THE INFLUENCE OF SECONDARY FLOWS IN THE THRUST ACTING ON THE AXIS OF A RADIAL LOX PUMP

P. Beck¹ - B. Wagner*-O. Haidn¹

1) Institut für Turbomaschinen und Flugantriebe (Extraordinariat für Raumfahrtantriebe, TU München, 85748 Garching, Deutschland, paulo.beck@tum.de and haidn@ltf.mw.tum.de)
*Institute of Space Propulsion (Deutsches Zentrum für Luft- und Raumfahrt e.V. DLR, 74239 Lampoldshausen, Deutschland, bernd.wagner@dlr.de)

ABSTRACT

The authors investigate the influence of secondary flows in the thrust acting on the axis of a liquid oxygen radial pump with operating conditions like those of a Vinci upper-stage rocket engine. For a high circumferential Reynolds number, pressure and velocity distributions are obtained by varying geometrical parameters known to influencing the flow in the cavities between the impeller and the housing. The results obtained with the numerical method are compared with those calculated with formulae relying on the rotation factor \( k \). The Finite-Volume Method is applied to three-dimensional impeller-housing models. The results indicate the axial thrust estimated using the formulae returns overestimated values as compared with those computed with the numerical method, such fact suggesting that corrections might be needed when one considers applying the design guidelines to radial pumps driving liquid oxygen.

KEYWORDS

LIQUID ROCKET ENGINE, LIQUID OXYGEN PUMP, AXIAL THRUST, SECONDARY FLOWS

NOMENCLATURE

\( c \) Tangential fluid velocity
\( c_f \) Friction coefficient
\( CFL \) Courant number
\( d \) Diameter
\( \dot{E} \) Driving power
\( F \) Force
\( F_s \) Safety factor for the error estimator
\( g \) Gravity acceleration
\( GCI \) Grid Convergence Index
\( H \) Headrise
\( k \) Rotation factor
\( \dot{m} \) Mass flow rate
\( NPSH_r \) Net Positive Suction Head (required)
\( \sigma \) Order of convergence
\( p \) Pressure
\( Q \) Volume rate
\( r \) Radial distance, radius
\( r_{12} \) Grid refinement ratio
\( Re \) Reynolds number
$s_{\text{ax}}$ Distance between the impeller and the hub wall (same as “gap A”)
$t_{\text{ax}}$ Blockage distance
$u$ Circumferential fluid velocity
$q$ Ratio of radii
$x_{\text{ov}}$ Overlap distance
$\beta$ Angular velocity
$\omega$ Rotational speed
$\delta$ Angle
$\rho$ Density
$\mu$ Molecular viscosity
$\eta$ Efficiency
$\Omega_s$ Specific speed
$\psi$ Head coefficient
$\phi$ Scalar quantity
$\epsilon$ Error estimator

INTRODUCTION
The radial pumps of chemical rocket engines shall be able to ingest multiphase propellants at near or boiling point, and to deliver those to the engine at pressures exceeding that of the combustion chamber without fluctuations and at constant flow rate. In addition, the mechanical components, in particular bearings and seals, must be capable to withstand the effects of the axial forces resulting from unsteady flow phenomena and pressure distributions. In this context, the authors investigate by means of a numerical method the influence of secondary flows in the thrust acting on the axis of a radial pump which is dimensioned to deliver 33.7 kg/s of liquid oxygen (LO$_x$) at 66 bar to an expander cycle, Vinci similar upper-stage rocket engine. For a high circumferential Reynolds number, distinct pressure and velocity distributions are obtained by varying geometrical parameters that according to the literature are known to influencing the flow in the cavities between the impeller and the housing. The results obtained with the numerical method for the axial thrust are compared with those calculated using formulae relying on the so-called rotation factor $k$, which is estimated from gap-defining variables. The numerical method is applied to three-dimensional impeller-housing models, derived from a baseline model by varying selected gap distances that per the literature chiefly influence the axial thrust. The steady Navier-Stokes transport equations particularized for incompressible, turbulent flows are used to describe the fluid motion, and the turbulence quantities are evaluated using the SST $k$-$\omega$ turbulence model. The resulting system of equations is linearized using the Finite-Volume method, which is applied to a set of low-Reynolds unstructured grids having different refinement levels for verification purposes. The results indicate the numerical method is capable to capturing the secondary flows developing in the narrow gaps of the model. In addition, the axial thrust estimated using the formulae returns overestimated values as compared with those computed by the numerical method, such fact suggesting that corrections might be needed when one considers applying the design guidelines available in the literature to pumps driving high circumferential Reynolds number liquid oxygen.

State of the Art
The flow phenomena in a rotor-stator system have been investigated for many years. Fundamental research was carried out by Batchelor (1951) and Stewartson (1953), and followed by the intensive experimental work of Daily and Nece (1960). Kurokawa and Toyokura (1976) applied analytical methods to calculate the axial thrust in radial flow turbomachinery. Their work was supported by experiments, and included radial through-flow investigations. More recently, Kurokawa, Kamiyto, and Shimura (1994) calculated the axial thrust behavior of the first stage LOx-pump of the H-II launcher, and numerical studies with the Reynolds Stress turbulence model applied to variety of flow conditions were performed on rotor stator cavities by Poncet, Chauve, and Schiestel (2005a),
Poncet, Chauve, and Schiestel (2005b), and by Haddadi and Poncet (2009). Shimura et al. (Shimura, Kawasaki, Uchiumi, Kimura, Hayashi, et al., 2012; Shimura et al., 2013; Shimura, Kawasaki, Uchiumi, Kimura, & Matsui, 2012) concentrated on the dynamic behavior of the axial thrust with balancing devices and grooves on the stator walls. Current work by Barabas et al. (2015) investigated the pressure and velocity distributions at very high circumferential Reynolds numbers, which is also of interest for the radial LOx pump of rocket engines.

**AXIAL THRUST**

The axial thrust acting on the impeller of a radial pump is originated by the pressure distribution on the shroud and hub walls, by the impulse force originated by the flow being redirected from the axial to radial direction, and by the unbalanced and coupling forces. For the sake of obtaining a theoretical approximation for the axial thrust, those latter forces are unknown and therefore are neglected. In addition, it is assumed that the impulse force is small as compared to the force due to pressure distribution, even for high density fluids as water or LOx. As a consequence of these assumptions, the axial thrust results only from the integration of the pressure distribution on the shroud and hub walls such that

\[
F = 2\pi \int_{r_1}^{r_2} pr dr
\]  

In Eq. [1], \( p \) is the pressure and \( r \) is a radial coordinate, and the net axial thrust acting on the impeller is given by the difference between the forces acting on the rear and front shrouds. Observations in narrow gaps between stator and rotor (Daily & Nece, 1960) showed that the flow is dominated by viscous effects near the wall. According to Gülich (2010), by integrating Eq. [1] and introducing the rotation factor \( k \equiv c_u/\omega \) to allow losses due to those viscous effects, where \( c_u = \beta \cdot r \) is the tangential fluid velocity and \( \omega = \beta \cdot r \) is the circumferential fluid velocity (\( \beta \) is the angular velocity of the fluid in the core flow, and \( \omega \) is the rotational speed), the force acting on a wall can be expressed as

\[
F_o = \pi r^2 \left\{ (1 - q^2) \left[ \Delta p - \frac{p}{4} u_2^2 k^2 (1 - q^2) \right] \right\}
\]  

In the above equation, \( \Delta p \) is the pressure difference relative to the outlet pressure, \( u_2 \) is the circumferential fluid velocity at the impeller tip, and \( q \) stands for the ratio of radii \( r_1 \) of an adjacent housing wall to the impeller tip radius \( r_2 \).

Following Gülich (2010), the rotation factor \( k \) in Eq. [2] can be replaced by a constant rotation factor \( k_0 \) under the assumption there is no net flow through, and no inlet swirl such that

\[
k_0 = \left\{ 1 + \left[ \frac{1}{\cos \delta_R \left[ 1 - r_1^2 \right] \left( \frac{r_2}{r_1} \right)^5} + 5 \frac{1}{\tan \delta_R \left( \frac{r_2}{r_1} \right)^4} \left[ 1 + \frac{r_2 - r_1 \tan \delta_R}{r_1 \tan \delta_R} - \frac{r_2 - r_1 \tan \delta_R}{r_1 \tan \delta_R} \right] \left( \frac{c_{f,w}}{c_{f,R}} \right)^{1/2} \right] \right\}^{-1}
\]  

The flow with constant \( k_0 \) corresponds to a forced vortex (or a “solid body rotation”) with a constant angular velocity \( \beta \) of the fluid. Referring to Figure 1, in Eq. [3] \( \delta_w \) is the inclination angle of the housing wall adjacent to the shroud wall, \( \delta_R \) is the inclination angle of the the shroud wall, and \( r_1 \) assumes either \( r_1 \) or \( r_{sp} \) values. The sum of radius of the impeller \( r_2 \) and the gap between the impeller tip and the housing wall (“gap A”) at the outlet defines \( r_2 \), and the term \((c_{f,w}/c_{f,R})\) is the ratio of the housing to the impeller wall friction coefficients. The gap between the hub wall and the adjacent housing wall (“gap E”) defines \( s_{ax} \) which is the limiting value of \( t_{ax} \) when \( x_{ov} > 0 \) (i.e., \( t_{ax} = s_{ax} \) when \( x_{ov} = 0 \)).
In this study, the axial thrust computed with equations [2] and [3] is compared with the CFD results for a LOx pump model which is described in the following section.

THE LOx PUMP MODEL

The radial pump in consideration is dimensioned to deliver 33.7 kg/s of LOx at 66 bar to an expander cycle, Vinci similar upper-stage rocket engine, according to the operating conditions, geometrical parameters and resulting overall performance indicators that follow.

Operating Conditions

The working fluid is LOx at 90 K. The density is $\rho = 1140$ kg/m$^3$, and the molecular viscosity is $\mu = 1.9685E-4$ Pa s. The impeller of the pump is designed to achieve a headrise $H = 600$ m with rotational speed 18,000 RPM ($\omega = 1885$ s$^{-1}$) and mass flow $\dot{m} = 33.7$ kg/s. By estimating the overall efficiency $\eta = 0.75$, volumetric efficiency $\eta_v = 0.90$, and mechanical efficiency $\eta_m = 0.95$, one obtains a hydraulic efficiency $\eta_h = 0.88$. At the inlet, the absolute flow angle is $90^\circ$ and the total inlet pressure is $p_{in} = 6.25$ bar. The volume rate is $Q = 0.0296$ m$^3$/s , and the meridional velocity at the inlet is $c_{in} = 28.8$ m/s at the mean line of the impeller blade.

Geometrical Parameters

The impeller has 6 passages, each passage 8 mm wide at the trailing edge of the blades. Referring to Figure 1, the outer radius of the impeller is $r_2 = 60$ mm, the hub eye radius is $r_{hub} = 21$ mm, the eye radius is $r_{eye} = 27.4$ mm, and the shaft radius is $r_{shf} = 14$ mm. The housing and shroud wall angles are $\delta_w = \delta_R = 5.2^\circ$ respectively. At the leading-edge of the blades, the blade angle at the hub side is $\beta_{1 Hub} = 27^\circ$, the blade angle at the mean-line is $\beta_{1 ML} = 19^\circ$, and the blade angle at the shroud side is $\beta_{1 shf} = 16^\circ$. The (constant) blade angle at the trailing-edge is $\beta_{2} = 22^\circ$ (all angles are measured from the suction side of a blade, and are relative to the tangential direction).

Overall Performance

Considering the operating conditions and the geometrical parameters, the Reynolds number is $Re \equiv \rho \omega r_2^2 / \mu \approx 487$, the specific speed is $\Omega_s \equiv \omega \sqrt{Q} / (gh)^{3/2} = 0.48$, the specific diameter is $d_s \equiv d_2 [(gh)^{1/4} / Q^{1/2}] = 6.12$, and the head coefficient is $\psi \equiv gH / u_2^2 = 0.46$. For $\eta = 0.75$, those performance indicators are within the region of the $\Omega_S - d_s$ diagram compiled by Douglas et al.
(1973) that matches LOx turbopumps. The required driving power is $\dot{E} = 265$ kW, and the required net positive suction head is $NPSH_r = 56$ m.

MATHEMATICAL MODEL

Transport Equations
The steady state Navier-Stokes conservation equations for continuity and momentum applied to an incompressible flow describe the fluid motion (Panton, 2006). The turbulence quantities are computed with the transport equations of the SST $k$-$\omega$ turbulence model (Menter, 1994) which include the low-Reynolds viscous damping (“low-Re Corrections”), curvature correction (modifies the turbulence production term to sensitize the standard eddy-viscosity models to the effects of streamline curvature and system rotation), and a production limiter of the turbulence energy (in order to avoid the buildup of turbulent kinetic energy in the stagnation regions).

Boundary Conditions
At the inlet, the total pressure $p_{in} = 6.25$ bar is prescribed. This value amounts to the pressure boost from the inducer ($\sim 7.5\%$ of the difference between the design pressure rise $p_H = 66.2$ bar and the suction pressure $p_{sc} = 2.5$ bar) and the suction pressure. At the outlet, the initial static pressure is set to 48.6 bar such that the target mass flow rate $\dot{m} = 33.7$ kg/s is achieved upon convergence of the numerical solution (the difference between the target and the computed mass flow rates is used to adjust the outlet static pressure at every iteration with help of Bernoulli's equation). The walls are non-slip, and the moving reference frame model is assigned to the cells of the LOx zone with absolute rotational speed of 18,000 RPM around the axis. The hub and shroud walls are assigned absolute rotational speed, and the impeller walls are assigned rotational speed relative to the LOx zone. In the latter walls, the rotational speed is set to zero. At the inlet and outlet boundaries, the turbulent intensity is set to 5%, and the turbulent viscosity ratio set to 10. In Figure 2, the meridional section of the impeller, the blade geometry, the shroud and hub walls, and the boundary conditions are pictured.

Numerical Method
The Finite-Volume Method (FVM) as described by Versteeg and Malalasekera (2007) is adopted to linearize the system of transport equations. The grids are unstructured with tetrahedral cells, and designed such that the dimensionless near-wall distance is $y^+ \sim 5$. The number of layers spanning the prismatic boundary layer is 30, and the geometrical growth rate is 1.2. In Figure 3, the intermediate grid is displayed, where the darker regions mean clustering of nodes or prismatic layers. The solver is the ANSYS® Fluent Release 16.2, and the flow and turbulence quantities are solved by coupling the pressure-based continuity and momentum equations. The velocity formulation is absolute, and the solution is converged to the steady state. The pressure-velocity scheme is the coupled one, gradients compute with the least squares cell based scheme, the pressure-interpolation scheme is second order, and the second-order upwind scheme is selected for discretization of the momentum, turbulent kinetic energy, and specific dissipation rate equations. The scalar AMG (“Algebraic Multi-Grid”) method with default settings is used for the solution of the linearized equations system of transport equations, with Courant number $CFL = 25$, explicit relaxation factor for momentum and pressure is 0.5, 0.4 for the turbulent kinetic energy and specific dissipation rate, and 0.1 for the turbulent viscosity.
RESULTS

Grid Verification
The estimation of the numerical uncertainty and grid independence of the results follows the generalized Richardson’s Extrapolation Method (Roache, 1998). In this method, the asymptotic value \( \phi_0 \) of a scalar \( \phi \) is given by

\[
\phi_0 \cong \phi_1 + (\phi_1 - \phi_2) / (r_{1,2}^O - 1)
\]

[4]

where \( \phi_1 \) and \( \phi_2 \) are the values computed with the finest and intermediate grids (refinement wise) respectively, \( r_{1,2} \) is the refinement ratio between those grids, and \( O \) is the computed order of convergence of the grid set which depends on the spatial discretization method. Considering a grid set composed of three grids with decreasing refinement levels (finest, intermediate, and coarsest), the refinement ratio \( r_{1,2} \) for a three-dimensional domain is given by

\[
r_{1,2} = [(\text{# of cells of the finest grid}) / (\text{# of cells of the intermediate grid})]^{1/3}
\]

[5]

The computed (or observed) order of convergence \( O \) is obtained by iteratively solving the equation

\[
\varepsilon_{2,3} % / (r_{2,3}^O - 1) = r_{1,2}^O \left[ \varepsilon_{1,2} % / (r_{1,2}^O - 1) \right]
\]

[6]

where the subscript “3” refers to the coarsest grid. In the previous equation, the Richardson Error Estimator is computed with

\[
\varepsilon_{n,n+1} % = 100(\phi_n - \phi_{n-1}) / \phi_{n-1}
\]

[7]

where the subscript \( n = [1,2,3] \) indicates the grid refinement level. The grid convergence index for the finest grid is calculated with

\[
GCI_{1,2} % = F_s \left| \varepsilon_{1,2} % \right| / (r_{1,2}^O - 1)
\]

[8]
where \( F_s = 1.25 \) is as a factor of safety over the error estimator given by Eq. [7]. The asymptotic value \( \phi_0 \) is considered to be grid independent when \( \alpha/r_{1,2}^O \approx 1 \), where \( \alpha = GCI_{1,2}/GCI_{1,2} \), and \( GCI_{1,2} \) is obtained in the same way as \( GCI_{1,2} \) but using the respective refinement ratio and error estimator considering the intermediate and coarsest grids.

For the purpose of uncertainty verification and grid independency of the results, a set of three computational grids is adopted (finest, intermediate, and coarsest refinement levels). The scalar \( \phi \) in Eq. [4] is the area-average static pressure at the outlet boundary. The results compiled in Table 1 indicate that the grid set is within the region of asymptotic convergence (i.e., \( \alpha/r_{1,2}^O \approx 1 \)). In addition, the computed order of convergence is \( O \approx 2 \), such value being expected as the discretization is second-order in space. The uncertainty of the asymptotic value is 0.02% for the finest grid, and 0.04% for the intermediate grid.

### Table 1: Uncertainty verification and grid independency of the results

<table>
<thead>
<tr>
<th>Grid n</th>
<th># of Cells</th>
<th>( \phi ) (outlet static pressure) (Pa)</th>
<th>( r_{n,n+1} )</th>
<th>( GCI_{n,n+1} ) %</th>
<th>( \alpha )</th>
<th>( \phi_0 ) (Pa)</th>
<th>( \alpha/r_{1,2}^O )</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>22691755</td>
<td>6613583</td>
<td>1.42</td>
<td>0.02</td>
<td>2.0517</td>
<td>6612624</td>
<td>1.000000</td>
</tr>
<tr>
<td>2</td>
<td>7850932</td>
<td>6614606</td>
<td>1.44</td>
<td>0.04</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>3</td>
<td>2589870</td>
<td>6616856</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

### Axial thrust

In order to analyze the effects of the secondary flows on the axial thrust, i.e., the flows in the vicinity of gaps “A” and “E”, grids with intermediate refinement level are adopted. In addition, the effect of coupling of the flow in the impeller sidewall gap and the main flow at the impeller outlet (i.e., the axial thrust and pressure distribution) is investigated by changing gap A and gap E for a given ratio \( x_{ov}/s_{ax} \).

In Table 2 the geometrical parameters are identified, as well as the recommended range for each one according to Gülich (2010). The ratio \( x_{ov}/s_{ax} \), and the aspect ratio gap A/r_2 had to be relaxed as the impeller radius \( r_2 \) is bounded by the operating conditions and the intended overall performance, and the overax \( x_{ov} \) is limited by the thickness of the impeller walls. The limiting factor of gap A is the resulting grid size and grid quality – the narrow region defining gap A requires a high density of nodes which causes the number of cells of the computational domain to become exceedingly high considering the availability of computational resources.

### Table 2: geometrical parameters of the models

<table>
<thead>
<tr>
<th>( s_{ax}/2r_2 ) (0.015 ~ 0.04)</th>
<th>( s_{ax} ) (gap E) (mm)</th>
<th>( x_{ov}/s_{ax} ) (2~4)</th>
<th>gap A/r_2 (0.007 ~ 0.01)</th>
<th>gap A (mm)</th>
<th>Model</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0150</td>
<td>1.8</td>
<td>1.7</td>
<td>0.0100</td>
<td>0.60</td>
<td>B</td>
</tr>
<tr>
<td>0.0275</td>
<td>3.3</td>
<td>0.6</td>
<td>0.0350</td>
<td>2.10</td>
<td>C</td>
</tr>
<tr>
<td>0.0275</td>
<td>3.3</td>
<td>0.6</td>
<td>0.0175</td>
<td>1.05</td>
<td>D</td>
</tr>
<tr>
<td>0.0275</td>
<td>3.3</td>
<td>0.6</td>
<td>0.0131</td>
<td>0.79</td>
<td>E</td>
</tr>
<tr>
<td>0.0275</td>
<td>3.3</td>
<td>0.6</td>
<td>0.0100</td>
<td>0.60</td>
<td>F</td>
</tr>
<tr>
<td>0.0400</td>
<td>4.8</td>
<td>0.4</td>
<td>0.0350</td>
<td>2.10</td>
<td>G</td>
</tr>
</tbody>
</table>

1 – Recommended ranges by Gülich (2010)

In Eq. [2], the \( \Delta p \) term is computed with

\[
\Delta p = \rho g H (1 - \psi/4\eta) (\eta_h/\eta)
\]

where the headrise, the head coefficient, and the efficiencies are computed from the simulation results \( H = 673 \text{ m}, \psi = 0.52, \eta_h = 0.82, \text{ and } \eta = 0.78 \). In Eq. [3], the values come from the geometrical parameters, and the ratio \( c_{f,w}/c_{f,R} \) is assumed to be the unity. In Table 3, the numerical
results are compared with the analytical results according to Equations [2] and [3]. The results indicate that the analytical results are consistently overestimated, in particular when the ratios gap $A/r_2$ and $x_{ov}/s_{ax}$ are small (Model F). Possible causes for this discrepancy are the underlying simplifications embedded in obtaining Eq. [3] originally developed by Zilling (1973), the impulse force not being accounted, and the hypothesis the boundary layers are separated by a central rotating core (Batchelor, 1951) which implies that $k$ is constant. The impulse force is estimated from theory as $F_i = \rho Q c_{1m} \approx 1012$ N, about 2.5% of the average thrust acting on the shroud wall in the axial direction. The resulting thrust in Table 3 ([iv]-[iii]) points towards the inlet side.

Table 3: the axial thrust – comparison of simulation versus analytical results

<table>
<thead>
<tr>
<th>Result source</th>
<th>Axial thrust (N)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Model B</td>
</tr>
<tr>
<td>Simul. hub wall [i]</td>
<td>40298</td>
</tr>
<tr>
<td>Eq. [2] ($r_i = r_{sp}$) [ii]</td>
<td>38657</td>
</tr>
<tr>
<td>Simul. shroud wall [iii]</td>
<td>49865</td>
</tr>
<tr>
<td>Eq. [2] ($r_i = r_d$) [iv]</td>
<td>49238</td>
</tr>
<tr>
<td>[iii]-[i]</td>
<td>9567</td>
</tr>
<tr>
<td>[iv]-[iii]</td>
<td>10581</td>
</tr>
<tr>
<td>$\Delta %$ Eq. [2] – Simul.</td>
<td>10.6%</td>
</tr>
</tbody>
</table>

In Figure 4, the rotational factor $k$ is plotted against the normalized radial distance $(r - r_i)/(r_2 - r_i)$ for both shroud (a) and hub walls (b). Models C and G deviate most at the upper limiting radius, thus indicating that the secondary flow through gap A plays a significant role in the estimation of the axial thrust.

![Figure 4: the profiles of $k$ at the middle of the cavity defined by gap E](image)

In Table 4, the computed and theoretical values of the rotational factor are presented. As one may observe, the theoretical values are overestimated as compared to the computed ones (obtained by approximating the $k$ profiles by a high order polynomial and calculating the average value), but not as much as in the axial thrust calculations.
Table 4: the rotational factor $k$ – comparison of simulation versus theoretical results

<table>
<thead>
<tr>
<th>gap (wall)</th>
<th>Rotational factor</th>
<th>Model B</th>
<th>Model C</th>
<th>Model D</th>
<th>Model E</th>
<th>Model F</th>
<th>Model G</th>
</tr>
</thead>
<tbody>
<tr>
<td>gap E (hub wall)</td>
<td>$k_0$</td>
<td>0.4765</td>
<td>0.4493</td>
<td>0.4594</td>
<td>0.4618</td>
<td>0.4638</td>
<td>0.4381</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>0.4660</td>
<td>0.4384</td>
<td>0.4466</td>
<td>0.4461</td>
<td>0.4476</td>
<td>0.4148</td>
</tr>
<tr>
<td></td>
<td>$\Delta % k_0 - k$</td>
<td>2.3</td>
<td>2.5</td>
<td>2.9</td>
<td>3.5</td>
<td>3.6</td>
<td>5.6</td>
</tr>
<tr>
<td>gap F (shroud wall)</td>
<td>$k_0$</td>
<td>0.4772</td>
<td>0.4505</td>
<td>0.4608</td>
<td>0.4633</td>
<td>0.4653</td>
<td>0.4402</td>
</tr>
<tr>
<td></td>
<td>$k$</td>
<td>0.4641</td>
<td>0.4444</td>
<td>0.4580</td>
<td>0.4590</td>
<td>0.4624</td>
<td>0.4273</td>
</tr>
<tr>
<td></td>
<td>$\Delta % k_0 - k$</td>
<td>2.8</td>
<td>1.4</td>
<td>0.6</td>
<td>0.9</td>
<td>0.6</td>
<td>3.0</td>
</tr>
</tbody>
</table>

In Figure 5, the profiles of the normalized meridional velocity $c_m^*$ at $y = 0.045$ m ($r - r_i)/(r_2 - r_i) \approx 0.65$ in Figure 4) are presented for both shroud (a) and hub (b) walls. One may observe that the velocity profiles place the flow within the turbulent regime for small and large gap $A/r_2$ values (Daily & Nece, 1960). In particular for Model B, where gap $A/r_2 = 0.01$, the profile clearly characterizes secondary, turbulent flows through small clearances. In Figure 6, the profiles of $k$ against the normalized ($\Delta z/s_{ax}$) gap F (shroud wall, plot (a)), and gap E (hub wall, plot (b)) are plotted. It may be observed that the distortion of the profiles is more noticeable at the rotating walls.
Pressure Contours and Streamlines

In Figure 7 the static pressure contours and streamlines are depicted at the meridional section of the models. The rake that seeds the streamlines is the same in all plots, and therefore it is possible to compare the morphology of the secondary flows as $A/r_2$ and $s_{ax}/2r_2$ vary. The recirculation zones develop quite freely at the rim of the impeller, except in Models B and F which are the ones with smaller $A/r_2$ ratio, and Model G displays a shift of the higher pressure region towards the origin. A mesh with a very high node density was used to compute Model B ($x_{ov}/s_{ax} = 1.7$) in order to obtain detailed pressure contours and secondary flow streamlines.
CONCLUSIONS

A generic radial LOx, Vinci-like pump has been investigated using a numerical method on full three-dimensional domains. Geometrical parameters at the rotor-stator cavities have been varied in order to observe the influence of them on the secondary flows, and the resulting axial thrust as part of a fundamental understanding of the processes during predesign studies. Due to the lack of reliable experimental data for LOx configuration at this rotational speed, the values for axial thrust are compared to results obtained with correlations taken from the literature. A consistent overestimation of the axial thrust can be found in the analytical results for any computed geometrical configuration, being this conclusion valuable when one considers the design process as geometrical changes in the rotor-stator cavities might be justified with analytical correlations by producing conservative estimates. In addition, the analysis of the rotation factor profile offers insights that might influence the design of LOx radial pumps. Future steps towards a realistic configuration are the consideration of a net flow through, cavitation, passive flow control methods, and the application of axial thrust balancing devices. Again, both activities will be accompanied by both numerical and analytical investigations.

ACKNOWLEDGEMENTS

P. Beck thanks the financial support from CNPq through the post-doctoral fellowship grant PDE 202973/2015-8, and B. Wagner thanks the Bavarian technology project KonRAT (“Komponenten von Raketentriebwerken für Anwendungen in Transportsystemen der Luft- und Raumfahrt”). This project within the Ludwig Bölkow Campus is financed by Bavarian Ministry of Economic Affairs and Media, Energy and Technology under contract No. LABAY83E.

REFERENCES


