



Agapi Bakogianni¹

Centre for Propulsion and Thermal Power,
Cranfield University,
College Road,
Cranfield, Bedfordshire MK43 0AL, UK
e-mail: agapi.bakogianni@cranfield.ac.uk

David John Rajendran

Centre for Propulsion and Thermal Power,
Cranfield University,
College Road,
Cranfield, Bedfordshire MK43 0AL, UK
e-mail: d.rajendran@cranfield.ac.uk

Eduardo Anselmi Palma

Centre for Propulsion and Thermal Power,
Cranfield University,
College Road,
Cranfield, Bedfordshire MK43 0AL, UK
e-mail: e.a.anselmi@cranfield.ac.uk

Vassilios Pachidis

Centre for Propulsion and Thermal Power,
Cranfield University,
College Road,
Cranfield, Bedfordshire MK43 0AL, UK
e-mail: v.pachidis@cranfield.ac.uk

Chloe Jo Palmer

Rolls-Royce plc,
P.O. Box 31, Moor Lane,
Derby DE24 8BJ, UK
e-mail: Chloe.Palmer@Rolls-Royce.com

Insights Into the Cavitation Behavior of a Boundary Layer Pump

Several researchers in literature have claimed that boundary layer pumps have an increased resistance to cavitation because of the absence of blades. Their conclusion was obtained analytically by applying the simple assumption of uniform radial velocity at the entrance of all the channels. This study considers the effect of flow distribution between the channels and assesses the full 3D geometry of a boundary layer pump which was tested in an experimental campaign by Morris. The pump's flow field is obtained with steady-state multiphase 3D-RANS simulations in which the inlet total pressure is gradually reduced. For each inlet total pressure point, the vapor formation is examined, and hence the cavitation mechanisms are analyzed, and the head drop curve is obtained. Results show that the mass flow distribution in the channels is highly nonuniform, which heavily affects vapor formation areas. There is a 127% difference between the lowest and highest channel mass flow. Less mass flow is entering the channels closer to the entrance of the pump, because this requires the flow to turn almost 90 deg from the axial to the radial direction. The viscous forces accelerate less mass more easily, and this leads to up to 22% lower pressure values in the low flow channels. Therefore, vapor starts forming in these channels, and the further the net positive suction head (NPSH) reduces, it sequentially propagates to adjacent channels toward the back like a domino and expands its volume inside each gap. [DOI: 10.1115/1.4069454]

1 Introduction

The Tesla pump is a configuration of a pump invented and patented by Nikola Tesla [1]. Its rotor comprises of a stack of parallel, flat, thin disks with small gaps between them. Each disk has a hole at the bore acting as the rotor's inlet. The fluid enters the machine axially, then turns, and enters the small gaps between the disks. Inside each gap, the shear forces from the rotating disks act on the fluid, accelerating it tangentially, hence the alternative names boundary layer pump and shear force pump. Due to the rotation of the fluid, centrifugal force also pushes it radially toward the disk outlet. Thus, the fluid follows a spiral trajectory inside the rotor and is then collected from the volute and exits the machine.

Previous research about Tesla pumps is limited and focuses on the rotor's modeling and performance using analytical and numerical techniques [2–4]. Many studies are solving axisymmetric

Navier–Stokes equations in a single gap by implementing approximation functions for the velocity profiles of both Tesla pumps and turbines [5–7]. It is documented that these simple numerical models can sufficiently predict the rotor only performance with minor effect from the implementation of the different recommended velocity profiles approximations. However, one-dimensional methods poorly predict the losses inside the volute and the interface between rotor and volute [8]. Full pump experimental studies are parametric and explorative and have only obtained relatively low efficiencies since there is no structured design methodology to be followed [9–12]. So far, Tesla pumps are used in a few applications where efficiency can be compromised in favor of other characteristics related to the absence of blades and their working principles. These are mainly in cement and oil & gas industry since they operate well with slurries and solid corrosive particles [13] and medicine for blood pumping with lower risk of hemolysis [14]. Despite the low efficiency, previous research has proposed that it might have a cavitation advantage due to its working principles. Centrifugal pumps tend to show an abrupt increase in local velocity and a corresponding decrease in pressure at the leading edge of the blade, which can result in vapor formation and

¹Turbo Expo: Turbomachinery Technical Conference & Exposition (GT2025), June 16–20, 2025, GT2025.

¹Corresponding author.

Manuscript received June 30, 2025; final manuscript received July 16, 2025; published online September 24, 2025. Editor: Jerzy T. Sawicki.

cavitation. However, Tesla pumps do not have blades, and the fluid is accelerated gradually by the viscous forces. Balje used analytical methods and empirical formulas to model the flow inside both a Tesla and a centrifugal pump and claimed that Tesla pumps can obtain about 10–20 times higher suction specific speeds than centrifugal pumps which implies a significant cavitation advantage [15]. He also suggested that the use of such a pump for aerospace fuel systems could be beneficial because the inducer part can be omitted to reduce weight. This advantage becomes even more relevant with the use of novel fuels like liquid hydrogen which is prone to phase change because of its storage conditions. Additional applications could be gas turbine thermal management systems where temperatures closer to the boiling point can be tolerated; hence, a lighter and more compact subcooler is required. However, this study draws its conclusions by assuming equally distributed mass flow between all channels, purely radial rotor entrance velocity, and same pressure field inside all channels. It calculates the required net positive head assuming that the rotor entrance is the lowest pressure region of the machine.

Few studies have addressed cavitation in Tesla pumps. One of the parametric experimental studies on water attempted a second case with reduced inlet pressure and noticed no change in the pump's performance. However, the lower inlet pressure tested was around seven times higher than the water vaporization pressure at the test temperature, no results were documented, and no head drop curve was obtained [10]. Another experimental study investigated vibration induced cavitation by operating a water Tesla pump under vibrations with frequencies up to 2000 Hz but no inlet pressure decrease. The vibrations did not significantly affect the performance [12]. The few examples of a full Tesla pump computational fluid dynamics (CFD) model that exist in literature are single phase and focus on capturing the flow field and the loss mechanisms [16]. Overall, there is a lack of studies on Tesla pump head drop curve, its cavitation mechanisms, and its two-phase flow field while cavitating.

The present study investigates cavitation performance in a Tesla pump using CFD techniques to obtain the flow field and locate the vapor formation areas. The geometry used is the one of a parametric study conducted by Morris which indicated one of the highest experimental efficiencies in literature, around 40% [10]. The results are used to gain an explanation of the Tesla pump cavitation mechanisms. Additionally, by gradually dropping the inlet pressure in the simulation boundary conditions and reviewing the results, the head drop curve is acquired.

2 Materials and Methods

The present study assesses the full Tesla pump geometry from Morris' experimental campaign with 3D-RANS steady-state CFD using ANSYS CFX software. The experimental Tesla pump geometry has the same arrangement as most of the existing Tesla pumps in literature. The disks have aligned holes from which rods pass through. Between two consequent disks and through the rod, there are thin spacer rings that maintain the interdisk gap. The rods are all connected to two thicker plates at either side of the stack of disks, and this comprises the whole rotor assembly. In Morris' study, there are 17 rods to maintain robustness, and the rotor assembly is inserted into a housing with the volute geometry.

The main dimensions of the geometry of interest are summarized in Table 1. The current study omits the secondary flow path between the rotor and the housing and the rods and spacers geometries. Thus, the interdisk gaps are purely cylindrical. The pump is modeled in three domains. The first domain is the inner, which is of cylindrical shape and lies inside the bore of the disks. The second domain is the rotor which consists of all the interdisk gaps. The third domain is the volute. A pipe-like extension is added to the pump inlet so that the boundary conditions do not affect the flow field near the first disks. This ensures that the flow is fully developed before turning and entering the channels.

The pipe, inner, and rotor domains are discretized with hexahedral mesh, and the volute is discretized with tetrahedral mesh. The

Table 1 Geometric characteristics of the Tesla pump

Parameter	Symbol	Value	Unit
Rotor inlet diameter	D_{in}	25.4	(mm)
Rotor outlet diameter	D_{out}	150.8	(mm)
Interdisk gap	d	0.152	(mm)
Disk thickness	t	0.127	(mm)
Volute base circle diameter	D_{vol}	155.8	(mm)
Volute width	b	47.303	(mm)
Number of channels	ch	170	

current study uses the same average element size for each domain as a previous numerical study by Palaveev et al. [16], which assessed the same geometry with single phase water, conducted a grid sensitivity study, and validated the results with the experimental ones from Morris [10]. However, a refinement is imposed at the inner-rotor domain interface to obtain matching sizes, smaller expansion factors, and stability for the two-phase flow. The gap axial divisions are reduced from 10 to 3 to obtain lower aspect ratio elements at the inner-rotor interface for a smaller computational size. This is important since the rotor comprises of 170 channels. After these modifications, the elements at this interface have an aspect ratio below 3 and then expand inside the adjacent domains with a rate of 1.2. Additionally, the maximum expansion factor in the inner domain has the value of 6, and the simulation is stable. The total number of elements is 51.1×10^6 , and the number of nodes is 28.9×10^6 . The modified mesh used in the current study is compared with the one from the validated Palaveev's study by plotting the characteristic curve of the pump in Fig. 1. The new mesh leads to a head rise which is in very good agreement with the previous results with a root mean squared error of 5%. The mesh is shown in Fig. 2.

The results are also compared with the experimental ones in Fig. 1, and the matching is sufficient considering that the experimental geometry includes 17 rods interfering with the rotor flow and a secondary flowpath for which no information was given in Morris' study. The slopes of the curves which reflect the pumping mechanism are nearly identical.

The fluid is a homogenous mixture of two phases of water, liquid, and vapor. Hence, there is no dispersed vapor phase, and the pressure and velocity fields are common for both phases. The liquid phase is incompressible with constant properties taken from REFPROP [17], while the vapor phase is modeled with real gas table data from IAPWS. The turbulence model is shear stress transport for both phases, and the area averaged y^+ value at the rotor walls is 28 which is acceptable since wall functions are automatically employed by

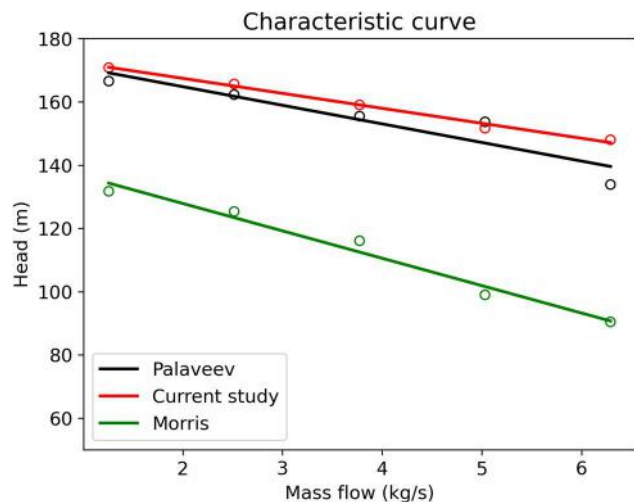


Fig. 1 Characteristic curves as obtained by Palaveev's numerical study, current study, and Morris' experimental study

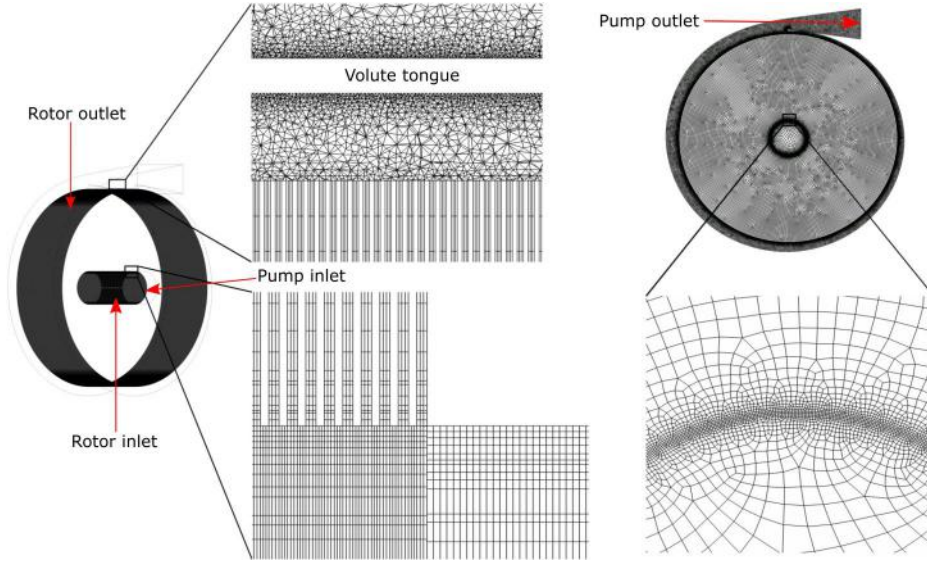


Fig. 2 Mesh details. Side view of the inlet and the volute tongue. Front view of inner-rotor interface.

CFX [18]. That justifies the matching of the characteristic curves for 10 and 3 gap axial divisions. The heat transfer model is set to isothermal at a constant temperature of 325 K, which is equal to the experimental one in Morris' test rig. In a two-phase simulation, isothermal modeling ensures that there is no temperature difference and heat transfer between the two phases. This is sufficient for a homogenous mixture model that mainly focuses on investigating the areas where vapor starts forming and how it propagates.

In the current isothermal model, phase change occurs when somewhere in the domain, the pressure drops below the saturation pressure, which is constant and for 325 K, equal to 13.5 kPa. The bubble growth and hence the vapor volume fraction are calculated by the simplified Rayleigh–Plesset model [19,20]. This is expressed in Eq. (1), where R is the radius of the bubble, p_b is the pressure inside the bubble, p_l is the pressure of the surrounding liquid, and ρ is the constant density of the liquid [21]

$$\frac{dR}{dt} = \frac{2}{3} \sqrt{\frac{p_b - p_l}{\rho}} \quad (1)$$

There are some investigations in literature regarding the heat transfer between the two phases. Vapor absorbs latent heat from the liquid phase to form. This leads to a local temperature drop at the surrounding liquid phase. Therefore, the local saturation pressure drops as well, and this effect inhibits cavitation. Similarly, the collapsing bubbles release the latent heat back and heat up the surrounding liquid. Certain researchers like Zwart, Gerber, and Belamri have modified the Rayleigh–Plesset model by adding coefficients to consider the evaporation and condensation effect. These coefficients are empirical, and there is high research interest about their values, which can be further corrected [22]. Previous studies have compared their numerical results with experimental data. Zwart, Gerber, and Belamri model was found to predict the vapor formation areas and the required net positive suction head (NPSH) very well, but overpredicted the local vapor volume fraction and underestimated the head [23]. Another study does not use coefficients and models the heat transfer between the two phases. It was found that the maximum temperature difference in the domain is below 1 K [24]. This is considered negligible for the purpose of the current work, and hence the chosen model is the isothermal simplified Rayleigh–Plesset.

At the inlet, the boundary condition of total pressure is imposed and at the outlet the mass flow. There is pure liquid at the inlet, and vapor is let to be generated inside the domain if the pressure drops

below the saturation pressure. There is no frame change model at the rotor–stator interfaces since there is no rotating domain. Instead, all the disk walls are rotating. The walls of the front and back side plates are also rotating, which corresponds to the real assembly conditions. The values of the boundary conditions agree with the conditions of the experimental study from which the geometry was taken. In the experimental study, the inlet total pressure is constant at 446 kPa. In the present study, additional simulations are run with gradually reducing inlet total pressure. The full boundary conditions are shown in Table 2. For convergence evaluation, the residuals, the total pressure rise, the shaft torque, the hydraulic efficiency and the domain's minimum static pressure are monitored.

In the present study, the pump's head is defined by the following equation, as the difference of the total pressure at the outlet $P_{tot,out}$ and the inlet $P_{tot,in}$ of the pump divided by the product of liquid density ρ and gravity acceleration g

$$H = \frac{P_{tot,out} - P_{tot,in}}{\rho g} \quad (2)$$

The NPSH is defined as the difference of the inlet total pressure and the saturation pressure P_{sat} divided by the product of density and gravity acceleration

$$NPSH = \frac{P_{tot,in} - P_{sat}}{\rho g} \quad (3)$$

The total volume of vapor present in the rotor domain is calculated as the volume integral of the vapor volume fraction f_v as given below:

Table 2 Simulation setup values

Parameter	Symbol	Value	Unit
Inlet total pressure	$P_{tot,in}$	446, 200, 100, 90, 80, 70, 60	(kPa)
Temperature	T	325	(K)
Rotational speed	ω	6000	(rpm)
Mass flow	$\dot{m}$	5	(kg/s)
Saturation pressure	P_{sat}	13,531	(Pa)

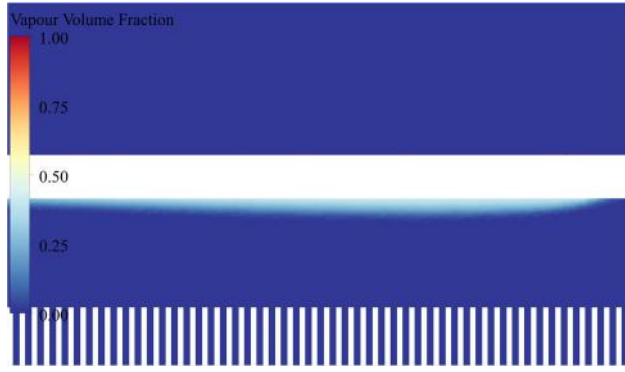


Fig. 3 Detailed view of the vapor volume fraction contour at the volute tongue for $P_{\text{tot,in}} = 200$ kPa

$$V_{\text{vap}} = \iiint f_v dV \quad (4)$$

The percentage of rotor's volume occupied by vapor is calculated as below, where V_{rot} is the total rotor's volume, the 170 interdisk gaps:

$$V_{\text{vap}\%} = \frac{V_{\text{vap}}}{V_{\text{rot}}} \times 100 \quad (5)$$

The mass of vapor present in the fluid domain is calculated as the volume integral of the product of vapor density ρ_{vap} and vapor volume fraction as given below:

$$m_{\text{vap}} = \iiint \rho_{\text{vap}} f_v dV \quad (6)$$

The total mass of fluid present inside the rotor is calculated as the sum of vapor and liquid mass as given below, where f_l is the liquid volume fraction:

$$m_{\text{rot}} = \iiint \rho_{\text{vap}} f_v + \rho f_l dV \quad (7)$$

Hence, the vapor mass percentage inside the rotor is

$$m_{\text{vap}\%} = \frac{m_{\text{vap}}}{m_{\text{rot}}} \times 100 \quad (8)$$

3 Results and Discussion

3.1 Vapor Formation and Propagation. In this section, the CFD results are postprocessed and examined. The vapor formation is visualized for the decreasing inlet pressure. For the initial case of inlet pressure of 446 kPa, there is no vapor present in the rotor. However, some vapor is visible at the volute tongue. This vapor occupies only 0.000116% of the volute volume. The experimental study from which the geometry is taken, parametrically examined different rotor configurations with the same volute. Therefore, the vapor at the volute tongue could be attributed to the fact that the volute might not be optimized for the rotor flow. It is also noted that no sufficient information was given in literature about the volute cross-sectional areas and its tongue fillet radius. It can be concluded that this vapor generation at the tongue is not related to the Tesla pump working principles and specific cavitation mechanisms but to the unmatched volute. For a decreased inlet total pressure of 200 kPa, there is still no vapor formation inside the rotor. There is vapor at the volute tongue which occupies 0.00066% of the total volute volume (Fig. 3).

Vapor generation around the volute tongue will be omitted in the following discussion. The vapor volume percentage will be calculated based on the rotor's volume only, as this is considered more relevant for examining the vapor propagation mechanisms.

Figure 4 illustrates the vapor volume fraction in the meridional plane and the vapor 3D shape with volume renderings. The cases shown are the ones where vapor is present inside the rotor.

Decrease of the total inlet pressure to 100 kPa leads to the first vapor formation at the rotor entrance, in the first channels closer to the machine inlet as seen in Figs. 4(a) and 4(b). The rotor vapor volume percentage is 0.012%. The 3D shape of the space that is occupied by vapor is not axisymmetrical and follows the direction of the rotation.

In Figs. 4(c) and 4(d) for inlet total pressure of 90 kPa, the vapor propagates to more channels toward the back. The shape of the vapor has also expanded in the azimuthal direction following the tangential acceleration of the flow by the viscous forces. The rotor vapor volume percentage is 0.034%.

Further decrease of the inlet total pressure to 80 kPa in (e) and (f), 70 kPa in (g) and (h), and 60 kPa in (i) and (j) shows the consistency of the cavitation mechanism. Vapor develops in more channels toward the back and expands in the azimuthal coordinate following

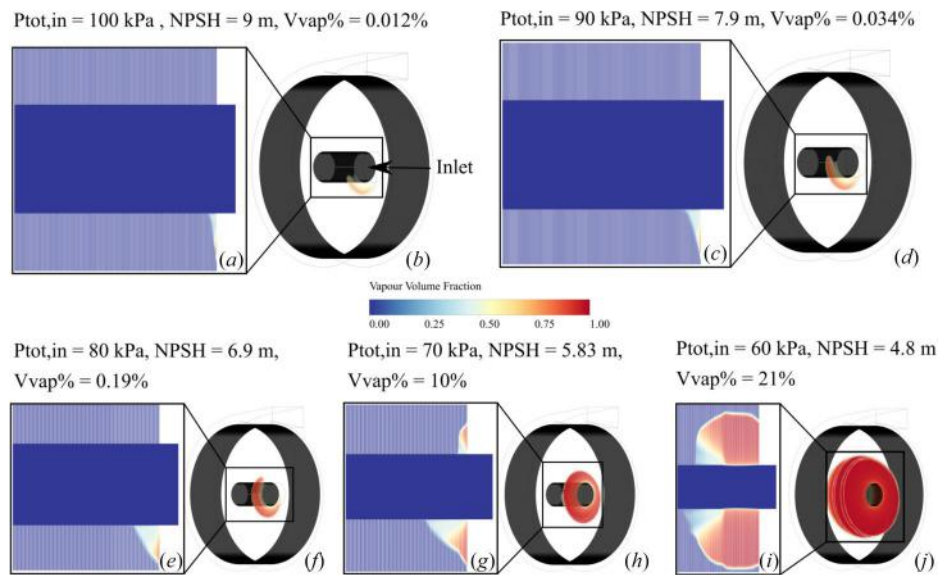


Fig. 4 Vapor volume fraction meridional contours and volume renderings for decreasing NPSH

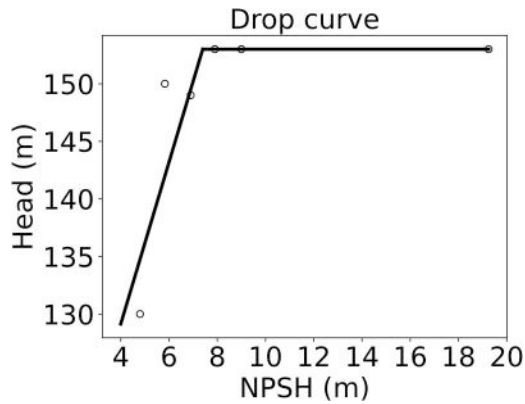


Fig. 5 Tesla pump head drop curve

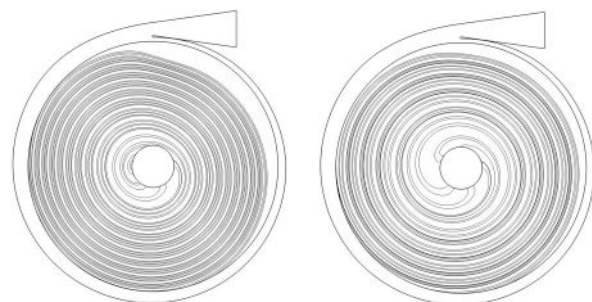
the direction of rotation. For inlet pressure of 80 kPa, the rotor vapor volume percentage is 0.19%. In the case of 70 kPa, vapor forms a full ring and completely blocks the first channels, leading to an increased rotor vapor percentage of 10%. For the inlet pressure of 60 kPa, vapor occupies most of the channels and blocks almost half of them closer to the pumps entrance. In that case, 21% of the rotor volume is occupied by vapor. In all cases, vapor changes its phase back into liquid after a certain radius due to pressurization. This radius is increasing with decreasing inlet pressure.

The NPSH head drop curve of the Tesla pump is indicated in Fig. 5.

3.2 Examination of the Fully Liquid Flow Field. To justify the cavitation mechanisms, the case of 446 kPa inlet pressure is further examined. This case is chosen because there is no vapor generated inside the rotor and the flow field is not affected by it. At a fully liquid field, the tendency of the system to form vapor can be assessed. Since the pump rotor consists of 170 channels in total, for indexing purposes, the channels are numbered sequentially from 1 close to the axial pump inlet to 170 next to the backwall.

As the first vapor forms and propagates for inlet pressure of 100 kPa and below, it can be seen by the volume renderings that its shape is not axisymmetric. The analysis of the velocity field of the pure liquid case of 446 kPa explains this behavior. In Fig. 6(a), the streamlines show that the volute affects the rotor flow. The part close to the volute tongue particularly pushes the flow back to the rotor causing a velocity field asymmetry and therefore vapor asymmetry when the inlet pressure drops. This happens because the volute is unmatched with the rotor flow as the geometry was taken by an explorative parametric experimental study and is not optimized. This is expected since it is known for centrifugal pumps as well that off-design conditions cause asymmetrical cavitation behavior [25].

In Fig. 7(a), the static pressure at the inlet of every channel is plotted against the channel index. It is illustrated that the pressure increases as we move through the channels from the inlet and further



(a) 1st channel, lowest mass flow (b) 152nd channel, highest mass flow

Fig. 6 Streamlines in the middle of the (a) 1st channel and (b) 152nd channel

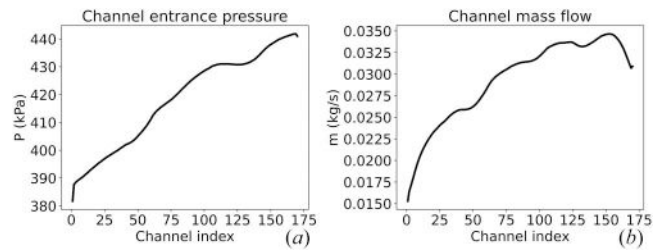


Fig. 7 (a) Distribution of channel entrance pressure and (b) distribution of mass flow passing through each channel for $P_{tot,in} = 446$ kPa

into the machine. This explains the tendency of the vapor to first appear at the channels closer to the entrance and then sequentially propagate toward the back. To further examine this observation, the mass flow that passes through each channel is plotted in Fig. 7(b). The entrance pressure and mass flow distributions exhibit significant nonuniformity, which contrasts with the assumptions commonly made in literature. The difference between minimum and maximum values reaches up to 16% for channel entrance pressures and 127% for channel mass flows. The general trend is that the mass flow increases from the 1st channel until the 152nd channel, then starts dropping until the 169th channel, and shows a small increase at the last channel. This behavior can be examined together with the flow field at the inner domain. The streamlines at the inner domain are shown in Fig. 8. The flow enters the machine axially and needs to turn almost 90 deg to enter the first channels; therefore, most of it moves further inside the machine and enters channels with higher index so that the turning of the flow into the radial direction is smoother. Hence, we have an increasing mass flow up to the 152nd channel where the highest mass flow is observed. After the 152nd channel, the effect of the backwall is prominent. The streamlines at the area inside the left highlighted rectangle justify why the 152nd channel has the highest mass flow since the flow hits the back wall and turns backwards. For the same reason, the mass flow starts reducing after that channel. There is a small peak at the last channel only. This is the minor effect of the back wall that forces the remaining amount of flow to turn 90 deg and enter the last channel. A detail view of Fig. 8 shows the velocity vector at the entrance of the last channels. The entrance velocity is not radial. Instead, it enters the channels at an angle. This angle approaches 90 deg the more we move to channels toward the back because of the backwall effect. This angled entrance causes nonuniform mass distribution and recirculation at the right corner of the entrance of each channel. The pressure is lower at the recirculation areas, and for this reason, when the inlet pressure further drops, the right half of each channel exhibits higher vapor volume fractions than the left half.

The observations about the effect of the backwall could initiate future research about the shape of the back plate which could be particularly designed to better accommodate the flow and turn it toward the entrance of the channels. A possible future back plate design could have a conical feature extending toward the inner domain instead of being flat.

Another important observation is that the mass flow does not consistently increase with the channel index. There are some mild drops observed around channels no. 50, 100, and 130. This happens because of flow recirculation. The right highlighted rectangle in Fig. 8 shows one of these areas where most of the flow recirculates instead of entering the channels. Redesign of the inner domain could resolve this issue and ensure a more uniform mass distribution.

The differential mass flow that enters each channel causes differences in the velocity field. In Fig. 6(b), the streamlines in the 152nd channel are illustrated. This is the channel from which most mass flow passes through. An interesting comparison can be made with the streamlines of the 1st channel in Fig. 6(a), the one with the lowest flowrate. The mass flow that passes through 152nd channel is +127% higher than the 1st channels flow. A 127% increase of mass flow from the 1st to the 152nd channel causes a 22% increase in

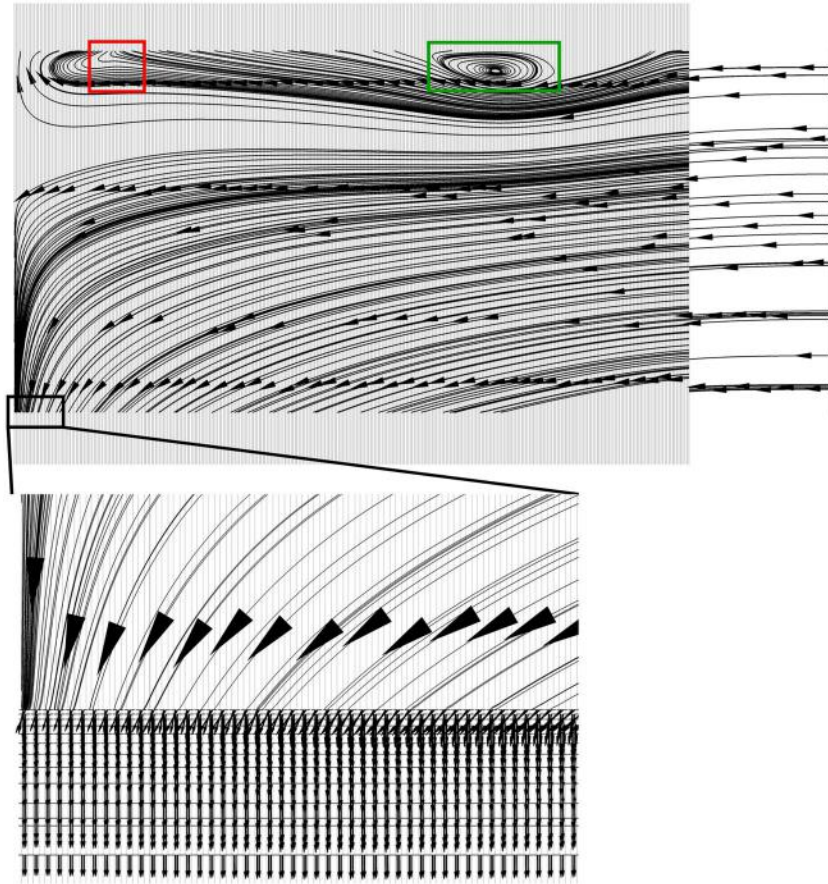


Fig. 8 Streamlines inside the inner domain, at the meridional plane, starting from pump's inlet

entrance pressure between the same channels. In the case of lower mass flow, the trajectory starts turning in lower rotor radius, while for increased mass flow the flow enters radially and remains radial for higher radius compared to the first channel. The initial radial velocity of the 152nd channel is higher because of the higher mass flow, and the viscous forces turn it less. Thus, there is less coiling of the streamlines and lower velocities at the outlet of the disks. However, in the 1st channel, the streamlines appear denser toward the rotor outlet where the velocities are higher. Apart from that, in the streamlines of the 152nd channel, there are some areas where the entrance velocity is not completely radial since it is affected by the flow recirculation inside the inner domain.

Despite the possible intuitive assumption that lower mass flow in the first channels would lead to lower entrance speed and higher entrance pressures, the pressure is lower, and vapor starts forming in these channels. Hence, further investigation of the pressure and velocity fields in different channels is needed. The channels that are chosen to be analyzed are the 1st and the 10th as representatives of the entrance area where the first vapor forms. The 85th channel is also chosen as the middle channel out of the 170 in the pump, and the 152nd channel as the one with the highest mass flow and as representative of the phenomena occurring toward the back of the machine. Samples are taken radially in the middle of the gap inside the 1st, 10th, 85th, and 152nd channel from the rotor entrance and halfway through the rotor as seen in Fig. 9. In that way, the pressure and velocity profiles as a function of the rotor radius can be acquired. The lines extend toward the negative direction of Y -axis because this is the area where the vapor starts forming with further inlet pressure decrease.

For further insight, the velocity and its radial and tangential components in the middle of the interdisk gap are also plotted in Fig. 10. In the velocity chart, there is a noticeable change in the

behavior of the curves at the point with approximately 20 mm radius which is approximately 12% of the rotor's radial span. For radii below that, the velocity is higher for channels closer to the back wall, while for radii above that, this trend inverses and the velocities are

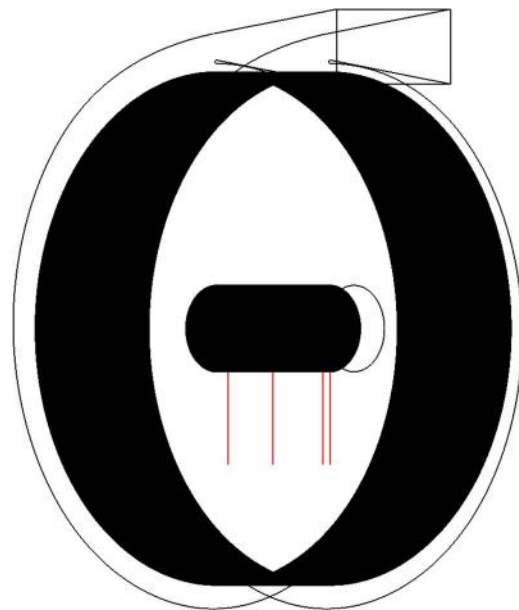


Fig. 9 Lines for sampling pressures and velocities in 1st, 10th, 85th, and 152nd channels

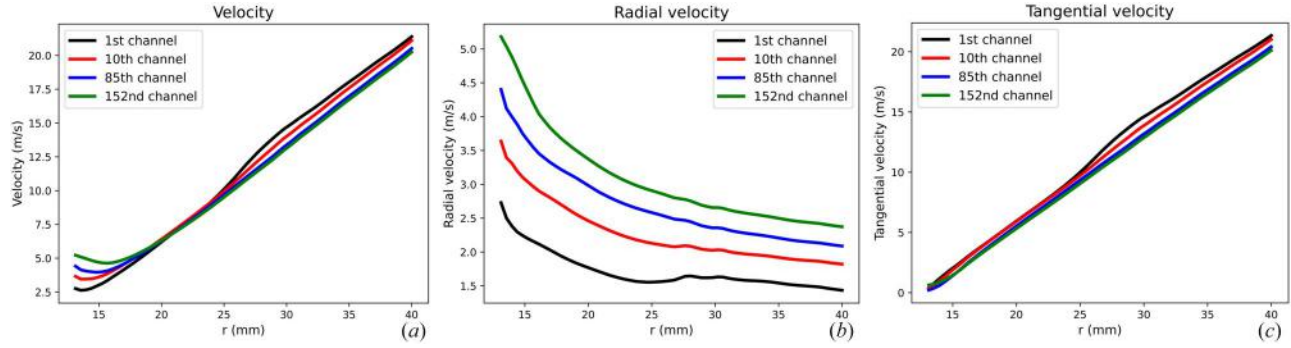


Fig. 10 (a) Velocity, (b) radial velocity, and (c) tangential velocity in the middle of the 1st, 10th, 85th, and 152nd channels against the radius

higher for the channels closer to the entrance. These trends can be justified by the examination of the radial and tangential velocities. In the radial velocity chart, the curves corresponding to gaps with higher mass flow, hence higher index number, indicate higher radial velocities. There is a continuous drop in the radial velocity values because of the increasing cross-sectional area that the flow meets as it moves to higher radii inside the rotor. Thus, since the reduction mechanism is the same for all, the radial velocity curves do not intersect each other. The trend of all the curves is reciprocal, as described by the Eq. (10), where m_{ch} is the mass flow passing through a channel, A_{ch} is the cross-sectional area of one channel, and $\overline{v_{r,ch}}$ is the mean radial velocity inside a channel.

$$m_{ch} = \rho A_{ch} \overline{v_{r,ch}} \quad (9)$$

$$\overline{v_{r,ch}} = \frac{m_{ch}}{2\pi\rho d} \frac{1}{r} \quad (10)$$

Plots illustrate the maximum radial velocity inside each channel, because the values are taken in the middle of the channel. However, the maximum and mean velocities follow the same trends because the velocity profiles are peaked at the center. This has been confirmed in literature experimentally with particle tracking velocimetry [6]. The same study suggests that a fourth degree polynomial sufficiently describes the shape of velocity profiles.

The tangential velocities on the graph show a linear increase with increasing radius. This is because the velocity of the rotating disk wall also increases linearly with the radius as described by Eq. (11). The middle of the gap develops tangential velocity because of the viscous forces. The value of this velocity is lower than the disk one but follows the same trend. The channels closer to the entrance indicate higher tangential velocities because less mass flow is accelerated more by the viscous forces

$$v_{tan,disk} = \omega r \quad (11)$$

The pressure profiles are plotted against the radius of the rotor using the sampling lines to provide a better understanding of the mechanisms of pressure increase within different rotor channels. In Fig. 11, the pressure is illustrated. The entrance pressure is higher the further inside we move into the machine. This observation agrees with the fact that vapor starts forming in the channels closer to the entrance. All pressure curves show lower pressure right at the channel entrance because of the angled flow entrance and thus the recirculation occurring at the left corner of each disk. This effect is less prominent the more the channel index increases because the flow enters at an angle closer to the radial direction. The pressure quickly increases and catches up with the rest of the curve because of the decrease of the radial velocity due to the rotor cross section area increase. After that, the pressure slightly reduces, and this trend appears in the curves as an almost flat part due to the small differences. For example, in the 1st channel, there is a 0.11% pressure decrease between radius 13.55 mm and 15.26 mm. This is

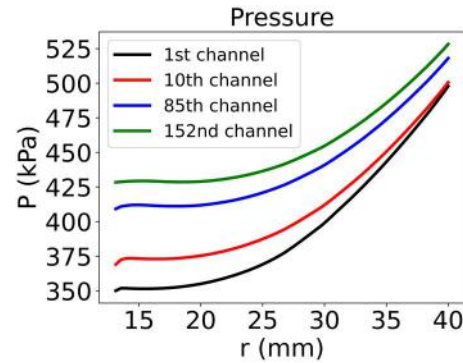


Fig. 11 Pressure profile against rotor radius at four different channels for $P_{tot,in} = 446$ kPa

attributed to the fact that at a certain radius, as can be noticed by the streamlines as well, flow starts turning by the viscous forces causing a suction effect at the radii below it. At radii above this area, the viscous forces take over, and the pressurization due to tangential acceleration and thus centrifugal forces become more evident. Therefore, the gradient of the curve changes and the pressure increases.

The area averaged shear stress–strain rate at the walls of each channel is plotted in Fig. 12 as an indicative of the viscous forces. Previous studies have numerically solved Navier–Stokes equations in axisymmetric cylindrical coordinates and have found that the velocity profile inside the gap and thus its gradient and the viscous force are affected by a parameter, dependent on the gap size, the kinematic viscosity, and the rotational speed [2]. This parameter is named BP from the researchers' initials and is shown in Eq. (12).

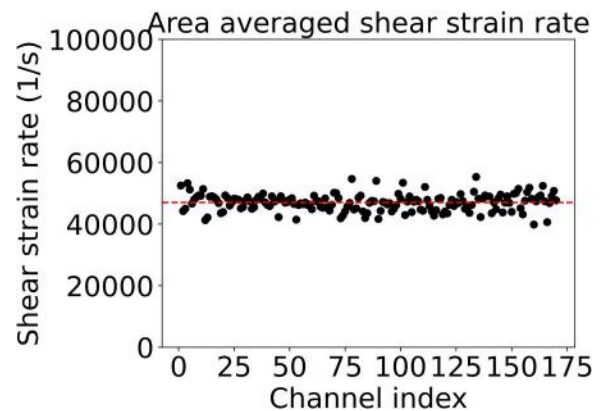


Fig. 12 Area averaged shear strain at the rotor disks surfaces of each channel

The values comprising this parameter are equal for each channel, so it is expected that the shear strain rate remains at a constant range. The fluctuations illustrated in Fig. 12 can be attributed to the recirculation that occurs inside the inner domain and lead to nonfully radial entrance velocities. This differs from the radial velocity assumption of the numerical solution that obtains the BP parameter. Consequently, since the viscous forces at the walls of each channel are independent of other flow phenomena and remain practically constant, the mass flow entering each channel emerges as the most important parameter for cavitation performance. The same values of viscous forces can accelerate lower mass flow to higher speeds

$$BP = \frac{d}{2} \sqrt{\frac{\omega}{\nu}} \quad (12)$$

As a conclusion, the behavior of the velocities inside the rotor can be attributed to the effects of radial and tangential velocities. Near the entrance of the rotor, the radial velocity component is more significant; hence, its decreasing trend is reflected in the total velocity as well. For higher radii, the flow is accelerated by the viscous forces, and the tangential component becomes more prominent, imposing an increase in the velocity. This results in a global minimum in each velocity curve. This global minimum is closer to the rotor entrance for the channels with less mass flow since in these the flow starts turning earlier. After that global minimum, there is an area susceptible to vapor formation where the velocity is high but there is not enough pressurization yet. This area is not the inlet of the channel but further inside the rotor where the tangential acceleration starts being significant. In the case of 100 kPa inlet pressure the first vapor starts forming, and the volume fraction is visibly higher at a radius of approximately 20 mm rather than at the inlet. The lobe shape of the vapor shows that it follows the rotational speed since it is mostly generated by the tangential acceleration. The above suggests that it is not only the radial entrance velocity to be examined, as has been done in literature, but all the components as the velocity increases inside the gap.

3.3 Examination of the Vapor Propagation. This section examines how the vapor propagates after its first formation. For this purpose, it assesses the cases of inlet total pressure of 100 kPa and below and when necessary, compares them with the initial fully liquid 446 kPa case.

Figure 13(a) shows the percentage of vapor occupying the rotors' volume plotted against the inlet total pressure. The volume curve represents how the vapor expands due to the drop of surrounding pressure according to real gas properties. This expansion blocks higher percentage of the rotor leading to a steeper head drop. For better connection with vapor generation mechanisms, the mass of vapor inside the rotor is plotted for different inlet pressures in Fig. 13(b). A similar nonlinear effect is observed as the gradient of the curve, hence vapor generation rate, increases with reducing inlet pressure.

As the inlet total pressure decreases, vapor propagates in more channels toward the back. The vapor contours shown in Fig. 4 imply that the vapor appears sequentially to adjacent channels like a "domino effect" because the flow still avoids the abrupt turning from

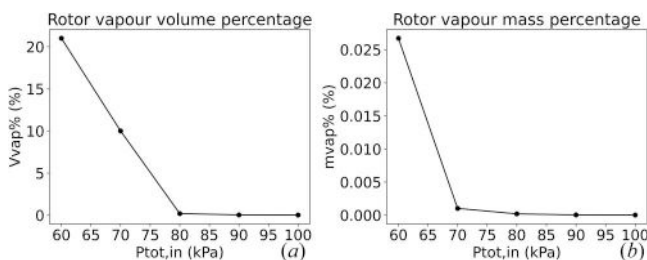


Fig. 13 (a) Rotor vapor volume percentage and (b) rotor vapor mass percentage for each inlet total pressure

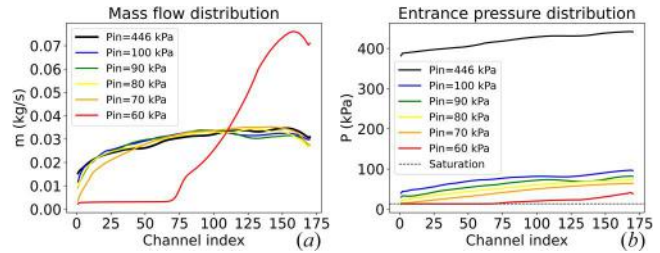


Fig. 14 (a) Mass flow distribution and (b) channel entrance pressure distribution for different inlet total pressures

the axial to the radial direction. As the first channels get blocked from vapor, the same mass flow should be distributed in fewer channels. The mass flow that passes through each channel is indicative of the entrance radial velocity. In Fig. 14(a), the mass flow distribution in the channels is plotted for the different inlet total pressures. Until the inlet total pressure of 70 kPa, the mass flows show small drops at the lower index channels because of the high vapor fractions. At the lowest total inlet pressure of 60 kPa, the differences are more significant because of the nonlinear vapor propagation. In that case, approximately 40% of the channels contain an average of 33% vapor volume which practically means channel blockage. This area is represented in the graph as a flat line of low mass flow for the first 70 channels. In the channels of higher index, the mass flow increases to compensate for the blocked region so that the pump delivers the same mass flow.

In Fig. 14(b), the pressure at the entrance of each channel is plotted for the different total inlet pressures. The trend of increasing pressure as we move to channels toward the back is still valid for all the curves regardless of the presence of vapor. For perspective, the black dashed horizontal line indicates the saturation pressure of water at 325 K. For the extreme case of 60 kPa total inlet pressure, approximately the first 70 channels have pressure below the saturation one. This is illustrated in the vapor volume fraction contour in Fig. 4(i) where the first 70 channels contain high fractions of vapor in their entrance. In the remaining channels, the higher fraction of vapor is formed further inside the rotor and not at the inlet because of the phenomenon of the tangential acceleration of the flow which was discussed in Sec. 3.2. As we move toward the channels at the back, the flow remains liquid up to higher radii and then changes phase. This is because the mass flow and hence the radial entrance velocity increase, and the flow starts accelerating by the viscous forces tangentially at higher radii. This is illustrated in Fig. 15 where the streamlines of the 160th channel are plotted. This channel is

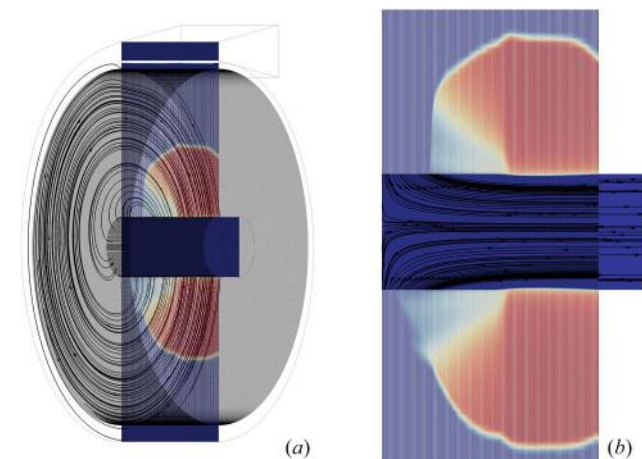


Fig. 15 (a) Streamlines in the middle of 160th channel and (b) streamlines in the meridional plane plotted along with the meridional contour of vapor volume fraction for $P_{tot,in} = 60$ kPa

chosen because it has no vapor to affect the liquid velocity field. The meridional vapor volume fraction is plotted alongside. The streamlines start turning and coiling at a radius where the vapor fraction in the adjacent channels toward the inlet of the machine is high. It can be concluded that, interestingly, the vapor appears first at higher radii inside the interdisk gap and then propagates toward the gap entrance. This is further confirmed by the observation of the vapor volume fractions of all the cases. Higher values always appear at higher radii rather than the inlet.

It becomes evident that it is not the channel entrance velocity that matters the most, but the tangential velocity and how this accelerates inside the rotor depending on the mass flow of each channel. When some of the first channels are blocked, all the flow needs to pass through lesser number of channels. However, as it has been discussed in Sec. 3.2., the viscous forces remain practically constant for each channel because they depend on the gap size, rotational speed, and fluid viscosity. This means that less channels can offer less viscous work to the same amount of mass and therefore cannot raise the pressure of the fluid as much. This is the mechanism that causes the head to drop as the vapor propagates to more channels.

3.4 Entropy Generation Rate Investigation. In this section, an analysis of the entropy generation rate is conducted. The highest entropy generation areas will be compared with the areas where vapor formation is expected. This can provide a connection of loss mechanisms and cavitation mechanisms.

The entropy generation rate represents the losses in a flow and can be decomposed in four components, according to the mechanism that produces them [26]. The first two components are the direct entropy generation rate $\dot{S}_D$ due to the velocity gradients and the indirect entropy generation rate $\dot{S}_I$ produced because of turbulence dissipation, which is the conversion of kinetic energy to thermal energy because of the viscous forces in a turbulent flow. The other two components are the heat conduction entropy generation rate due to mean temperature gradients $\dot{S}_{\overline{\partial T}}$ and the heat conduction entropy generation rate due to fluctuating temperature gradients $\dot{S}_{\partial T'}$.

The strain rate tensor is expressed in Eq. (13), where u , v , and w are the velocity components in axes x , y , and z , respectively. Axis z is the rotation axis of the machine

$$|\hat{S}| = \left[2 \left(\left(\frac{\partial u}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial y} \right)^2 + \left(\frac{\partial w}{\partial z} \right)^2 \right) + \left(\frac{\partial u}{\partial y} + \frac{\partial v}{\partial x} \right)^2 + \left(\frac{\partial u}{\partial z} + \frac{\partial w}{\partial x} \right)^2 + \left(\frac{\partial v}{\partial z} + \frac{\partial w}{\partial y} \right)^2 \right]^{\frac{1}{2}} \quad (13)$$

The direct entropy generation rate is calculated by the strain rate tensor as expressed in Eq. (14), where μ is the dynamic viscosity and T is the temperature

$$\dot{S}_D = \frac{\mu}{T} |\hat{S}|^2 \quad (14)$$

The indirect entropy generation rate is calculated by Eq. (15), where ρ is the density and ε is the turbulence dissipation rate

$$\dot{S}_I = \frac{\rho \varepsilon}{T} \quad (15)$$

The heat conduction entropy generation rates are calculated by Eqs. (16) and (17), where λ is the molecular thermal conductivity and λ_t is the turbulent thermal conductivity

$$\dot{S}_{\overline{\partial T}} = \frac{\lambda}{T^2} \left[\left(\frac{\partial T}{\partial x} \right)^2 + \left(\frac{\partial T}{\partial y} \right)^2 + \left(\frac{\partial T}{\partial z} \right)^2 \right] \quad (16)$$

$$\dot{S}_{\partial T'} = \frac{\lambda_t}{T^2} \left[\left(\frac{\partial T}{\partial x} \right)^2 + \left(\frac{\partial T}{\partial y} \right)^2 + \left(\frac{\partial T}{\partial z} \right)^2 \right] \quad (17)$$

The total entropy generation rate is calculated by all its components as described in Eq. (18). However, in this study, since the simulations are isothermal, there is no turbulence dissipation effect and temperature gradients are equal to zero. Thus, the total entropy generation rate for an isothermal case is simplified into Eq. (19), which will be used in the following analysis:

$$\dot{S} = \frac{T}{\rho} \left(\dot{S}_D + \dot{S}_I + \dot{S}_{\overline{\partial T}} + \dot{S}_{\partial T'} \right) \quad (18)$$

$$\dot{S}_{iso} = \frac{T}{\rho} \dot{S}_D \quad (19)$$

The contour of isothermal entropy generation rate in the middle of the pump (85th channel) for the fully liquid case of 446 kPa is illustrated in Fig. 16(a). For better visualization, the color-scale is logarithmic because of the wide range of the values. The volute is the region with the highest entropy rates. The area averaged entropy generation rate at the middle plane of the volute domain is $6.2 \text{ m}^2/\text{s}^3$, while at the rotor it is $2.5 \text{ m}^2/\text{s}^3$. This is attributed to the volute design which is not optimized for the specific operating point and causes losses. This effect concerns the volute design and is not inherently connected with the Tesla pump configuration, its working principles, and its cavitation mechanisms. High entropy areas are the rotor–volute interface, the volute tongue, and the inner–rotor interface. At the entrance of the rotor, there is a ring of high entropy generation with nonaxisymmetric shape that expands more toward the opposite direction of the volute tongue because of the unmatched volute geometry. Additionally, there is a high losses area close to the outlet of the rotor affected by the volute. This area can also be noticed in the streamlines of Fig. 6. In Fig. 16(b), the same contour is illustrated for the 1st channel, the closest to the pump's entrance. The behavior is similar, but the values of entropy generation rate are higher and the volute effect inside the rotor's flow is more prominent because of the lower mass flow passing through this gap. The area close to the rotor's inlet where losses are noticed coincides with the area where vapor is expected to occur with inlet pressure decrease.

The isothermal entropy generation rate eliminates terms that produce heat because of fluid viscosity, turbulence, and temperature gradients. However, the conclusions regarding the areas with the highest losses align with the findings of previous studies on entropy generation in the same geometry which included heat transfer effects [16]. Therefore, the isothermal entropy generation is sufficient to draw qualitative conclusions and make connections with cavitation mechanisms. Moreover, cavitation is mostly related to velocity gradients which comprise the strain tensor from which the isothermal entropy generation rate is calculated. The velocity field gradients include acceleration due to flow area decrease or sharp edges and shearing between two neighboring flows [21].

In Fig. 17(a), the area averaged entropy generation rate at the entrance of every channel is plotted against the channels index to show its distribution. After the 15th channel approximately, there is continuous drop in the entropy generation rate. This is because the turning of the flow from the axial to the radial direction and into the channels entrance is smoother and less losses are produced. However, in the first 15 channels, there is also the effect of dump

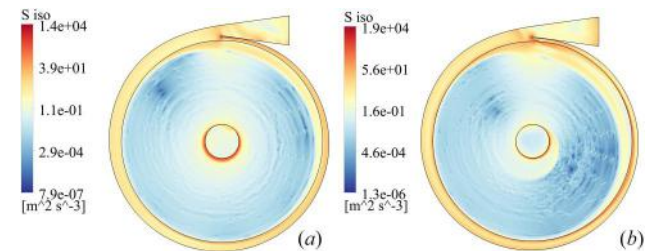


Fig. 16 Isothermal entropy generation contour for $P_{tot,in} = 446 \text{ kPa}$ in the (a) 85th channel and (b) 1st channel

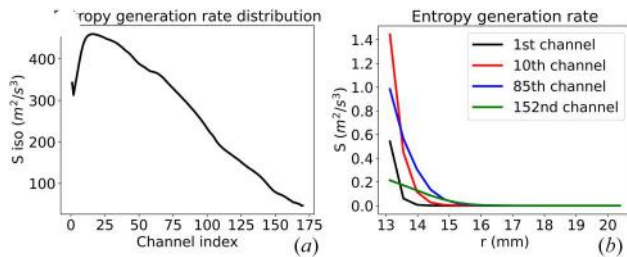


Fig. 17 Area averaged entropy generation rate: (a) at the entrance of every channel and (b) against rotor radius

losses at the entrance which are higher for higher mass flows leading to an increasing entropy generation rate until the 15th channel. By the mass flow distribution in the channels, the gradient at the first 15 channels is steeper, so this effect is more prominent (Fig. 7(b)).

For the examination of the entropy generation rate inside the rotor, certain sampling lines are used. These lines are the same as in Sec. 3.2 and illustrated in Fig. 9 but have reduced range for better visualization of the steep changes close to the inlet. The lines are in the middle of the 1st, 10th, 85th, and 152nd channels. They start from the rotor's entrance and expand to a radius of 20 mm. The 1st and 10th channels are chosen as representatives of the entrance section of the machine, the 85th is in the middle, and the 152nd is the one with the highest mass flow. The direction of the lines is toward the areas where vapor appears with further inlet total pressure drop. The results of the entropy generation sampling are shown in Fig. 17(b). The entropy rates are lower as we move from the 10th channel and toward the back which has been indicated in Fig. 17(a) as well. Higher values are observed close to the entrance of the channels because of the dump losses and the abrupt acceleration of the flow. The entropy generation rate drops very quickly as the flow moves inside the rotor. The steepness of the first dropping section of the curves is lower as we move to channels toward the back with higher mass flows. This means that more losses are caused in the region where the flow enters the rotor and remains almost radial for a certain radius span. The effectiveness of the shear forces increases when the flow starts turning and follows the rotation of the disks.

Excluding the volute induced losses which are not related to the working principles of such a pump, most losses in the rotor occur in the first channels that are also more susceptible to vapor formation. The above implies that a design with better mass flow distribution in the channels will benefit both the efficiency of the machine and its cavitation performance. The improvement of efficiency by avoiding the flow turning directly from the axial to the radial direction at the entrance of the rotor has been introduced by previous researchers [9]. They suggested a design where the disks are not perpendicular to the axis of rotation but have an angle to accommodate easier entrance of the flow. However, they had not considered that this arrangement would also have better cavitation behavior along with its performance, which is something observed in the current study.

4 Conclusion

The current study analyses the cavitation mechanism of a Tesla pump with a 3D CFD full pump approach. The main conclusions are the following:

- (1) The pressure and mass flow distributions among the different channels are not uniform as assumed in previous literature. Instead, there is a difference between minimum and maximum values that reaches 16% for channel entrance pressures and 127% for channel mass flows. This happens because the flow avoids turning 90 deg from the axial to the radial direction. Recirculation of the flow inside the inner domain can also cause small nonuniformity in the channels mass flow distribution.
- (2) Vapor starts forming in the channels closer to the pump's inlet because these channels indicate lower mass flows which can

be accelerated to higher velocities by the shear forces. With further inlet total pressure decrease, vapor forms sequentially to adjacent channels toward the back like a domino effect. This is because the flow still avoids entering the channels closer to the entrance and follows a path in which less turning is required.

- (3) Higher vapor volume fractions are observed further inside the rotor gap where the tangential velocity component becomes more prominent but there is not enough pressurization yet. The radius where this happens is dependent on the mass flow of the channel. The higher the mass flow, the higher the radius.
- (4) If several channels are blocked, fewer channels need to pump the full mass flow, and this leads to a detrimental head decrease since the viscous work that can be provided by each channel is limited. Because of this effect, the head of a Tesla pump seems to drop steeply.
- (5) By the losses' investigation study conducted, it is revealed that high entropy generation areas inside the rotor coincide with vapor formation areas. The above suggests that a design improved for cavitation would exhibit increased efficiency as well.
- (6) The Tesla smooth pumping advantage is still valid as seen in the pressure rise profile inside the rotor. Uniform mass flow distribution among the channels would improve Tesla pump cavitation performance.

Acknowledgment

A. Bakogianni acknowledges with gratitude the scholarship awarded to her by the Foundation for Education and European Culture (Nicos and Lydia Tricha).

Funding Data

- Cranfield University (Funder ID: 10.13039/501100000859).
- Rolls-Royce plc (Funder ID: 10.13039/501100000767).

Data Availability Statement

The datasets generated and supporting the findings of this article are obtainable from the corresponding author upon reasonable request.

References

- [1] Tesla, N., 1913, "Fluid Propulsion," Patent No. US1061142A, filed Oct. 21, 1909, and issued May 6, 1913.
- [2] Breiter, M. C., and Pohlhausen, K., 1962, "Laminar Flow Between Two Parallel Rotating Disks," Aeronautical Research Laboratory, Wright-Patterson Air Force Base, OH, Report No. AD 275562.
- [3] Boyack, B. E., and Rice, W., 1971, "Integral Method for Flow Between Corotating Disks," *ASME J. Basic Eng.*, **93**(3), pp. 350–354.
- [4] Anselmi, E. P., Fiaschi, D., Manfredi, G., Nicotra, G., and Talluri, L., 2021, "Design and Optimization of a Tesla Pump for ORC Applications," *Proceedings of Sixth International Seminar on ORC Power Systems*, Munich, Germany, Oct. 11–13, pp. 1–10.
- [5] Sengupta, S., and Guha, A., 2012, "A Theory of Tesla Disc Turbines," *Proc. Inst. Mech. Eng., Part A: J. Power Energy*, **226**(5), pp. 650–663.
- [6] Schosser, C., 2016, "Experimental and Numerical Investigations and Optimisation of Tesla-Radial Turbines," Ph.D. dissertation, University of Bundeswehr, Munich, Germany.
- [7] Song, J., Ren, X., Li, X., Gu, C., and Zhang, M., 2018, "One-Dimensional Model Analysis and Performance Assessment of Tesla Turbine," *Appl. Therm. Eng.*, **134**, pp. 546–554.
- [8] Freschi, R., Bakogianni, A., Rajendran, D. J., Palma, E. A., Talluri, L., and Roumeliotis, I., 2023, "Flow Field Explorations in a Boundary Layer Pump Rotor for Improving 1D Design Codes," *Designs (Basel)*, **7**(1), p. 29.
- [9] Hasinger, S. H., and Kehrt, L. G., 1963, "Investigation of a Shear-Force Pump," *ASME J. Eng. Power*, **85**(3), pp. 201–206.
- [10] Morris, J. H., 1980, "Performance of Multiple-Disk-Rotor Pumps With Varied Interdisk Spacings," David W. Taylor Naval Ship Research and Development Center, Bethesda, MD, Report No. AD-A088010.
- [11] Wang, B., Okamoto, K., Yamaguchi, K., and Teramoto, S., 2014, "Loss Mechanisms in Shear-Force Pump With Multiple Corotating Disks," *ASME J. Fluids Eng.*, **136**(8), p. 081101.
- [12] Dodsworth, L. J., 2016, "Operational Parametric Study of a Prototype Tesla Pump," MASC thesis, Dalhousie University, Halifax, NS, Canada.

- [13] Southwestern Controls, 1954, "Southwestern Controls Disk Pumps," Houston, TX, accessed Apr. 1, 2023, <https://swcontrols-diskpumps.com/>
- [14] Medvitz, R., Boger, D., Izrael, V., Rosenberg, G., and Paterson, E., 2011, "Computational Fluid Dynamics Design and Analysis of a Passively Suspended Tesla Pump Left Ventricular Assist Device," *Artif. Organs*, **35**(5), pp. 522–533.
- [15] Balje, O. E., 1965, "A Study of the Performance Potential of Conventional and Shear Force Pumps," Aerospace Research Laboratories, Wright-Patterson Air Force Base, OH, Report No. 469130.
- [16] Palaveev, K., Rajendran, D. J., Anselmi, E., Santhanakrishnan, M., and Pachidis, V., 2023, "Insights Into the Flow Field and Performance of a Boundary Layer Pump," *ASME Paper No. GT2023-102186*.
- [17] Lemmon, E. W., Bell, I. H., Huber, M. L., and McLinden, M. O., 2018, "NIST Standard Reference Database 23: Reference Fluid Thermodynamic and Transport Properties-REFPROP," Gaithersburg, MD.
- [18] Salim, S. M., and Cheah, S. C., 2009, "Wall Y+ Strategy for Dealing With Wall-Bounded Turbulent Flows," International Multi-Conference of Engineers and Computer Scientists (IMECS), Hong Kong, Mar. 18–20, pp. 1–6.
- [19] Rayleigh, L., 1917, "VIII. On the Pressure Developed in a Liquid During the Collapse of a Spherical Cavity," *London, Edinburgh, Dublin Philos. Mag. J. Sci.*, **34**(200), pp. 94–98.
- [20] Plesset, M. S., 1949, "The Dynamics of Cavitation Bubbles," *ASME J. Appl. Mech.*, **16**(3), pp. 277–282.
- [21] Orlandi, F., Montorsi, L., and Milani, M., 2023, "Cavitation Analysis Through CFD in Industrial Pumps: A Review," *Int. J. Thermofluids*, **20**, p. 100506.
- [22] Bakir, F., Rey, R., Gerber, A., Belamri, T., and Hutchinson, B., 2004, "Numerical and Experimental Investigations of the Cavitating Behavior of an Inducer," *Int. J. Rotating Mach.*, **10**(1), pp. 15–25.
- [23] Ye, Y., Zhu, X., Lai, F., and Li, G., 2017, "Application of the Semi-Analytical Cavitation Model to Flows in a Centrifugal Pump," *Int. Commun. Heat Mass Transfer*, **86**, pp. 92–100.
- [24] Shi, G., Wei, Y., and Liu, S., 2022, "Cavitation Flow Characteristics of Water and Liquid Oxygen in the Inducer Considering Thermodynamic Effect," *Energies (Basel)*, **15**(14), p. 4943.
- [25] Lei, T., Shan, Z. B., Liang, C. S., Chuan, W. Y., and Bin, W. B., 2014, "Numerical Simulation of Unsteady Cavitation Flow in a Centrifugal Pump at Off-Design Conditions," *Proc. Inst. Mech. Eng. C: J. Mech. Eng. Sci.*, **228**(11), pp. 1994–2006.
- [26] Khait, A., Noskov, A., Alekhin, V., and Bianco, V., 2018, "Analysis of the Local Entropy Generation in a Double-Circuit Vortex Tube," *Appl. Therm. Eng.*, **130**, pp. 1391–1403.



DSPACE

<https://dspace.org/>

Insights into the cavitation behavior of a boundary layer pump

Bakogianni, Agapi; Rajendran, David John; Anselmi Palma, Eduardo; Pachidis, Vassilios; Palmer, Chloe Jo

2025-12

American Society of Mechanical Engineers (ASME)

<https://doi.org/10.1115/1.4069454>

Bakogianni A, Rajendran DJ, Palma EA, et al., (2025) Insights into the cavitation behavior of a boundary layer pump. Journal of Engineering for Gas Turbines and Power, Volume 147, Issue 12, December 2025, Article number 121012