losses in the diffuser and the volute, are also included.

2.2. Experimental Set-up

The Cavitating Pump Rotodynamic Test Facility (CPRTF) at Alta has been used to carry out tests under non-cavitating and cavitating conditions in water for the characterization of the hydraulic performance of the machine, the study of relevant fluid-dynamic phenomena and the experimental validation of the proposed model [8,21]. Figure 2 shows a schematic of the closed-loop CPRTF with its flow control systems, monitoring devices, rotating dynamometer and the positions of the temperature and pressure sensors used in present experiments.

![Diagram of CPRTF](image)

**Figure 2.** Plan view (left) of the CPRTF, its main systems, devices and pressure sensors used in present tests. Detailed view (right) of the VAMPIRE test pump mounted in the CPRTF test section, with the positions of the pressure taps used in present tests (adapted from [22]).

In a series of experiments funded by the European Space Agency (ESA) [5,23], the CPRTF has been equipped with pressure transducers mounted at different positions on the pump casing and volute (figure 2 on the right). Table 1 lists the main characteristics of the pressure transducers used for the performance characterization of the present test machine. The flow rate has been measured in the suction and discharge lines by means of electromagnetic flowmeters (Fisher-Rosemount, mod. 8732E, 0-100 l/s range, 0.5% precision). A brushless motor (MOOG, model FASF3V8029, 0-5000 rpm rotational speed, 0-30 kW power) drives the pump and its impeller, which is mounted in the CPRTF test section. Additional details of the experimental set-up and its devices are reported in [8,21].

The mixed-flow centrifugal turbopump, called VAMPIRE, used in the present activity has a six-bladed backswept impeller, a vaneless diffuser and a single spiral volute (see [21] for more details). It has been designed and optimized for maximum efficiency at nominal operating conditions and zero blade clearance by means of the proposed model in combination with a 2nd order gradiential maximization method. The VAMPIRE pump has been experimentally characterized at 1500 rpm and a blade tip Reynolds number equal to \(3.41 \times 10^6\) for three relative clearances \(c_{\text{rel}} = 0.9, 5.6\) and 11.2%) corresponding to radial gaps of 0.16, 1, and 2 mm, respectively. The head degradation due to clearance effects has been accounted for in model by means of the following empirical relation:

\[
\frac{\Psi}{\Psi_{c=5.6}} = -0.7413 \frac{c_{\text{rel}}}{100} + 1.0493. \quad (3)
\]
Finally, the results of the comparison of the predicted and measured pumping performance of the VAMPIRE pump are illustrated in figure 5 (left) [21].

Table 1. Characteristics of the pressure transducers used for the experimental campaign.

| Transducer ID | Model          | Full Scale (FS) | Accuracy |
|---------------|----------------|-----------------|----------|
| P₁ (abs)      | DRUCK PMP1400  | 1 bar           | ±0.25% FS|
| P₆ (abs)      | UNIK 500 PMP5073 | 6 bar           | ±0.1% FS |
| ΔP₂ (diff)    | UNIK5000 5073  | 5 bar           | ±0.1% FS |
| ΔP₃ (diff)    | DRUCK PMP4170  | 1 bar           | ±0.08% FS|
| ΔP₄ (diff)    | KULITE BMD 1P 1500 | 7 bar       | ±0.1% FS |

2.3. Computational Method

Fu et al. [15] performed a series of frozen-rotor numerical simulations of the VAMPIRE turbopump with 11.2% relative clearance. This approach underestimated the head coefficient by more than 10% for flow rates below 80% of design conditions. For this reason, a sliding-mesh approach has later been adopted [16,17,9,24] in a second series of simulations carried out by means of the SIEMENSTARCCM+ commercial code [27] using a standard finite-volume approach on unstructured computational grids. The governing equations, based on the Unsteady Reynolds-Averaged Navier-Stokes (URANS) approach for dealing with turbulence effects, are the three-dimensional incompressible flow equations:

\[
\frac{\partial u_i}{\partial x_i} = 0 \quad \text{and} \quad \frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho} \frac{\partial p}{\partial x_i} + \nu \frac{\partial^2 u_i}{\partial x_j \partial x_j} + \frac{\partial \tau_{ij}^t}{\partial x_j},
\]

where \( u_i \) is the time-averaged velocity in the \( i^{th} \) direction, \( p \) is the time-averaged pressure, and \( \nu \) and \( \nu \) are the fluid density and viscosity. \( \tau_{ij}^t \) is the so-called Reynolds stress-tensor, which contains the effects of the unresolved turbulent fluctuations on the mean flow field. In this work, the Reynolds stress-tensor has been closed with the \( k - \omega \) SST turbulence model [25]. Standard wall functions have been applied to describe the flow behavior inside the boundary layer.

The domain of the numerical simulations (figure 3) has been divided in three regions: the inlet pipe and the volute, which is fixed in the absolute reference frame, and the impeller, which rotates steadily at 1500 rpm. A sliding mesh technique has been employed at the interfaces (“Diffuser interface” and “Inlet interface” in figure 3) in order to allow for the unsteady interactions between the rotating and stationary grids.

The boundary conditions have been prescribed by assigning the static (noncavitating) pressure at the outlet of the volute, a parabolic velocity profile with the desired flow rate at the upstream cross-section of the inlet pipe, and viscous no-slip conditions on the remaining solid surfaces.

The simulations have been performed at 60%, 80%, 100%, and 120% of the design flow rate for two values, 5.6% and 11.2%, of the relative tip clearance and the same rotational velocity as in the validation tests (1500 rpm).

The computational grid is polyhedral with a total of about 14 million nodes in the three regions, specifically 0.25·10⁶ nodes in the inlet pipe, 12.5·10⁶ in the impeller and 1.25·10⁶ in the volute. The mesh on the impeller surface is illustrated in figure 4. The number of grid points is not significantly affected by the small changes of the computational domain in the simulations with different tip clearances. The grid independence has been checked by comparing the results to those obtained on a finer grid (not shown here for the sake of brevity).

An upwind scheme has been used for the discretization of the convective terms and a central one for the viscous terms, both second-order accurate. Time has been advanced implicitly by means of a second-order backward scheme with a constant time-step \( \Delta t = 1.2 \cdot 10^{-4} \) s, corresponding to about 1 deg rotation of the impeller.
The quantities of interests discussed in Sec. 2.1 have been monitored, namely: the average circumferential and radial absolute velocities, \( \bar{v}_2^{CFD} \) and \( \bar{u}_2^{CFD} \), and the mean flow angle w.r.t. the radial direction:

\[
\phi_2^{CFD} = \arctan \left( -\frac{\bar{v}_2^{CFD}}{\bar{u}_2^{CFD}} \right)
\]

at the impeller exit. Flow pressures have been recorded at the locations of the relevant transducer taps in the experiments.

3. Results and Discussion

Figure 5 shows the comparison between the head coefficient and hydraulic efficiency of the VAMPIRE pump measured in water tests for different values of the impeller clearance and the predictions of the analytical model corrected for tip leakage effects. The almost perfect overlap of the head characteristics at 0.9% relative clearance demonstrates the capability of the model to effectively predict the relevant integral parameters for the assessment of centrifugal turbopump performance over a wide range of operating conditions above and below the design point. Figure 6 shows the pump performance obtained in the experimental tests and the numerical simulations. The line with downward-pointing-triangles has been obtained using equation (3) to extrapolated at zero clearance the experimental pumping characteristic at 5.6% relative clearance. Table 2 reports the relative deviations of the simulations from the experimental data at 60%, 80%, 100%, and 120% of the design flow rate. At design conditions and 5.6% relative clearance the head coefficient predicted by the simulations is \( \sim 0.31 \) with an error of +2% w.r.t. the experimental data. At 11.2% relative clearance the same error at design conditions is -1.3%. With no clearance the error of the simulations is \( \sim 0\% \) w.r.t. the experimental pumping characteristic extrapolated to zero clearance. At 120% of the design flow rate and 11.2% relative clearance the head coefficient is underestimated by 5.6%, while for the other flow rates and clearances the relative deviation between experimental and numerical results ranges from -1.8% to +2.7%.

The comparison shows that numerical simulations are in good quantitative agreement with the experimental findings. In particular, for the zero-clearance case, according to CFD results the head coefficient has the same value and trend as the extrapolated experimental performance curve. As a whole, these results confirm the validity of the numerical simulations, which can then be used for comparison with the analytical model.

Figure 7 shows the nondimensional difference between the static pressure at the outlet of the impeller and the volute predicted by the analytical model and calculated by the numerical simulations at 60%, 80%, 100%, and 120% of the design flow rate. The predicted static head coefficient is \( \sim 2.4\% \) higher than the calculated value at the impeller exit and \( \sim 6.2\% \) lower at the volute outlet (table 3). The analytical model accurately predicts the head performance of the impeller at design conditions, but it
overestimates the losses in the diffuser and volute. This finding is consistent with the observation that in the analytical model the distributed mixing losses occurring in the diffuser and the volute are concentrated at the inlets of these components. At low flow rates, the head coefficient calculated at the impeller exit overpredicts the pump performance by more than 7.9%. This is probably due the absence of backflow and leading-edge stall effects in the analytical model.

![Graph](image1.png)

**Figure 5.** Comparison between the measured head coefficient and hydraulic efficiency of the VAMPIRE pump for different values of the impeller clearance and the predictions of the analytical model corrected for tip leakage effects [21].

![Graph](image2.png)

**Figure 6.** Comparison of the experimental and numerical static head characteristics $\Psi_s(\phi)$ of the VAMPIRE pump.

Table 3 illustrates the comparison between the modulus of the non-dimensional average azimuthal flow velocity $v_2$ at the impeller exit (station 2). The difference between the results of the analytical model and the simulations is ~4% at higher flows (100% and 120% of design conditions) and becomes 1% and 9% at 80% and 60% of the design flow, respectively. Table 3 reports also the comparison of the
deviation angle $\bar{\varphi}_2$, which is underestimated by the analytical model at the operating conditions taken into consideration. The maximum discrepancy on deviation angle is -4% at 120% of the design flow rate.

\begin{table}
\centering
\begin{tabular}{c|ccc}
\hline
$\Phi/\Phi_D$ & $e = \frac{\psi_{s,CFD} - \psi_{s,EXP}}{\psi_{s,EXP}}$ & $e = \frac{\psi_{s,AN} - \psi_{s,CFD}}{\psi_{s,CFD}}$ & $d_{\bar{\varphi}} = \frac{\bar{\varphi}_{AN} - \bar{\varphi}_{CFD}}{\bar{\varphi}_{CFD}}$ \\
\hline
60% & +2.6% & +1.2% & -0.6% \\
80% & +0.9% & -1.8% & -0% \\
100% & +2% & -1.3% & -0% \\
120% & -2.7 & -5.6% & -0% \\
\hline
\end{tabular}
\caption{Error $e$ of the numerical and experimental head at the clearances under consideration.}
\end{table}

\begin{table}
\centering
\begin{tabular}{c|cccc}
\hline
$\Phi/\Phi_D$ & $d_{s,imp}$ & $d_{s,vol}$ & $d_V$ & $d_{\bar{\varphi}}$ \\
\hline
60% & +7.9 & +21.7 & +9.2 & -1.0 \\
80% & +8.6 & +12.9 & +1.0 & -1.2 \\
100% & +2.4 & -0.8 & -4.1 & -3.6 \\
120% & +0.8 & -6.2 & -4.2 & -4.0 \\
\hline
\end{tabular}
\caption{Percentage difference, $d$, of the analytical model with respect the numerical simulations for the pump performance, deviation angle, and impeller circumferential velocity at its exit section.}
\end{table}

Table 4 shows a comparison between the circumferential velocities in the blade channels obtained from the analytical model and the numerical simulations at an axial section near the impeller exhaust in the absolute (statoric) reference frame. At 80% of the flow rate the velocities have the same trends. The circumferential velocity is minimum near the suction side and the root of the blades and increases near the tip of both the suction and pressure sides of the blades. At this operating condition, leading edge separation was observed in the numerical simulation, and it affects each blade channel. At design flow rate, the velocity trend is similar. However, the numerical simulations show that leading-edge stall is occurring in the three blade channels highlighted in the table with the letter S. For these blade channels the velocity trends are not similar. At 120% of the design flow rate, the velocities have the same trends,
but the analytical model overestimates the intensity of the circumferential velocity along the blade pressure side. Therefore, it appears that the analytical model is capable to capture the structure of the circumferential velocity field where the slowly varying hypothesis in $z$ for the helical pitch and the backswep angle are more accurately respected.

**Table 4.** Comparison of the circumferential velocities in the blade channels obtained with the analytical model with the numerical simulation at an axial section near the impeller exhaust.

| $\phi / \phi_D$ | Circumferential velocity for an axial section at 75% of the impeller height ($z_{ext}$) in the volute reference frame. |
|-----------------|---------------------------------------------------------------------------------------------------------------|
|                 | Analytical Model                                           | Numerical Simulation                                           |
| **80%**         | ![Analytical Model](image1)                                | ![Numerical Simulation](image2)                               |
| **100%**        | ![Analytical Model](image3)                                | ![Numerical Simulation](image4)                               |
| **120%**        | ![Analytical Model](image5)                                | ![Numerical Simulation](image6)                               |
4. Conclusions
The results of the present research indicate that the numerical simulations of the flow in a centrifugal turbopump carried out by means of an incompressible URANS solver adequately match the experimental findings for flow rates equal to 60%, 80%, 100%, and 120% of the design value and impeller relative clearances equal to 0%, 5.6%, and 11.2%. The main difference occurs at 120% of the design flow rate, where the error in the estimate of the static pressure gain is $-5.6\%$ for a relative $c_\% = 11.2$. For all of the other operating conditions the relative errors range between $-2.7\%$ and $+2.6\%$. Thus, the numerical simulations can be considered validated and, moreover, confirm the empirical linear relation reported in [5].

The comparison with the results of the simulations at zero-clearance indicate that the analytical model:

- accurately predicts the simulated performance of the pump at and above design flow rate (100% and 120% of the design value);
- overestimates the simulated performance of the pump at lower flow rates (80% of the design value), most likely because under these conditions backflow and leading-edge separation losses, not included in the model, become significant;
- underestimates the simulated value of the surface averaged circumferential flow velocity at the impeller exit by up to $-4\%$ at 100% and 120% of the design flow rate;
- overestimates the simulated value of the flow velocity at the impeller exit by 1% at 80% of the design flow rate;
- underestimates the simulated value of the surface averaged deviation angle of the flow at the impeller exit by 1% to 4% over the range of operating conditions under investigation (from 80% to 120% of the design flow rate);
- agrees with the simulated trend of the circumferential velocity field in the blade channels over the range of operating conditions under investigation (from 80% to 120% of the design flow rate).

Hence the proposed analytical model captures with reasonable accuracy the characteristics of the internal flow in mixed-flow centrifugal turbopumps mostly affecting their hydraulic performance. The present study confirms therefore its capability to provide useful engineering solutions to the preliminary design problem of these machines at a negligible fraction of the computational cost required by 3D numerical simulations.

Acknowledgments
The authors would like to express their gratitude to Riccardo Simi and Gabriele Brotini, researchers at Pisa University, Italy, and Dr. Naoki Nagao, visiting researcher at Pisa University from JAXA, Japan, for their constructive comments and friendly encouragement. Finally, the close interaction with Dr. Bella G. Nokkä and her colleagues at GMYS-Space was greatly appreciated.

Nomenclature

| Symbol | Description |
|--------|-------------|
| $b$    | impeller outlet width |
| $N$    | number of blades     |
| $z$    | axial coordinate     |
| $w$    | axial velocity       |
| $v$    | azimuthal velocity   |
| $V$    | velocity in the volute reference frame |
| $r$    | radial coordinate    |
| $\theta$ | azimuthal coordinate |
| $\tilde{u}$ | displacement flow |
| $\hat{u}$ | fully-guided flow |
| $\Omega$ | rotational speed |

Subscripts/Superscripts

- $AN$ analytical
- $EXP$ experimental
- $i$ impeller
- $num$ numeric

head coefficient, $\Psi_s$ defined with static pressure

stream function of the displacement flow

flow coefficient

turbopump efficiency
References

[1] Williams R W, Skelley S E, Chen W-C and Williams M 2001 Comparison of unshrouded impeller analysis and experiment 37th Joint Propulsion Conference and Exhibit

[2] Kachler T and Beaurain A 2005 VULCAIN X: Hydrogen/Oxygen Reusable Liquid Rocket Engine Demonstration 41st AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit Joint Propulsion Conferences (American Institute of Aeronautics and Astronautics)

[3] Kawashima H, Kurosu A, Kobayashi T and Okita K 2018 Progress of LE-9 Engine Development 2018 Joint Propulsion Conference AIAA Propulsion and Energy Forum (American Institute of Aeronautics and Astronautics)

[4] Furst R B, Douglass H W, Schmidt H, Keller R B, Campen H and Viteri F 1973 Liquid rocket engine centrifugal flow turbopumps (Cleveland, OH, United States: NASA Lewis Research Center)

[5] Valentini D, Pasini A, Pace G, Torre L and d’Agostino L 2013 Experimental Validation of a Reduced Order for Radial Turbopump Design 49th AIAA/ASME/SAE/ASEE Joint Propulsion Conference Joint Propulsion Conferences (American Institute of Aeronautics and Astronautics)

[6] Negishi H, Ohno S and Kobayashi T 2019 Numerical Study of Tip Clearance Effects in a Centrifugal Pump with Unshrouded Impeller for Liquid Rocket Engines AIAA Propulsion and Energy 2019 Forum AIAA Propulsion and Energy Forum (American Institute of Aeronautics and Astronautics)

[7] Arnone A, Boncinelli P, Munari A and Spano E 1999 Application of CFD techniques to the design of the Ariane 5 turbopump 14th Computational Fluid Dynamics Conference Fluid Dynamics and Co-located Conferences (American Institute of Aeronautics and Astronautics)

[8] Rapposelli E, Cervone A, d’Agostino L and Agostino 2002 A New Cavitating Pump Rotordynamic Test Facility 38th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit (At Indianapolis, IN, USA)

[9] Kim C, Kim S, Choi C-H and Baek J 2017 Effects of inducer tip clearance on the performance and flow characteristics of a pump in a turbopump Proc. Inst. Mech. Eng. Part J. Power Energy 231 398–414

[10] Negishi H, Ohno S, Ogawa Y, Aoki K, Kobayashi T, Okita K and Mizuno T 2017 Numerical Analysis of Unshrouded Impeller Flowfield in the LE-X Liquid Hydrogen Pump 53rd AIAA/SAE/ASEE Joint Propulsion Conference AIAA Propulsion and Energy Forum (American Institute of Aeronautics and Astronautics)

[11] d’Agostino L, Torre L, Pasini A, Baccarella D, Cervone A and Milani A 2008 A Reduced Order Model for Preliminary Design and Performance Prediction of Tapered Inducers: Comparison with Numerical Simulations 44th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit Joint Propulsion Conferences (American Institute of Aeronautics and Astronautics)

[12] d’Agostino L, Pasini A and Valentini D 2011 A Reduced Order Model for Preliminary Design and Performance Prediction of Radial Turbopumps 47th AIAA/ASME/SAE/ASEE Joint Propulsion Conference & Exhibit Joint Propulsion Conferences (American Institute of Aeronautics and Astronautics)

[13] Ashihara K and Goto A 2001 Turbomachinery Blade Design Using 3-D Inverse Design Method, CFD and Optimization Algorithm GT2001 (Volume 1: Aircraft Engine; Marine; Turbomachinery; Microturbines and Small Turbomachinery)
[14] Zangeneh M, Goto A and Takemura T 1994 Suppression of Secondary Flows in a Mixed-Flow Pump Impeller by Application of 3D Inverse Design Method: Part 1 — Design and Numerical Validation GT1994 (Volume 1: Turbomachinery)

[15] Fu Y, Fan M, Pace G, Valentini D, Pasini A and d’Agostino L 2018 Experimental and Numerical Study on Hydraulic Performances of a Turbopump With and Without an Inducer FEDSM2018 (Volume 3: Fluid Machinery; Erosion, Slurry, Sedimentation; Experimental, Multiscale, and Numerical Methods for Multiphase Flows; Gas-Liquid, Gas-Solid, and Liquid-Solid Flows; Performance of Multiphase Flow Systems; Micro/Nano-Fluidics)

[16] Kergourlay G, Younse M, Bakir F and Rey R 2007 Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison Int. J. Rotating Mach.

[17] Yan P, Chu N, Wu D, Cao L, Yang S and Wu P 2016 Computational Fluid Dynamics-Based Pump Redesign to Improve Efficiency and Decrease Unsteady Radial Forces J. Fluids Eng. 139

[18] d’Agostino L, Valentini D, Pasini A, Torre L, Pace G and Cervone A 2017 On the Preliminary Design and Performance Prediction of Centrifugal Turbopumps—Part 2 Cavitation Instabilities and Rotordynamic Effects in Turbopumps and Hydroturbines: Turbopump and Inducer Cavitation, Experiments and Design ed L d’Agostino and M V Salvetti (Cham: Springer International Publishing) pp 157–78

[19] d’Agostino L, Angelo P, Pace G, Valentini D, Lucio T and Cervone A 2012 A Reduced Order Model for Optimal Centrifugal Pump Design 14th International Symposium on Transport Phenomena and Dynamics of Rotating Machinery, ISROMAC 2012 (Honolulu, HI, USA)

[20] d’Agostino L, Valentini D, Pasini A, Torre L, Pace G and Cervone A 2017 On the Preliminary Design and Performance Prediction of Centrifugal Turbopumps—Part 1 Cavitation Instabilities and Rotordynamic Effects in Turbopumps and Hydroturbines: Turbopump and Inducer Cavitation, Experiments and Design ed L d’Agostino and M V Salvetti (Cham: Springer International Publishing) pp 137–56

[21] Pace G, Pasini A, Torre L, Valentini D and d’Agostino L 2012 The Cavitating Pump Rotordynamic Test Facility at ALTA S.p.A.: Upgraded Capabilities of a Unique Test Rig Space Propulsion Conference (Bordeaux, France)

[22] Valentini D 2015 Modelling and Testing of Chemical Propulsion Rocket Subsystems Ph.D. Dissertation (Università di Pisa)

[23] Pace G, Valentini D, Torre L, Pasini A and d’Agostino L 2014 Experimental Characterization of Rotordynamic Forces Acting on Space Turbopumps Space Propulsion Conference (At Cologne, Germany)

[24] Bilanceri M, Beux F and Salvetti M V 2010 An implicit low-diffusive HLL scheme with complete time linearization: Application to cavitating barotropic flows Comput. Fluids - COMPUT FLUIDS 39 1990–2006

[25] Menter F R 1994 Two-equation eddy-viscosity turbulence models for engineering applications AIAA J. 32 1598–605