

NUMERICAL STUDY ON THE INFLUENCE OF A VOLUTE NECK ON COMPRESSOR PERFORMANCE

Vlad Goldenberg
SoftInWay, Inc.
Burlington, MA

Ben Conser
SoftInWay, Inc.
Burlington, MA

ABSTRACT

In this work, the influence of the volute neck on compressor performance was analyzed for several different volutes, each with different neck sizes. Volute designs using several neck sizes were analyzed, with the extremes being a volute with no neck, as well as a volute with a large neck size where the volute cross section centroids have a constant radial position.

To perform this work, several single-stage compressor flowpaths were developed whose differences only involve the geometry of the vaneless space and volute downstream of the impeller. The design speedline was analyzed in CFD for each flowpath. The results of these CFD analyses were postprocessed to determine how the different volutes influence the compressor performance.

The outcome of these analyses is that there is a performance benefit to be gained from having a volute with some neck due to the slight increase in pressure recovery and reduction in pressure loss through the volute. However, increasing the neck size too much results in diminishing performance improvements and eventually performance reduction. This indicates that for a particular compressor design, there is a neck size which will optimize the volute performance, resulting in slightly higher efficiency and pressure rise capability across the operating range.

For the case analyzed, there is an optimum in both pressure recovery and total pressure loss as the volute neck extends 31% into the overall height of the volute cross section. With this volute neck, the peak compressor efficiency can be improved from 84.6% to 86.0%, while the overall pressure ratio increases by 1.9%.

Keywords: Volute; Volute Neck; Volute Lip

NOMENCLATURE

A1	volute inlet area
A2	volute outlet section area at the end of the diffusion pipe
Ath	volute throat area

CAD	computer aided design
CFD	computational fluid dynamics
CMD	controlled mean diameter volute type
Cp	pressure recovery coefficient
D2m	meanline diameter at the volute throat
HECC	high efficiency centrifugal compressor
hk	equivalent volute outlet height
k	turbulent kinetic energy in kw-SST turbulence model
LenDif	volute diffuser pipe length
$\hat{n}$	normal vector relative to cross section surface
Neck Size	volute neck size (absolute)
P-Type	volute type with constant inlet diameter and varying external diameter
P_s	static pressure
P_t	total pressure
ρ	density
RANS	Reynolds Averaged Navier Stokes
R_{inlet}	volute inlet radius
$R_{minimum}$	minimum volute radius
$\vec{r}$	radial position relative to cross section centroid
S	flow surface area
S_{neck}	relative volute neck size
$\vec{v}$	flow velocity
$\vec{v}_\theta$	tangential component of flow velocity
$v_{\theta,s}$	tangential component of flow velocity measured relative to flow surface area (S)
v_m	meridional component of flow velocity
$\frac{\vec{v}_m}{\omega_s}$	mass flow averaged meridional flow velocity
ω_s	angular flow velocity measured relative to flow surface area (S)
ω	specific dissipation rate in kw-SST turbulence model
ζ	total pressure loss coefficient

1. INTRODUCTION

Any turbomachine discharge that must be turned from annular duct flow into a pipe or other separate system requires a collection device like a volute or collector to turn annular flow to pipe or duct flow. In the case of a centrifugal compressor with a vaneless diffuser, where the flow has a high tangential velocity, a volute is the most effective device for converting the high speed tangential velocity from a radial annular flow into a downstream pipe or duct. A volute is, when well-designed, deliberately contoured on the internal surface to enable controlled diffusion and turning of the flow into the desired direction in such a way that relatively small portion of the flow velocity impacts the end walls and therefore dynamic pressure loss. Of course, the volute is designed for a particular operating condition, and the flow pattern at off-design conditions is likely to be suboptimal. Furthermore, it must be acknowledged that the volute has a highly concave internal surface and manufacturing assembly capabilities and associated costs play a large role in dictating the design from a practical standpoint.

The scientific and engineering literature on volute flow path fluid dynamic performance and how the design material shape impacts such performance is immense. A recent study involving genetic optimization, for example, is documented in [1]. The goal of the present work is highly focused on one very specific yet extremely important aspect of volute design and fluid mechanical performance. It applies to asymmetric type volutes specifically where the radial outflow turns and diffuses along a wall and turns helically toward one side forming a swirl pattern. There may exist a small protrusion of material that effectively extends the rotationally periodic and axisymmetric portion of the flow and forms the end-walls of a vaneless diffuser into the volute. It is shown schematically in FIGURE 1.

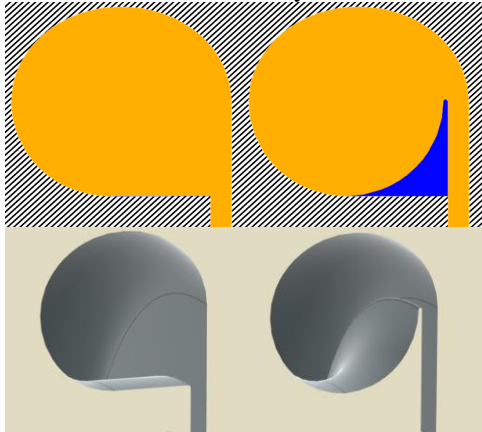


FIGURE 1: Schematic representations of what is referred to as the neck of a volute. Contrast of no neck (left) to a neck (right) with the neck materials shown in blue. 3D CAD representations are provided in the bottom images.

Another term that may be used is “lip” and it is possible other terms are in use. In this paper the term neck is used. As can be seen from FIGURE 1, the flow emerges from the radial space outward toward the volute often with a significant tangential velocity. It is in fact the radial velocity that becomes a swirling

pattern in the volute cross-section. The neck portion essentially is additional material that extends the end-wall of the diffuser and guides the fluid on both the vaneless diffuser side and volute side (with the opposite wall of the neck), also delaying the mixing of the incoming fluid with the helical swirl pattern already established in the volute. It should be noted that the neck is distinct from the tongue (or often called “cutwater” in pumps). The tongue defines the separation between the scroll portion and the downstream pipe, while the neck defines the separation between the radial annular passage at the inlet of the volute and the volute scroll. While the two are related and may be associated with the same material, they are NOT the same feature.

Common design wisdom is that at least some neck is beneficial for smooth and efficient flow. Clearly, the neck forms a concave surface and larger necks generally imply more complex manufacturing and/or assembly processes to ensure a sufficiently smooth internal surface, so there is a high interest in understanding and quantifying the extent to which the neck is beneficial. A review of publicly available designs reveals that practical machines span the entire continuum with respect to volute necks, as exemplified in FIGURE 2. One is able to see designs that have no neck at all (as on the left of the figure) and also designs with a very large neck (as on the right of the figure).

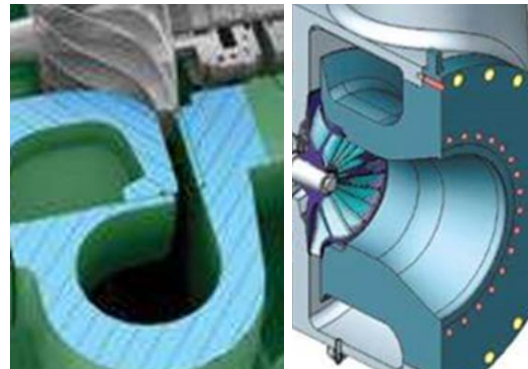


FIGURE 2: Two relevant and opposing examples found online of a volute with no neck (left, from [2]) and one with a large neck (right, from [3])

The question of the volute neck came into particular focus for these authors during a particular product development project that the authors were executing in the development of a novel centrifugal compressor for an aerospace application. As is common in new product development, during the prototyping, testing, and qualification phase, questions arose from the manufacturing team regarding the substantial neck that was incorporated into the design and whether it could be eliminated or its length reduced to reduce the complexity and ultimately the production cost. As is also common in industrial practice, these inquiries arose after all design reviews were successfully passed and the design had been approved for qualification, the test facility configuration plan was approved and on the facility schedule, and therefore very little time or resources available to re-evaluate the required design changes proposed. Nominally, it was necessary to evaluate potential design candidates. The

practical reality is that for the majority of organizations, as was the case for the authors, the volute reduced-order models available (for example that suggested by Aungier [4]) were not sensitive to nuances such as volute neck lengths, and the only recourse was relatively resource-intensive CFD calculation. Using several CFD calculations for a number of design alternatives and operating conditions, a satisfactory solution was found. Nevertheless, this particular experience inspired a more thorough and systematic investigation of the volute neck that is the subject of the present work.

By analyzing the performance of several compressor designs with different volutes, the specific influence of volute neck on performance can be determined. While the neck will influence overall machine performance parameters such as pressure ratio, efficiency, and power consumption, performance across the stationary downstream parts of the flowpath can be isolated to get a better understanding of the volute's performance specifically. To quantify the volute's performance, non-dimensional parameters such as pressure recovery coefficient and total pressure loss coefficient can be analyzed, with more efficient volutes typically having higher pressure recovery and lower total pressure loss. Additionally, parameters such as swirl number, tangential flow velocity, and meridional flow velocity can be analyzed at several locations along the volute flowpath. The trends of these parameters can then be compared between the different volute designs to provide better insight into the benefits of the different volute neck sizes.

1.1 Related Work

The authors are not aware of any other report or publication that specifically address research into the aerodynamic effects of a volute neck. Nevertheless, it is recognized that there is a plethora of previous research on volute design and aerodynamics and many of those results, especially concerning optimization and characterization, at least implicitly relate to the neck. Very early work for turbine exhaust volutes for marine power [5] experimentally examined basic integral geometry parameters and their relationship to aerodynamic performance. [6] Shows variations of volute geometries during genetic optimization that clearly implicitly have variable neck sizes, although the work is intended to demonstrate effective optimization rather than characterize a particular feature effect. Several different forms and integral parameters of volute geometry were investigated in [7] with respect to resulting performance metrics. Additional research on shape optimization of volutes was performed by [8], [9], and [10]. Zhang, et. al. [8] show comparisons of optimization methods that include clearly distinct volute shapes and necks, without particular focus on the effect of any feature. Hottois et. al. [9] use an adjoint-based optimization and employ a parameterization that can lend itself to quantifying the neck size, although no attempt seemed to be made to focus on that feature and its aerodynamic ramifications. In contrast, the parameterization used by Adjei, et. al. [10] was arbitrary, even though it appears there was implicit variability in what may be measured as a neck. Work by Sitkin [11] examined various volute forms in the context of optimization, although no particular distinguishing feature was highlighted.

The study by Melani, et. al. [12] did indeed systematically examine key design parameters and their influence on aerodynamic performance. However, in this study, focus was on integral parameters related to area distribution and tongue shape, rather than the neck. Similarly, [13] investigated aerodynamic performance and noise related to the particular shape of the volute tongue, rather than neck.

2. METHODS

2.1 1D Analytical Methods

To perform the analysis, first a baseline compressor model needed to be established. To do this, a compressor geometry and operating point were specified based on literature. The majority of the flowpath from NASA's High Efficiency Centrifugal Compressor (HECC) was used for this analysis [14] [15] [16]. The flowpath from literature consists of a single compressor stage with an inlet duct, impeller, short vaneless space, vaned diffuser, L-bend, and exit guide vanes. For our analysis, the vaned diffuser, L-bend, and exit guide vane portions of the flowpath were eliminated. Instead, the vaneless diffusion space was extended radially, with the flow exiting from the vaneless diffuser into a volute. All other upstream flowpath components remained the same. The upstream flowpath components were imported into AxSTREAM™ [17], a software for turbomachinery design and optimization. The vaneless diffuser and volute were then manually added to the downstream portion of the AxSTREAM™ flowpath. It should be noted that while AxSTREAM can be used to perform 1D Meanline and 2D Streamline analyses of compressor flowpaths, for this application AxSTREAM was only used as a flowpath geometry modification tool. The aerodynamic performance of these flowpaths was analyzed later on using a separate 3D CFD tool. An example of the meridional view of one of the analyzed flowpaths, as well as a complementary 3D-view of the same design, is provided in FIGURE 3.

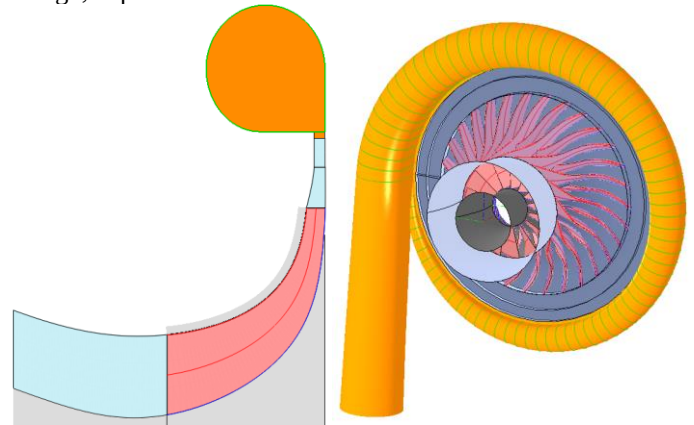


FIGURE 3: Example of analyzed compressor meridional flowpath

Several sets of volutes were designed for this flowpath. Note that the volutes were designed to have constant values for outlet flow area, throat area, and maximum radius. Due to the constant flow area and maximum radius constraints, the minimum volute radius ends up being nearly the same across all the volute

designs. The inlet radius to the volute was then adjusted to control the volute neck size. FIGURE 4 provides a diagram indicating where these volute radii are measured from.

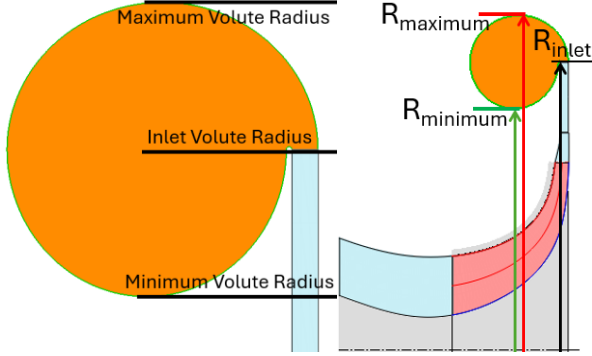


FIGURE 4: Definition of volute radii, measured from axis of rotation

The neck size describes how much the volute inlet radius (R_{inlet}) extends past the minimum volute radius ($R_{minimum}$). This is described in Equation (1). All the volutes analyzed in this study had a minimum volute radius of 274mm. For volute profiles where no part of the volute cross section extends radially below the volute inlet radius, those profiles are said to have zero neck.

$$Neck\ Size = R_{inlet} - R_{minimum} \quad (1)$$

To compare volutes of different absolute size, a relative neck size parameter (S_{neck}) can be used which compares the neck size to another physical dimension parameter of the volute, effectively non-dimensionalizing it. One option for this parameter is the volute equivalent outlet height (hk), which describes the width of the outlet area at the 360° location of the volute, as shown in FIGURE 5. Using this parameter, then S_{neck} could be described as in Equation (2).

$$S_{neck} = \frac{Neck\ Size}{hk} \quad (2)$$

The construction of the volute involved the specification of both the scroll portion of the volute, as well as the straight outlet diffusion pipe portion (often called the “conical diffuser”). For the scroll portion, the meanline diameter at the volute throat ($D2m$) was fixed to the same value for each volute, as shown in FIGURE 5. The volute outlet section area at the end of the diffusion pipe ($A2$) was specified as the same value for each volute. The ratio between $A2$ and the volute throat area (A_{th}) was also specified as the same value, resulting in the same throat area for each volute. The diffuser length ($LenDif$) was also the same for each volute. The volute areas and diffuser length are highlighted in FIGURE 6. The volute equivalent outlet height (hk) was also specified to ensure that the maximum volute radius at the 360° position was the same across all designs. Additionally, an elliptic form coefficient was varied between each volute to control the roundness of the volute profile, ensuring the volute profiles smoothly transitioned from one volute profile with a nearly flat hub wall extension to eventually a volute profile with a rounded hub wall extension. Lastly, the

same area distribution was specified which linearly increased the volute cross sectional area around its scroll portion.

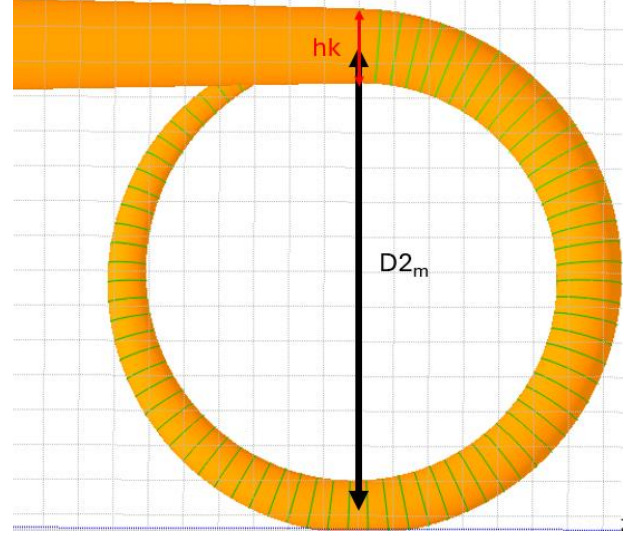


FIGURE 5: Volute equivalent outlet height and meanline outlet diameter

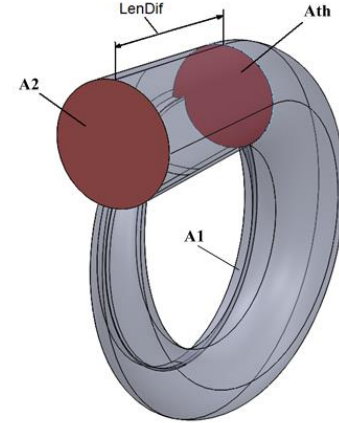


FIGURE 6: Volute flow areas and diffuser length

TABLE 1: Parameters of analyzed volutes

Volute Type	Inlet Volute Radius [mm]	Neck Size [mm]	S_{neck}	Elliptic Form Coefficient	Label from FIGURE 7
P-Type	274	0	0	n/a	a
CMD	274	0	0	0.5	b
CMD	286	11.2	0.105	0.6	c
CMD	297	22.5	0.210	0.7	d
CMD	308	33.7	0.315	0.8	e
CMD	319	44.9	0.420	0.9	f
CMD	331	56.2	0.525	1	g

Seven geometric variations of the diffuser and volute geometry which were synthesized according to the parameters described in the foregoing paragraph were consequently analyzed, where the radial length of the diffuser was adjusted depending on the positioning of the volute neck. As mentioned previously, all cases maintained the same maximum volute radius and outlet flow area. The extreme cases included both a volute with no neck, as well as a volute with a large neck size where the volute cross section centroids have a constant radial position. Six separate cases with varying volute neck positions were generated. The volute type for these volutes is the ‘controlled mean diameter’ (CMD) volute type. Additionally, a seventh no-neck P-type volute was analyzed, whose asymmetric cross sections have a varying external diameter and a constant inlet diameter. A comparison of the different volute forms can be found in TABLE 1 or by reviewing FIGURE 7. Note that henceforth in this paper, the different CMD-Type volutes will be referred to and differentiated by their neck size, rounded to the nearest millimeter, although an appropriate generalization would call for reference to S_{neck} .

A more detailed description of the volute geometries is provided in TABLE 7 in the Appendix. However, a detailed description of the upstream duct, impeller, and vaneless diffuser is not included in this paper, given that these details can already be found in the literature for NASA’s HECC [14].

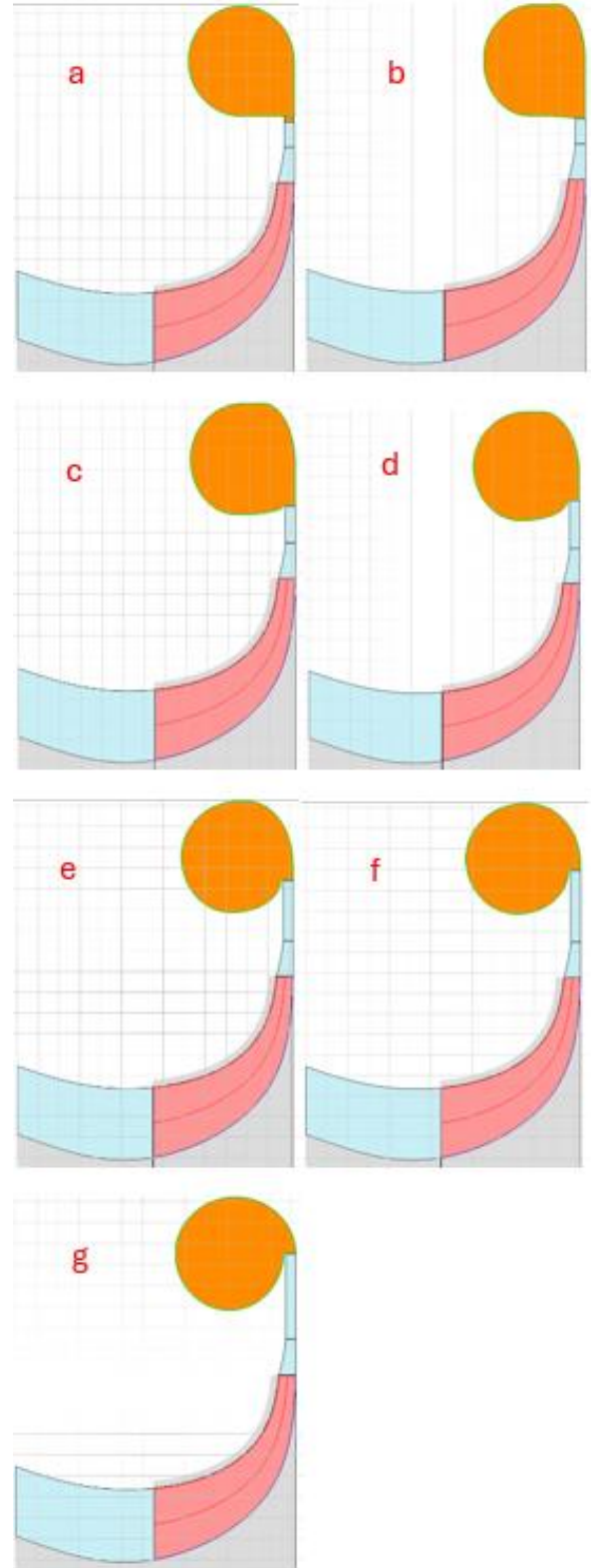


FIGURE 7: Volute form for a) P-Type volute, no neck b) CMD-Type volute, no neck c) CMD-Type volute, 11mm neck d) CMD-Type volute, 22mm neck e) CMD-Type volute, 34mm neck f) CMD-Type volute, 45mm neck g) CMD-Type volute, 56mm neck

2.2 3D CFD Analytical Methods

The CAD files of the fluid volume were exported from AxSTREAM™ for each volute design and subsequently analyzed in 3D CFD using STAR-CCM+ [18]. Two CAD parts were exported for each project: one rotationally periodic rotor part corresponding to a single blade passage domain (consisting of the inlet duct, impeller, and vaneless diffuser) and another part for the volute domain. Note that the vaneless diffuser was included in the rotor domain given that it is also rotationally periodic, so analyzing just a slice of the vaneless diffuser provides sufficient data to evaluate the diffuser's full flow field. An example of one of the analyzed fluid domains is shown in FIGURE 8.

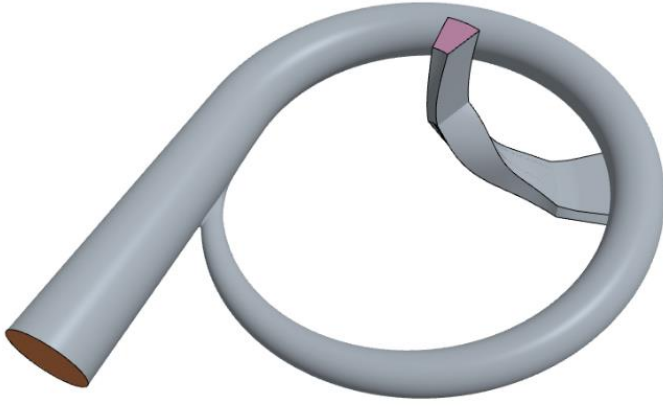


FIGURE 8: Example of CFD fluid domain analyzed in STAR-CCM+

The rotor part was assigned to a rotating region, while ensuring that only the impeller hub and blade surfaces were specified as rotating. The volute part was assigned to a stationary region. A mixing plane interface was used between the rotor region and the volute region, while a periodic interface transferred data between the blade passage periodic surfaces. Placing the mixing plane as far downstream of the impeller trailing edge enabled the physical mixing of the wake and jet structure of the impeller outlet prior to becoming numerically mixed, preserving maximum physical reality as possible within the confines of the mixing plane model, in addition to maximizing the grid resolution by taking advantage of the periodic nature of the flow in the diffuser.

Steady-state Reynolds Averaged Navier Stokes (RANS) computation was performed. This approach involves the solution of the fundamental physical equations for material mass, momentum, and energy conservation integrated over a finite control volume [19]. These conservation equations are provided in Equation (3) – Equation (5) and account for continuity (mass conservation), force-momentum, and energy conservation, respectively. A coupled flow solver was used which solved the conservation equations in a coupled manner (i.e. solved simultaneously as a vector of equations using a pseudo-time marching approach).

$$\frac{\partial}{\partial t} \iiint \rho dV + \oint \rho \mathbf{v} \cdot d\mathbf{a} = \iiint S_u dV \quad (3)$$

$$\frac{\partial}{\partial t} \iiint \rho \mathbf{v} dV + \oint \rho \mathbf{v} \otimes \mathbf{v} \cdot d\mathbf{a} = - \oint p \mathbf{l} \cdot d\mathbf{a} + \oint \mathbf{T} \cdot d\mathbf{a} + \iiint f_b dV + \iiint S_u dV \quad (4)$$

$$\frac{\partial}{\partial t} \iiint \rho E dV + \oint \rho H \mathbf{v} \cdot d\mathbf{a} = - \oint q \cdot d\mathbf{a} + \oint \mathbf{T} \cdot \mathbf{v} d\mathbf{a} + \iiint f_b \cdot \mathbf{v} dV + \iiint S_u dV \quad (5)$$

The kw-SST turbulence model was used in the CFD simulation. The transport equations for the kinetic energy k and specific dissipation rate ω are provided in Equation (6) and Equation (7), respectively [20]. An overview of the CFD simulation properties is provided in TABLE 2.

$$\frac{\partial}{\partial t} (\rho k) + \nabla \cdot (\rho k \mathbf{v}) = \nabla \cdot [(\mu + \sigma_k \mu_t) \nabla k] + P_k - \rho \beta * f_\beta * (\omega k - \omega_0 k_0) + S_k \quad (6)$$

$$\frac{\partial}{\partial t} (\rho \omega) + \nabla \cdot (\rho \omega \mathbf{v}) = \nabla \cdot [(\mu + \sigma_\omega \mu_t) \nabla \omega] + P_\omega - \rho \beta * f_\beta * (\omega^2 - \omega_0^2) + S_\omega \quad (7)$$

TABLE 2: Properties of CFD simulations

CFD Property Category	CFD Property
Mesh Type	Polyhedral
Turbulence Model	kw-SST
Equation of State	Ideal Gas
Solver	Coupled Flow Solver
Rotation Model Method	Moving Reference Frame
Type of Geometry	- Single Rotationally Periodic Blade Passage - Volute
Interfaces	- Mixing Plane Interface at Rotor Outlet and Volute Inlet Boundaries - Periodic Interface at Rotor Periodic Boundaries
Inlet Boundary Condition	Total Pressure and Temperature
Outlet Boundary Condition	Static Pressure

A representative view of the volute cross section is provided in FIGURE 9. Notice there are several boundary layers near the physical wall surfaces. Twelve boundary (prism) layers were specified for this mesh. The mesh density gradually increases moving away from the wall and into the bulk flow region based on a prism layer stretch factor of 1.5. A mesh size was specified such that approximately 12 elements span the channel height of the bulk flow region of the rotor outlet / volute inlet, for a total of approximately 36 elements across the channel height accounting for the prism layers at the walls.

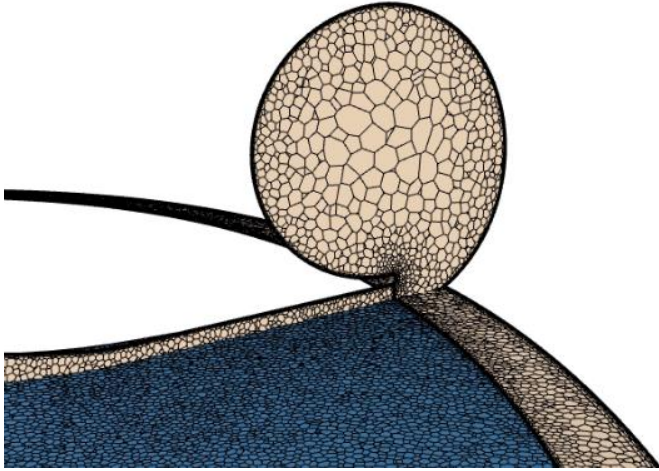


FIGURE 9: Mesh cross-section through the volute plane and rotor isosurface

Each region was meshed using a polyhedral mesh. The number of mesh elements varied slightly with the higher neck size designs having more mesh elements. An overview of the mesh statistics across all the designs is provided in TABLE 3. While a mesh sensitivity study was not conducted for this analysis, a mesh sensitivity study performed previously for NASA's HECC can be found in literature [14]. In that study, mesh sizes ranging from 1.5 – 4.5 million elements resulted in similar CFD performance predictions. Given the number of mesh elements in the rotor falls within this range, this provides more confidence that the mesh in our study is sufficiently refined. The steady-state CFD analysis of the NASA's HECC in literature [14] also resulted in good agreement between CFD prediction and test data. Given the similarities in the overall flowpath between the analyzed flowpath and NASA's HECC, this provides additional confidence that the performance predictions of our fluid domain in CFD will also match real-world behavior. Additionally, volute-focused studies referenced, including [12], [9], and [8], used similar or smaller numbers of elements successfully, providing further confidence of the propriety of the presently-used computational grid.

TABLE 3: Mesh element statistics

Domain	Range of Number of Mesh Elements
Rotor	2.2 – 2.3 million
Volute	2.9 – 4.4 million
Rotor and Volute	5.1 – 6.7 million

The range of the mesh wall Y^+ values for the different volute designs at the design operating point is provided in TABLE 4. In the impeller domain, the average Y^+ value remains low for all the designs. While there are some mesh elements near the blade surfaces that experience high Y^+ values, these high values are consistent across all the designs. Additionally, since it is primarily the volute performance which is of interest, the impeller mesh is acceptable. For the volute domain, the average Y^+ value also remains low. Additionally, the maximum Y^+ value

remains low (<5) for all volutes except one (CMD 0mm volute) where there is a local region of elevated Y^+ values near the volute inlet. It is not anticipated for this local region of higher Y^+ values to significantly shift the machine performance prediction from CFD, so the volute mesh is also acceptable.

TABLE 4: Mesh wall Y^+ statistics

Domain	Wall Y^+ Values
Impeller	Average: 4.8 – 5.1
	Minimum: 0.024 – 0.048
	Maximum: 301 – 418
Volute	Average: 1.8 – 2.2
	Minimum: 0.23 – 0.70
	Maximum: 4.0 – 115

For each volute, a speedline was calculated at the design speed with 6-8 operating points analyzed for each speedline. Air was chosen as the working fluid. A fixed value for total temperature and total pressure were specified at the rotor inlet boundary, while static pressure was specified at the volute outlet boundary which varied depending on the operating point. Rotational speed was also specified for the rotating surfaces. Also note that while an initial vector field was specified, the final inlet flow angles were calculated based on the CFD solution. A summary of the fixed CFD operating conditions is given in TABLE 5 while the variable static pressure outlet boundary conditions are marked by an 'X' in TABLE 6.

TABLE 5: Fixed operating condition specifications in CFD

	Analysis Property
Working Fluid	Air
Total Inlet Temperature (Rotor)	15 °C
Total Inlet Pressure (Rotor)	1 atm
Rotational Speed	19620 rpm

TABLE 6: Volute static outlet pressure boundary conditions in CFD

		Volute Type and Neck Size [mm]						
		P-Type	CMD 0mm	CMD 11mm	CMD 22mm	CMD 34mm	CMD 45mm	CMD 56mm
Volute Static Outlet Pressure [bar]	2.00	X	X	X	X	X	X	X
	2.16	X						
	2.50		X	X	X	X	X	X
	2.90	X						
	3.00		X	X	X	X	X	X
	3.33	X	X	X	X	X	X	X
	3.40		X	X	X	X	X	X
	3.42	X						
	3.45	X	X	X	X			
	3.50			X	X	X	X	X
	3.55			X	X	X	X	X
	3.60					X		

Mass flow averaged and surface averaged parameters were extracted at several boundary surfaces. Additional parameters were extracted across a derived surface created near the impeller trailing edge to separate and quantify the performance across the impeller blade as well as the downstream stationary flow components. The extracted CFD data was then externally post-processed to compare the performance of the different volute designs.

3. RESULTS AND DISCUSSION

A speedline was calculated at the design speed using CFD for each volute type. Note that due to static pressure being specified at the outlet boundary, the stall point shown in the performance curves is not the exact operating point of stall. However, the comparison of the regions of the performance curves which were analyzed still allow conclusions to be drawn about the different volute geometries. Performance curves showing the total-total pressure ratio, total-total isentropic efficiency, and power consumption versus mass flow rate at the design speed for different volutes are shown in FIGURE 10 - FIGURE 12.

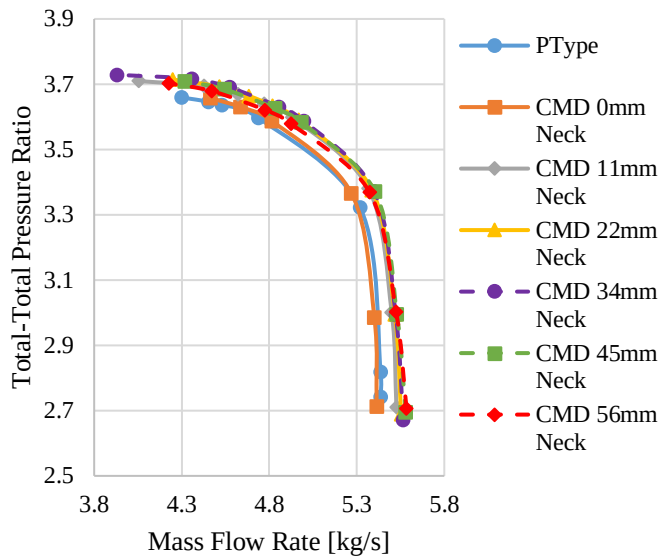


FIGURE 10: Total-total pressure ratio vs. mass flow rate for different volutes

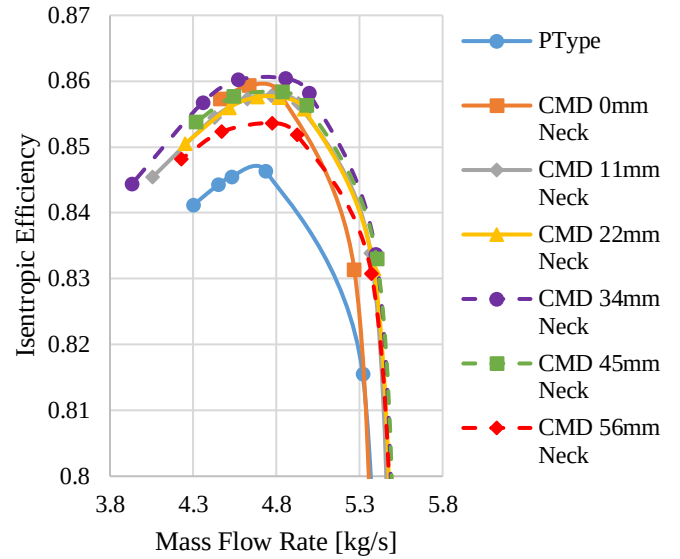


FIGURE 11: Total-total isentropic efficiency vs. mass flow rate for different volutes

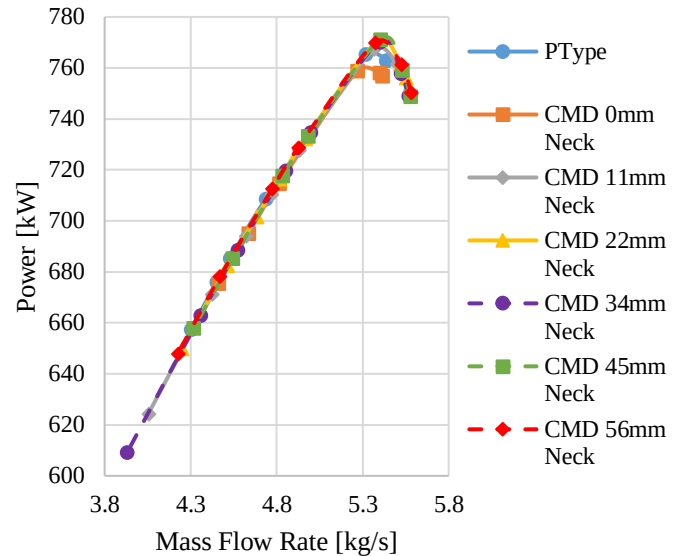


FIGURE 12: Power consumption vs. mass flow rate for different volutes

Additional plots were generated isolating the performance of the stationary diffusion elements of the flowpath (vaneless diffuser and volute). To analyze these downstream flowpath elements, the pressure recovery coefficient and total pressure loss coefficient were calculated across the speedline for each volute. The pressure recovery coefficient describes the static pressure recovery across the downstream flowpath elements relative to the dynamic pressure entering the vaneless diffuser (Equation (8)). The total pressure loss coefficient describes the total pressure loss across the stationary downstream flowpath elements relative to the dynamic pressure entering the vaneless

diffuser (Equation (9)). The trends of these parameters for each volute design are provided in FIGURE 13 and FIGURE 14.

$$C_p = \frac{P_{s2} - P_{s1}}{P_{t1} - P_{s1}} \quad (8)$$

$$\zeta = \frac{P_{t2} - P_{t1}}{P_{t1} - P_{s1}} \quad (9)$$

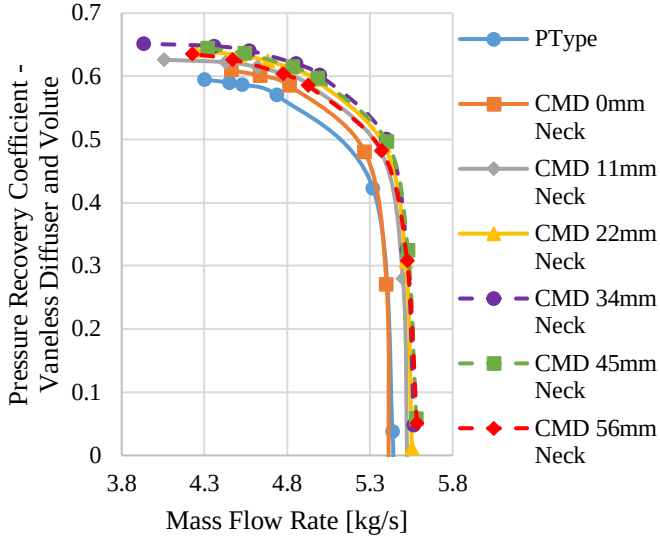


FIGURE 13: Pressure recovery coefficient across vaneless diffuser and volute vs. mass flow rate for different volutes

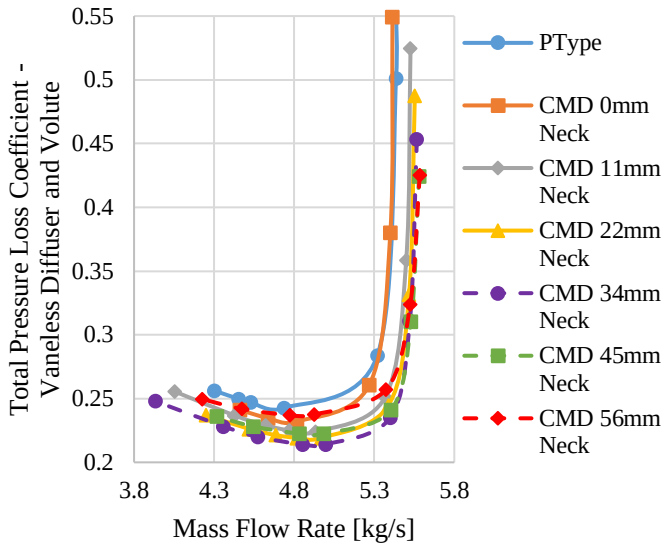


FIGURE 14: Total pressure loss coefficient across vaneless diffuser and volute vs. mass flow rate for different volutes

These results indicate that there is some performance benefit to be gained from adding a volute neck. Even adding a relatively small neck sees improvements in pressure rise capability and efficiency. Additionally, the volute experiences increased

pressure recovery and reduced pressure loss. Eventually, an optimal neck size is reached where the maximum pressure rise and efficiency is achieved. The optimal neck size for this study is 34mm, corresponding to an S_{neck} of 0.315. Increasing the neck size past this optimum size results in a reduction in pressure rise and efficiency.

One unexpected result of this analysis is that adjusting the volute type moved the flow rate at which choke occurs. Choke behavior is typically dictated by normal shock development in the impeller. Since the impeller portion of the flowpath remained the same between analyses, it was expected the choke behavior would remain the same as well. Upon further analysis, it was revealed that these designs experienced simultaneous choking behavior in both the volute and impeller. FIGURE 15 and FIGURE 16 show a high Mach number local flow region which developed near the exit of the volute at the choking operating point for one of the designs (P-Type volute in this case). FIGURE 17 shows a normal shock development near the leading edge of the impeller for the same flowpath design and choking operating point. It is hypothesized that if the volute study were repeated where the volute sizes were increased, then choking behavior would be controlled solely by the normal shock development in the impeller, leading to the same choke flow rate across all the designs.

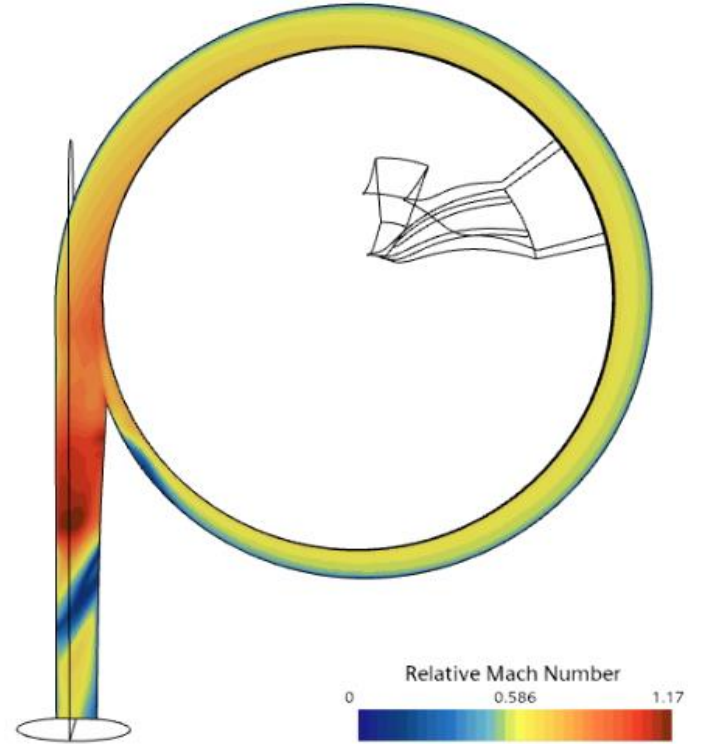


FIGURE 15: Relative Mach number along volute inlet midplane for P-Type volute

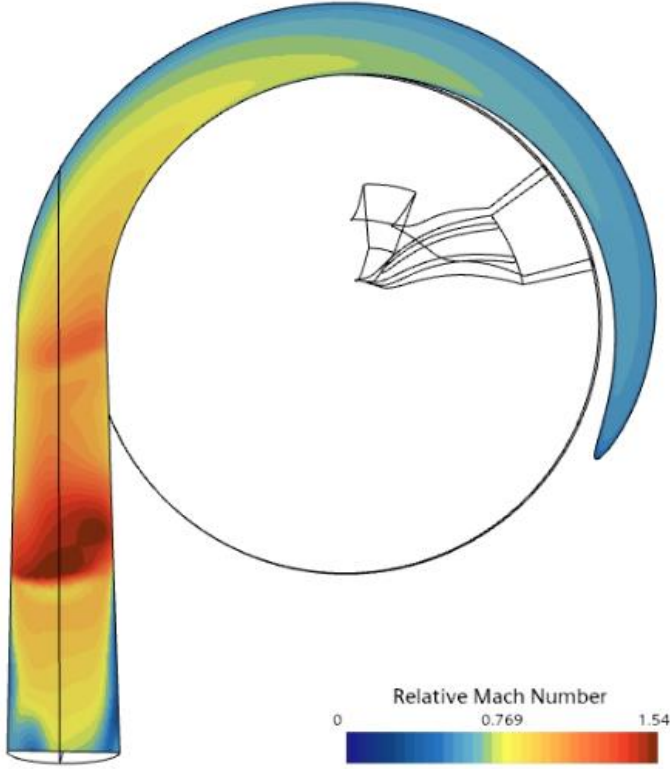


FIGURE 16: Relative Mach number along volute outlet midplane for P-Type volute

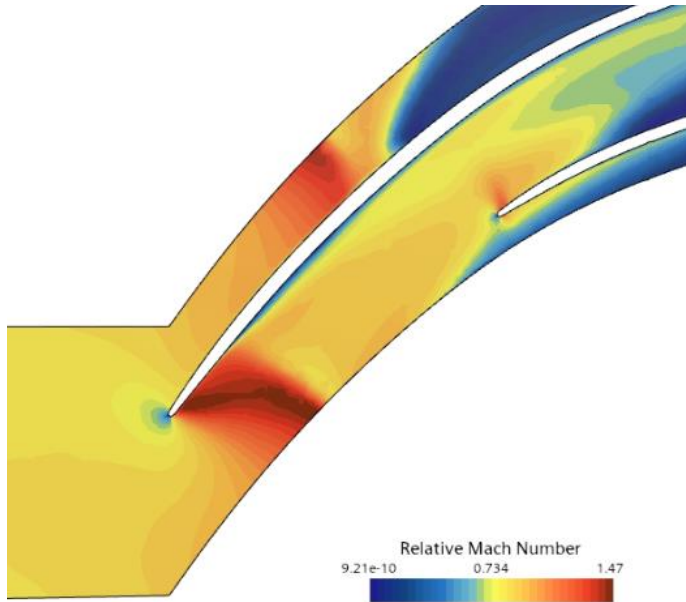


FIGURE 17: Relative Mach number near tip section of impeller leading edge

To further understand how the volute neck affected the flow, methods were used to quantify the amount of flow swirl and meridional flow at different positions around the volute. To do this, cross sections were taken at specific locations along the volute and integral flow parameters were calculated for each cross-sectional surface. The locations of each cross section are highlighted in FIGURE 18. Note that the location of the ‘Volute Outlet’ plane is slightly offset from the actual CAD volute outlet boundary to mitigate fallacies arising from boundary-specification effects from its analysis.

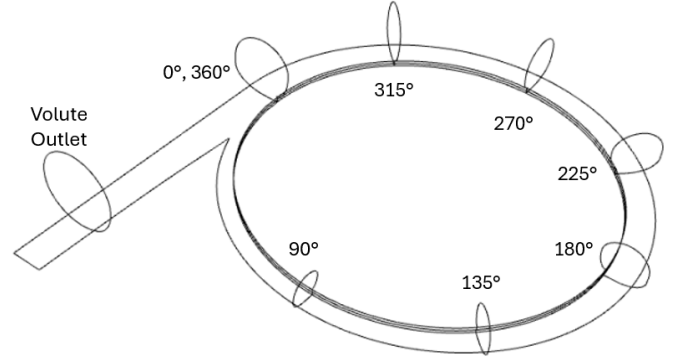


FIGURE 18: Volute cross section locations

While there are several similar definitions of swirl number found in literature [21] [22] [23] [24], they all relate the angular momentum flux across a surface to the axial momentum flux across a surface. The definition of swirl number used in this analysis is given in Equation (10). Breaking down the swirl number equation further, the swirl number is a function of the angular flow velocity (Equation (11)), meridional velocity (Equation (12)), tangential flow velocity (Equation (13)), flow surface area (Equation (14)), and fluid density (ρ).

$$Swirl\ Number = \frac{\oint |\vec{r} \times \vec{v}_\theta| \rho (\vec{v} \cdot \hat{n}) dS}{\left(\frac{\oint r dS}{\oint dS} \right) \oint |v_m| \rho (\vec{v} \cdot \hat{n}) dS} \quad (10)$$

$$\vec{\omega}_s = \vec{r} \times \vec{v}_\theta = r \vec{v}_{\theta,S} \quad (11)$$

$$v_m = \vec{v} \cdot \hat{n} \quad (12)$$

$$\vec{v}_\theta = \vec{v} - (\vec{v} \cdot \hat{n}) \hat{n} \quad (13)$$

$$S = \oint dS \quad (14)$$

FIGURE 19 compares the swirl number distribution at different angular locations around the scroll portion of the volute for the different volute types, while FIGURE 20 highlights the percentage difference in swirl number with the different volute types. In both figures, the x-axis shows the angular position around the volute scroll, while the y-axis shows the mass flow rate (operating point). The color axis shows either the local swirl number (FIGURE 19) or the percent difference in local swirl number relative to a baseline configuration, which was selected to be the CMD 0mm Neck (FIGURE 20). FIGURE 19 indicates

that low flow rates near stall, swirl peaks near the 0° and 180° positions. As flow rates increase, the amount of swirl increases along the entire volute. Near choke flow rates the location of peak swirl shifts slightly towards the 90° and 135° positions, with larger neck designs having peak swirl at an earlier angular volute position.

When looking at the percentage difference in swirl number in FIGURE 20, adding a larger neck leads to the most change in swirl near the 225° volute position. There is also a large change in swirl number at the 0° position near choke flow due to the position of this cross section being close to the high Mach number region near the volute exit.

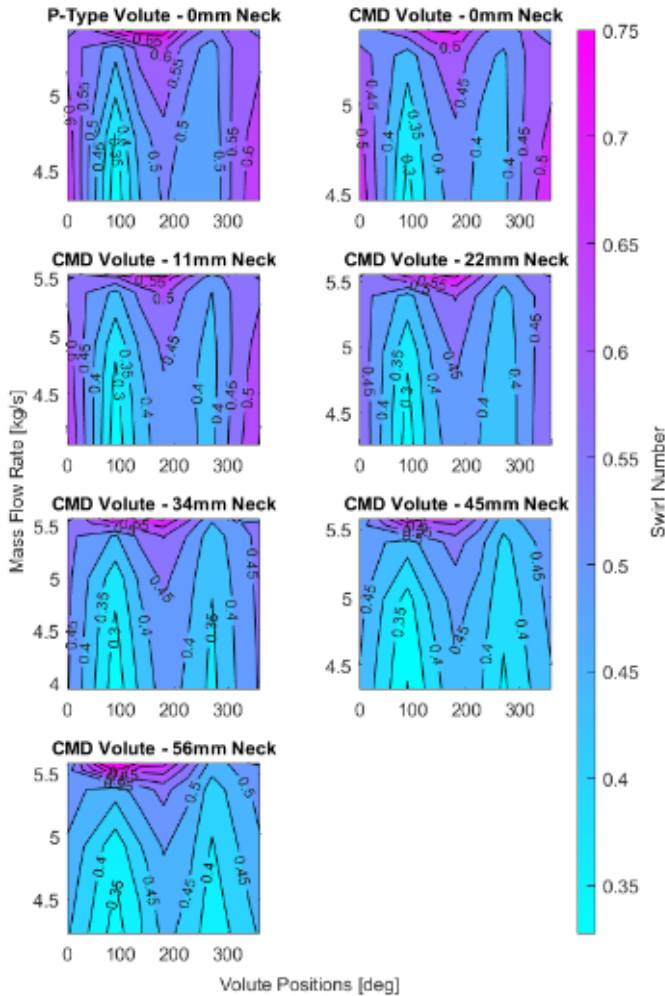


FIGURE 19: Swirl number distribution for different angular positions around the volute scroll and different mass flow rates

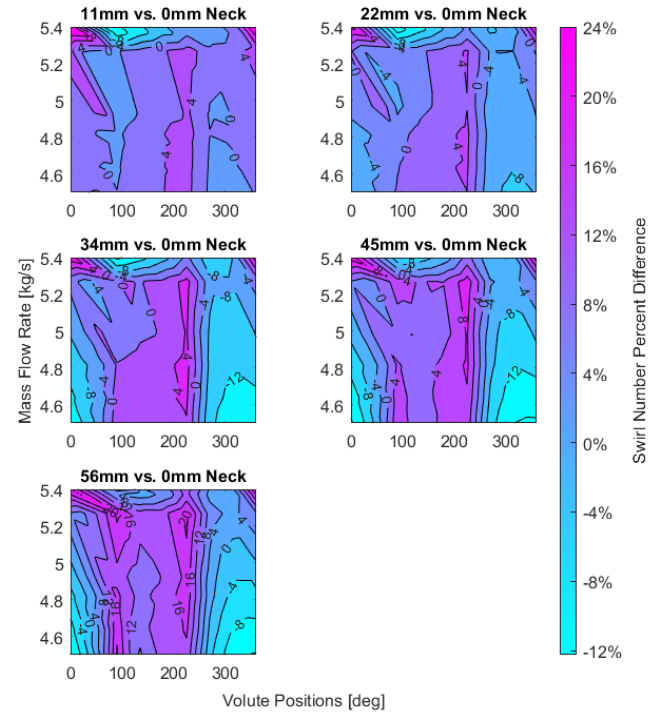


FIGURE 20: Swirl number percent difference distribution for different angular positions around the volute scroll and different mass flow rates

Analyzing the swirl number at the volute outlet (FIGURE 21) shows that volutes with reduced swirl tend to lead to better performance. If we ignore the CMD 0mm neck volute, then increasing the neck size leads to a minimization of swirl number for the volutes, since the volutes with 11mm, 22mm, and 34mm all have a similar low swirl number while also having better performance. Further increase in neck size leads to increased swirl number near the outlet, which results in higher losses and worse performance. It should be acknowledged that both P-type and CMD 0mm neck cases, as seen in FIGURE 21, are exceptions to the trend. This is in spite of the fact that both P-type and CMD 0mm neck cases are geometrically nearly identical and indeed show nearly identical and show similar and consistent trends for overall compressor and diffusion system performance as evidenced by FIGURE 10, FIGURE 13, and FIGURE 14.

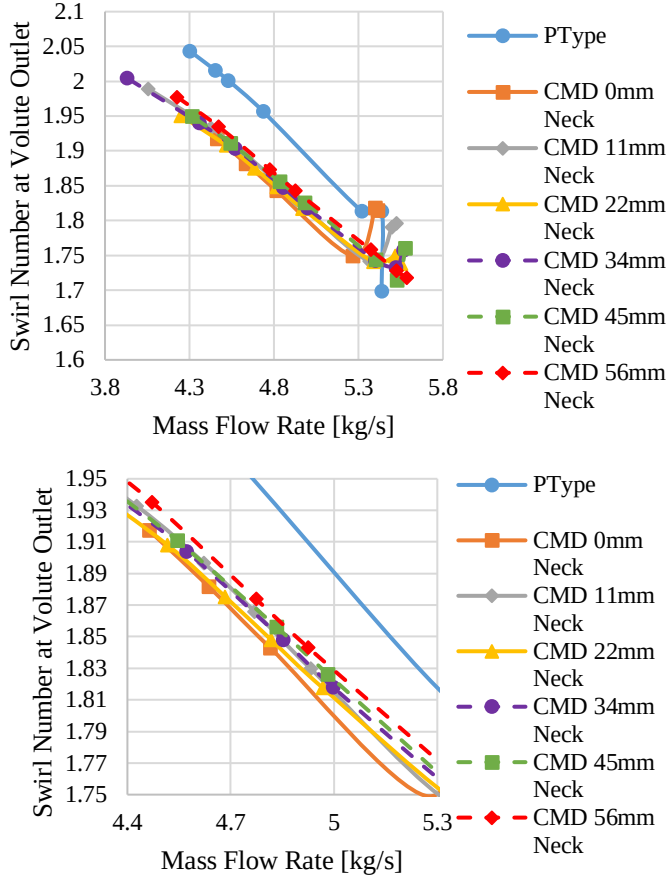


FIGURE 21: Swirl number at volute outlet across entire flow range (top) and zoomed-in portion of flow range (bottom)

In addition to the flow swirl, the meridional velocity was also analyzed throughout the volute. To do this, a mass flow averaged meridional velocity ($\overline{v_m}$) was computed at each volute cross section using Equation (15). The trend of meridional velocity at the volute outlet is shown in FIGURE 22, where volutes with larger neck sizes tend to have slightly reduced meridional flow velocities.

$$\overline{v_m} = \frac{\oint |v_m| \rho (\vec{v} \cdot \hat{n}) dS}{\oint \rho (\vec{v} \cdot \hat{n}) dS} = \frac{\oint v_m^2 \rho dS}{\oint \rho v_m dS} \quad (15)$$

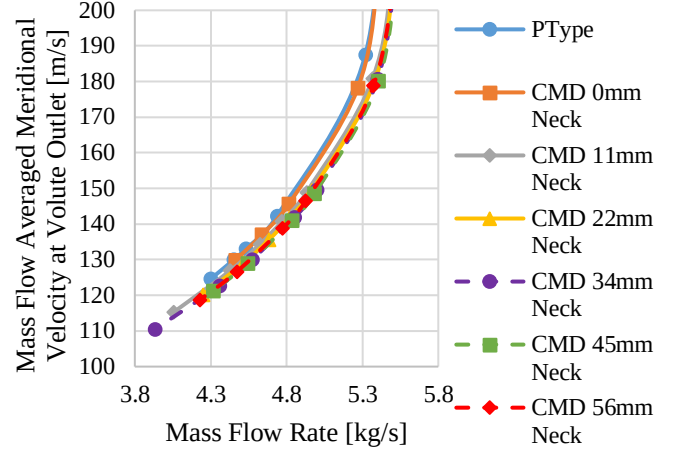


FIGURE 22: Mass flow averaged meridional velocity at volute outlet

The series of images in FIGURE 23 show how the tangential and meridional components of flow at the design point change for the different volute types (rows) and volute cross section locations (sets of columns). Reviewing the tangential velocity scenes shows that for volute designs with no neck, there is a local region of flow near the volute's low-radius end wall with very little tangential velocity. As the volute neck increases, the region of low tangential velocity near the low-radius end wall shrinks, and the flow velocity is more consistent with the neighboring velocity field.

As the flow enters the volute, its tangential flow component causes it to eventually hit the volute wall and start to turn in the volute meridional plane. For volutes with small necks, the flow turns in such a way that there is flow separation near the volute wall (i.e. the tangential velocity goes to near zero). This is indicative that there is a local flow recirculation region here, leading to increased volute losses. The meridional flow contours also indicate this trend, with relatively larger local regions of low meridional flow near this region of the volute walls, further indicating that there is a region of local flow recirculation.

For volutes with large necks, the flow follows along the outer wall of the volute with little separation. While the peak flow velocities decrease with volutes of increasing neck size, the high tangential velocity regions do not dissipate as much near the outer surfaces of the volute cross section. The meridional flow contours look similar for the volutes with medium and high neck sizes. The combination of these tangential and meridional flow results indicates that there is slightly better flow diffusion and static pressure recovery for the medium neck designs. This trend matches those seen in FIGURE 13 and FIGURE 14.

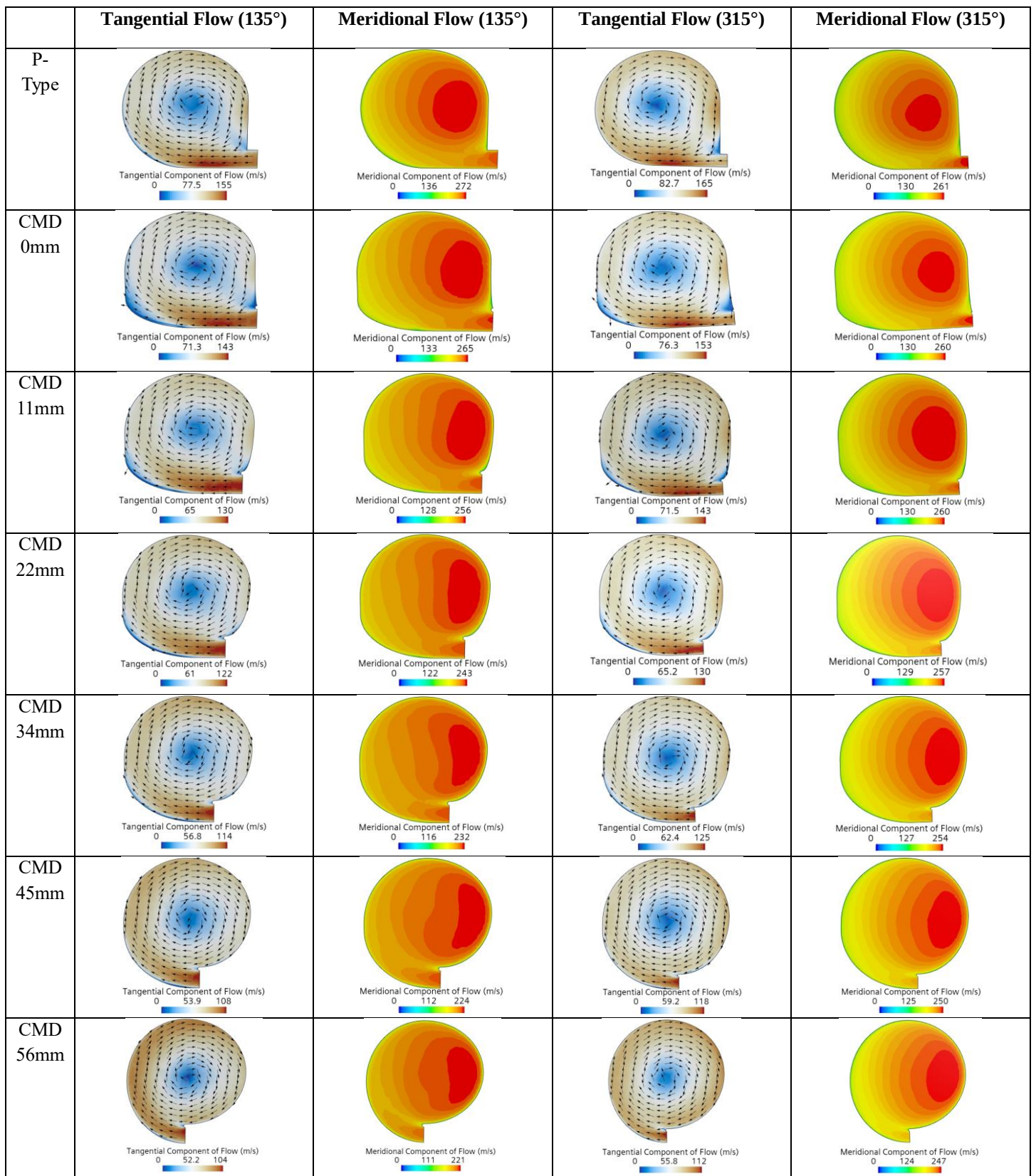


FIGURE 23: Tangential flow vector field and meridional flow scalar field for each volute at 135° volute location (left 2 columns) and 315° volute location (right 2 columns) at the design point

4. CONCLUSION

The analytical results of this volute neck study indicate that having a volute with some neck results in an improvement in machine performance thanks to the slight increase in pressure recovery and reduction in pressure loss through the volute. However, there comes a point where further increasing the neck size results in diminishing performance improvements and ultimately a slight reduction in pressure rise and efficiency due to the reduced pressure recovery near the volute walls. Therefore, for a particular compressor design there will be an optimal neck size which optimizes the volute performance across the operating range.

For the analyzed compressor, there appears to be an optimum in both pressure recovery and total pressure loss as the volute neck extends 31% into the overall height of the volute cross section ($S_{neck} \approx 0.31$). For this optimum volute, the pressure recovery coefficient peaks at 0.65, compared to 0.59 for the P-Type no neck volute. Correspondingly, the optimum volute lowers the total pressure loss coefficient to a minimum of 0.21, compared to 0.25 for the P-Type volute. Lastly, the peak compressor efficiency can be improved from 84.6% to 86.0% by switching from a P-Type to the optimum volute, with a corresponding increase in overall pressure ratio of 1.9%.

ACKNOWLEDGEMENTS

This study was internally supported by SoftInWay Inc.

REFERENCES

- [1] M. Heinrich, Genetic Optimization of Turbomachinery Components using the Volute of a Transonic Centrifugal Compressor as a Case Study, Freiberg: Technische Universität Bergakademie Freiberg, 2016.
- [2] Ingersoll Rand, "IRP-En-Brochure-process-air-and-gas-compressors-MSG-Centrifugal-Compressors-Air-Separation-and-Industrial-Gases.pdf," [Online]. Available: <https://encrypted-tbn0.gstatic.com/images?q=tbn:ANd9GcTo5pu4jaxlEVIy5s36xk7GGsXyKFNIuecqKaW6YAhcw&s>. [Accessed 3 12 2025].
- [3] Compression Service Technology, "Theory at a Glance," CST, [Online]. Available: <https://www.cstfirenze.com/knowledge/theory-at-a-glance>. [Accessed 4 December 2025].
- [4] R. Aungier, Centrifugal Compressors, New York: ASME Press, 2000.
- [5] L. Xingsu, P. Kunyuan and W. Zomin, "Aerodynamic Design and Experimental Study of Marine Gas Turbine Exhaust Volute," in *ASME 1981 International Turbine Conference and Products Show*, Houston, Texas, 1981.
- [6] M. Heinrich and R. Schwarze, "Genetic Algorithm Optimization of the Volute Shape of a Centrifugal Compressor," *International Journal of Rotating Machinery*, vol. 1, p. 4849025, 2016.
- [7] T. Ceyrowsky, A. Hildebrandt, M. Heinrich and R. Schwarze, "Assessment of the Loss Map of a Centrifugal Compressor's External Volute," in *ASME Turbo Expo 2020: Turbomachinery Technical Conference and Exposition*, Virtual, Online, 2020.
- [8] H. Zhang, S. Zhang, S. Wan, Y. Lia, J. Yang, S. Xu and W. Wei, "Shape Optimization of Exhaust Volute for a Mixed Working Fluid Centrifugal Compressor," *International Journal of Gas Turbine, Propulsion, and Power Systems*, vol. 15, no. 4, pp. 16-24, July 2024.
- [9] R. Hottois, A. Châtel and T. Verstraete, "Adjoint-Based Design Optimization of a Volute for a Radial Compressor," *International Journal of Turbomachinery, Propulsion, and Power*, vol. 8, no. 4, p. 41, October 2023.
- [10] R. A. Adjei, C. Fan, S. Ijaz and H. Zou, "Aerodynamic Optimization for Stability Improvement of a Centrifugal Compressor Using Arbitrary-Shape Volute Design," *Case Studies in Thermal Engineering*, vol. 73, p. 106654, 2025.
- [11] P. Sitkin, Optimizations of the Volute of a Centrifugal Compressor, Helsinki: Lappeenranta University of Technology, 2019.
- [12] P. F. Melani, F. Balduzzi, A. Bianchini, G. Ferrara, P.-A. Hoffer, S. Montesino and M. Brenner, "Influence of Key Design Parameters on the Aerodynamic Performance of a Centrifugal Compressor Volute for Turbocharger Applications," in *ASME Turbo Expo 2020: Turbomachinery Technical Conference and Exposition*, Virtual, Online, 2020.
- [13] J. Lei, G. Qin and Q. Cui, "Aerodynamic Performance Improvement and Noise Control for the Multi-Blade Centrifugal Fan in Air Conditioner by Using Concave Volute Tongue," in *ASME Turbo Expo 2024: Turbomachinery Technical Conference and Exposition*, London, United Kingdom, 2024.
- [14] G. Medic, O. P. Sharma, J. Jongwook, L. W. Hardin, D. C. McCormick, W. T. Cousins, E. A. Lurie, A. Shabbir, B. M. Holley and P. R. Van Slooten, "High Efficiency Centrifugal Compressor for Rotorcraft Applications (NASA/CR - 2014-218114)," NASA, East Hartford, CT, 2014.
- [15] H. Harrison, "NASA High Efficiency Centrifugal Compressor Data Archive," NASA, Cleveland, OH, 2023.
- [16] H. M. Harrison, E. O. McNichols and M. R. Blaha, "NASA Small Engine Components Compressor Test Facility: High Efficiency Centrifugal Compressor Vaneless Diffuser and Transition Duct Configurations," in *ASME Turbo Expo*, Boston, MA, 2023.
- [17] L. Moroz, Y. Govoruschenko, P. Pagur and L. Romanenko, "Integrated Conceptual Design Environment for Centrifugal Compressors Flow Path Design," in *ASME International Mechanical Engineering Congress and Exposition*, 2008.

- [18] Siemens Digital Industries Software, *Simcenter STAR-CCM+*, version 2410.1, Siemens, 2024.
- [19] Siemens Digital Industries Software. Simcenter STAR-CCM+ User Guide, version 2410.1, *Equations of Fluid Flow*, Siemens, 2024, pp. 8637-8638.
- [20] Siemens Digital Industries SoftWare. Simcenter STAR-CCM+ User Guide, version 2410.1, *Transport Equations*, Siemens, 2024, pp. 8809-8810.
- [21] G. Vignat, D. Durox and S. Candel, "The suitability of different swirl number definitions for describing swirl flows: Accurate, common and (over-) simplified formulations," *Progress in Energy and Combustion Science*, vol. 89, no. 100969, 2022.
- [22] C. Stone and S. Menon, "Swirl Control of Combustion Instabilities in a Gas Turbine Combustor," in *Proceedings of the Combustion Institute*, Atlanta, GA, 2022.
- [23] Y. Zhang, D. Shimokuri, Y. Mukae and S. Ishizuka, "Flow Field in Swirl-Type Tubular Flame Burner," *JSME International Journal*, vol. 48, no. 4, pp. 830-838, 2005.
- [24] T. C. Claypole and N. Syred, "The Effect of Swirl Burner Aerodynamics on NO_x Formation," in *Symposium (International) on Combustion*, Cardiff, United Kingdom, 1981.

APPENDIX

TABLE 7: Detailed parameters of volute designs

	P-Type	CMD 0mm Neck	CMD 11mm Neck	CMD 22mm Neck	CMD 34mm Neck	CMD 45mm Neck	CMD 56mm Neck
$R_{maximum}$ [mm]	386.6	384.0	384.0	384.0	384.0	384.0	384.0
R_{inlet} [mm]	274.3	274.3	285.5	296.8	308.0	319.3	330.5
$R_{minimum}$ [mm]	274.3	274.3	274.3	274.3	274.3	274.3	274.3
Neck Size [mm]	0	0	11.2	22.5	33.7	44.9	56.2
hk [mm]	107.0	107.0	107.0	107.0	107.0	107.0	107.0
S_{neck}	0	0	0.105	0.210	0.315	0.420	0.525
$D2m$ [mm]	330.5	330.5	330.5	330.5	330.5	330.5	330.5
Inlet Channel Width [mm]	9.5	9.5	9.5	9.5	9.5	9.5	9.5
$A2$ [mm ²]	13525	13525	13525	13525	13525	13525	13525
A_{th} [mm ²]	9017	9017	9017	9017	9017	9017	9017
$LenDif$ [mm]	500	500	500	500	500	500	500
Elliptic Form Coefficient	n/a	0.5	0.6	0.7	0.8	0.9	1