Simulation of Turbulent Swirling Flow in Convergent Nozzles

A.Nouri- Borujerdi, A. Kebriaee
School of Mechanical Engineering
Sharif University of Technology, Tehran, Iran

Abstract

This work simulates boundary layer behavior of a turbulence incompressible viscous flow in a convergent nozzle. The inlet velocity vector has two components in axial and tangential directions. The governing boundary layer equations are solved by integration method. The results show that the radial and tangential boundary layer thicknesses depend on the velocities ratio, Reynolds number and nozzle angle. The peak of radial and tangential boundary layer thicknesses are located at \( z/L \approx 0.2 \) and \( z/L \approx 0.8 \) from the nozzle inlet respectively. It is found that large inlet velocities ratio decreases the radial boundary layer thickness due to centrifugal force. The pressure loss in the nozzle is very large and it is caused by the momentum changes. The contribution of the shear stress is not significant on the pressure loss because of the short length of the nozzle. The results are also in a good agreement with the pervious data.

Nomenclature

\[ C_f \] friction coefficient
\[ D \] nozzle diameter, m
\[ L \] nozzle length, m
\[ P \] Pressure, atm.
\[ R \] Radius of nozzle cross section, m
\[ Re \] Reynolds number, \( Re = \frac{Ri u_i}{v} \)
\[ S \] swirl number, \( S = \frac{\text{swirling momentum flux}}{\text{axial momentum flux}} \)
\[ u \] velocity, m/s
\[ V \] velocity vector
I. Introduction

Investigation of swirling flow is an important topic to study pressure-swirl injectors, industrial hydrocyclones and swirl chamber burners. Taylor [1] was pioneer in the study of boundary layer swirling flow. He applied Pohlhausen method to study the behavior of the boundary layer growth in the conical coordinate and especially in pressure-swirl injectors. He determined radial and tangential velocities distribution across the boundary layer and growth of the boundary layer thickness along the nozzle for laminar flow.

Binnie and Harris [2] investigated swirling boundary layer before formation of air core in the nozzle axis. They found that the effect of surface tension in the nozzle core was negligible compared to other forces. Weber [3] introduced a universal function for turbulent shear stress on the wall and extended Taylor’s attempts to the turbulent boundary layer. Kreith and Sonju [4] presented a linearized theory for the average decay of turbulent swirling flow.
through a pipe. They examined the flow in the range of $10^4 < \text{Re} < 10^5$ and showed swirling decay is more considering in the low Reynolds numbers. Moreover, the swirl intensity decreases with higher gradient in the entrance region of the pipe. Kiya et al. [5] applied a finite difference method invented by Leigh and Terril method to investigate the laminar boundary layer swirling flow in the entrance region of a circular pipe. They reported swirling decay is faster than an exponential trend at near the inlet. Back [6] used a numerical method to solve self-similar boundary layer equation of swirling flow. He found out heat transfer in a convergent nozzle increases with swirling flow increment. Yajnik and Subbaiah [7] considered the swirl effects on turbulent flow with similarity solution. They admitted the similarity of velocity behavior for sufficiently large Reynolds numbers, provided that reversal flow does not occur in the solution zone. Fabian and Oates [8] described the boundary layer inside the cone of a swirling flow. Their solution was based upon the Karman-Pohlhausen method for both laminar and turbulent flows. Their results were in close agreement with known solutions previously obtained in the limits of swirling flow without axial component or axial flow without swirling flow. Bloor and Ingham [9] applied Pohlhausen method utilized by Taylor to model the three-dimensional boundary layer equation involving a swirling flow. Singh et al. [10] used the asymptotic expansion method to investigate the laminar axisymmetric swirling flow of a viscous incompressible fluid in the entrance region of a pipe. They divided the swirling flow into three regions and investigated each zone behavior separately. Kumari and Nath [11] applied the finite difference method to study pressure drop and heat transfer in the laminar boundary layer of swirling flow. They stated dominant proportion of the skin friction was longitudinal friction rather than the tangential one. They also predicted the distribution of the axial and tangential velocities near the wall. Kumari and Nath [12] also extended their implicit finite difference method to study heat and mass transfer in the unsteady laminar boundary layer of swirling flows. Their results illustrated that the swirl velocity at the edge of the boundary layer is non free vortex type in contrast to a steady circular motion. In addition, the longitudinal and swirl velocities at the edge of the boundary layer vary inversely as a linear function of time. The skin-friction and heat transfer are strongly affected by the swirl velocity, mass transfer, and Prandtl number. Padmanabhan [13] employed the similarity solution to investigate a laminar blood flow in the cardiovascular system involving tangential inlet and consequently swirling component velocity. He determined the distribution of shear stress in aorta.

Bhattacharyya [14, 15] continued Taylor and Weber attempts with another turbulent velocity profile. He considered the effect of Reynolds number through a convergent nozzle and employed 1/10th power law rather than
1/7th power law in his study. Akiyama and Ikeda [16] investigated the turbulent boundary layer of a swirling flow in a pipe. They used the turbulent shear stress on the wall defined by Weber, but with different axial and tangential velocities distribution. Also, they studied analytically the behavior of the tangential velocity decay along the pipe to justify their experimental data. Najafi et al. [17] applied the forced vortex flow in the core of a pipe flow to integrate boundary layer equations. They presented the skin friction coefficient and boundary layer growth along the pipe and justified their practical attempts. Conch [18] reviewed different mathematical techniques such as asymptotic expansions, similarity solutions and boundary layer flows to study hydrocyclones behavior. He applied different boundary conditions with various geometries to find three components of velocity vector and pressure distribution along the industrial hydrocyclones.

In the present work, an analytical solution is proposed on turbulent boundary layer of a swirling flow to determine swirling decay and pressure loss through a conical swirling chamber. To model the radial and tangential velocity components across the boundary layer, a 1/7th power law profile is applied. The pressure gradient normal to the wall has been considered more accurately than the previous attempts. Final results are studied on pressure-swirl atomizer geometry.

II. Governing Equations

Using the boundary layer approximations for an axi-symmetric convergent nozzle, i.e., $u_\theta \ll u_r$, $u_\phi$, $\partial(\ )/\partial \phi = 0$ and $\partial^2 / \partial r^2 \ll \partial^2 / \partial \theta^2$, the continuity and spherical Navier-Stokes equation in r-, $\phi$- and $\theta$- components can be simplified respectively as follows, Fig. (1).

\[
\frac{\partial u_r}{\partial r} + r \frac{\partial u_\theta}{\partial \theta} = 0
\]

\[
U_r \frac{\partial u_r}{\partial r} + u_\theta \frac{\partial u_r}{\partial \theta} - \frac{u_\phi^2}{r} = -\frac{1}{r^2} \frac{\partial p}{\partial r} + \frac{1}{r^2} \frac{\partial}{\partial r} \left[ \nu \frac{\sin \theta}{r} \frac{\partial u_r}{\partial \theta} \right]
\]

\[
U_r \frac{\partial u_\phi}{\partial r} + u_\theta \frac{\partial u_\phi}{\partial \theta} + \frac{u_\phi u_r}{r} = \frac{1}{r^2} \frac{\partial}{\partial r} \left[ \nu \frac{\sin \theta}{r} \frac{\partial u_\phi}{\partial \theta} \right]
\]
\[- \frac{u^2}{r} \cot \theta = - \frac{1}{\rho r} \frac{\partial p}{\partial \theta}\]  

**Boundary Conditions:** the boundary conditions of the governing equations are as follows.

\[u_r(r, \theta_o) = 0, \quad u_r(r, \theta_o - \frac{\delta}{r}) = u_{rc}\]
\[u_\phi(r, \theta_o) = 0, \quad u_\phi(r, \theta_o - \frac{\delta}{r}) = u_{\phi c}\]
\[u_r(r_i, \theta) = u_{r i}, \quad u_\phi(r_i, \theta) = u_{\phi i}\]
\[p(r, \theta_o - \frac{\delta_{\max}}{r}) = p_{\delta_{\max}}, \quad p(r_i, \theta_o) = p_i\]
\[u_\phi(r_o, \theta_o) = 0\]  

**III. Solution Procedure**

To integrate Eqs. (1-4), the 1/7- power law profile is used in \(r\)- and \(\phi\) – directions respectively for a turbulent viscous flow. Based on Weber [3] and Najafi et al. [17], the 1/7-power law is valid for all inlet Reynolds numbers. Bhattachryya [14] has solved the integral boundary layer method for inlet Reynolds number of 1800 with 1/7-power law. In another research, Bhattachryya [15] extended his work for higher inlet Reynolds number up to three times and used a velocity profile with 1/7- and 1/10- power laws. He concluded the 1/7-power law has sufficient accuracy to model the flow field in the swirl boundary layer for higher Reynolds number.

\[\frac{u_r}{u_{rc}} = \left[\frac{y}{\delta_r}\right]^{\frac{1}{7}}\]  
\[\frac{u_\phi}{u_{\phi c}} = \left[\frac{y}{\delta_\phi}\right]^{\frac{1}{7}}\]  

where \(y = r(\theta_o - \theta)\) is measured inward and normal to the \(r\)-direction. \(u_{rc}\) is radial core velocity outside the boundary layer and is calculated by mass balance equation across the nozzle cross section as:

\[\frac{\dot{m}}{\rho} = \int_0^{2\pi} \int_0^{\theta_o - \delta_\phi / r} u_{rc} r^2 \sin \theta d\theta d\phi + \int_0^{2\pi} \int_{\theta_o - \delta_\phi / r}^{\theta_o} u_r r^2 \sin \theta d\theta d\phi\]

where \(\dot{m} = 2\pi \rho u_{r i} r_i^2 (1 - \cos \theta_o)\)
Introducing Eq. (6) into Eq. (8) and sin $\theta$ is replaced by sin $\theta_0$ in the second integrand for small value of $\delta_r$, the radial core velocity is obtained as:

$$\frac{u_{\infty}}{u_{r-i}} = \left[ 1 + \frac{\delta_r}{8r \cot \theta_0 / 2} \right] \frac{r^2}{r^2_{e}}$$

(9)

Because of the potential flow outside the boundary layer in the $\phi -$ direction, the tangential core velocity can be obtained by the free vortex motion as:

$$u_{\phi-c} \sin \theta = u_{\phi_{i}} r_{i} \sin \theta_0$$

(10)

Inserting Eqs. (6, 7, 9, 10) into Eq. (8) and integrating the result across the boundary layer, pressure is calculated by:

$$p = p_{\delta_{\text{max}}} + \int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \frac{u_{\phi}^2 \cot \theta_0}{r^2_{e}} d \theta$$

if $\delta_{\phi} > \delta_r$ (15a)

$$p = p_{\delta_{\text{max}}} + u_{\phi_{c}}^2 \cot \theta_0 \left[ \theta - \left( \theta_0 - \frac{\delta_{\phi}}{r} \right) \right] \quad \theta_0 - \frac{\delta_{\phi}}{r} < \theta < \theta_0 - \frac{\delta_\phi}{r}$$

if $\delta_{\phi} < \delta_r$ (15b)

$$p = p_{\delta_{\text{max}}} + \frac{u_{\phi_{c}}^2 \cot \theta_0}{r} \left( \delta_{\phi} - \delta_r \right) + \int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \frac{u_{\phi}^2 \cot \theta_0}{r^2_{e}} d \theta$$

$$\theta_0 - \frac{\delta_{\phi}}{r} < \theta < \theta_0$$

if $\delta_{\phi} < \delta_r$ (15c)

where $p_{\delta_{\text{max}}}$ is the pressure at the edge of the boundary layer and can be calculated by the Bernoulli’s equation.

To calculate the boundary layer thickness, $\delta_r$, Eq. (6) is integrated across the $\theta -$ direction as:

If $\delta_{\phi} > \delta_r$

$$\int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \left[ \frac{\dot{c}u_c^2}{c} + \frac{2u_r^2 - u_{\phi}^2}{r} \right] d \theta = \frac{\dot{\theta} u_c}{\rho \varepsilon_r} + \int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \frac{\dot{\theta} p}{\rho \varepsilon_r} d \theta = \frac{\dot{\theta} u_c}{\rho \varepsilon_r} - \frac{\dot{\theta} p}{\rho \varepsilon_r}$$

(16a)

If $\delta_{\phi} < \delta_r$

$$\int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \frac{\dot{c}u_c^2}{c} d \theta = \frac{\dot{\theta} u_c}{\rho \varepsilon_r} + \int_{\theta_0 - \delta_{\phi}/r}^{\delta_{\phi}/r} \frac{2u_r^2 - u_{\phi}^2}{r} d \theta + \int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} \frac{2u_r^2 - u_{\phi}^2}{r} d \theta = \int_{\theta_0 - \delta_{\phi}/r}^{\theta_0} - \frac{\dot{\theta} p}{\rho \varepsilon_r} d \theta = \frac{\dot{\theta} u_c}{\rho \varepsilon_r} - \frac{\dot{\theta} p}{\rho \varepsilon_r}$$

(16b)

The same procedure can be applied to Eq. (7) for calculation the boundary layer thickness $\delta_{\phi}$ as:

If $\delta_{\phi} > \delta_r$

...
\[
\int_{\theta - \delta_{\phi}}^{\theta} \frac{\partial u_{\phi}}{\partial r} d\theta - \frac{u_{\phi}u_{\theta}}{r} \right|_{\theta - \delta_{\phi}/r}^{\theta} + 3 \int_{\theta - \delta_{\phi}/r}^{\theta} \frac{u_{\phi}d\theta}{r} - \frac{\tau_{\theta\phi}}{\rho r} = 0 \quad (17a)
\]

If \(\delta_{\phi} < \delta_{r}\)

\[
\int_{\theta - \delta_{\phi}/r}^{\theta} \frac{\partial u_{\phi}}{\partial r} d\theta + \int_{\theta - \delta_{\phi}/r}^{\theta} \frac{\partial u_{\phi}}{\partial r} d\theta - \frac{u_{\phi}u_{\theta}}{r} \right|_{\theta - \delta_{\phi}/r}^{\theta} + 3 \int_{\theta - \delta_{\phi}/r}^{\theta} \frac{u_{\phi}d\theta}{r} - \frac{\tau_{\theta\phi}}{\rho r} = 0 \quad (17b)
\]

Utilizing the dimensional analysis, the shear velocities are assumed to be proportional to their respective velocity components as:

\[
\tau_{x\phi} = \frac{u_{r}}{\sqrt{u_{r}^{2} + u_{\phi}^{2}}} \quad (18a)
\]

\[
\tau_{r\phi} = \frac{u_{\phi}}{\sqrt{u_{r}^{2} + u_{\phi}^{2}}} \quad (18b)
\]

Also, the shear stress is assumed by the correlation of the Weber’s data [2] as:

\[
\tau = 0.0225 \rho (u_{r}^{2} + u_{\phi}^{2})^{7/2} \left[ \frac{v}{y} \right]^{1/4} \quad (19)
\]

Introducing Eqs. (18, 19) into Eqs. (16, 17) and solving the result simultaneously by the fourth order Runge-Kutta method, one is able to determine the two unknowns variables \(\delta_{r}\) and \(\delta_{\phi}\) under the following the boundary conditions.

\[
\delta_{r}(r_{i}, \theta) = 0, \quad \delta_{\phi}(r_{i}, \theta) = 0 \quad (20)
\]

**IV. Results and Discussion**

Consider an incompressible turbulent viscous flow with a rotational and axial motion in a convergent nozzle with inlet radius \(R_{i} = 6 \text{ mm}\) and outlet radius \(R_{o} = 2 \text{ mm}\), (Fig. 1). The governing boundary layer equations are solved analytically by the integration method. To validate the results, the governing equations have also been solved numerically for nozzle angle \(\theta_{o} = 15^\circ\), Reynolds number \(Re = R_{i}u_{i}/v = 5 \times 10^{4}\) and inlet velocity ratio \(u_{\phi i}/u_{x i} = 0.4\).

The numerical solution is accomplished by finite difference based on finite volume method. The explicit algebraic
Reynolds stress is also applied to model the turbulent flow. The results of these two techniques are shown in Figs. (2, 3). The figures include skin friction coefficient and swirl number along the nozzle respectively. Comparison of two results shows a good agreement between them. The quantity of \( (r - r_o)/(r_o - r) \) on the x-axis indicates the ratio of distance of any point on the nozzle wall to the nozzle length. \( r_i \) and \( r_o \) are respectively the inlet and outlet radii of the nozzle.

Figure (4) illustrates radial velocity boundary layer thickness of a steady state incompressible viscous flow for \( \text{Re} = 5 \times 10^3 \), \( u_{\phi i}/u_{r i} = 2.5 \) and \( \theta_o = 30^\circ \). It is normalized by the radius of the nozzle inlet. The figure includes the results of the present work as well as the results of Najafi el al. [17] by the integration solution. Taylor utilized a uniform profile for the tangential velocity to calculate pressure distribution across the boundary layer as:

\[
p = p_{\text{ex}} + u_{\phi i}^2 \cot \theta_o \left[ \theta - \left( \theta_o - \frac{\delta_{r}}{r} \right) \right]
\]

The results of these two approaches are exactly the same except for a small region located near the peak. The second difference is that Taylor’s solution fails at large inlet velocity ratios while the present work can solve the boundary layer equations under large inlet velocity ratios. The differences originate from more applied centrifugal force in Taylor’s solution.

Fig. (5a) illustrates again the radial velocity boundary layer thickness for nozzle angle \( \theta = 40^\circ \), inlet velocity ratio \( u_{\phi i}/u_{r i} = 1.25 \) and different Reynolds numbers in the range of \( 10^3 < \text{Re} < 5 \times 10^3 \). The boundary layer thickness starts from zero at the nozzle inlet and reaches a maximum value at a distance of 20% of the nozzle length. Then, it decreases gradually towards the nozzle outlet. The reason is that the radial velocity increases proportionally to the inverse of the nozzle radius due to the mass continuity. In a similar manner, Fig (5b) depicts the radial velocity boundary layer thickness but it has been dimensionless by the local nozzle radius. In this case, the first section of the curve remains nearly constant especially for \( \text{Re} > 3 \times 10^3 \) and then goes to zero gradually at the nozzle inlet.

Fig. (6a) reports the radial velocity boundary layer thickness at \( \theta_o = 40^\circ \), \( \text{Re} = 3 \times 10^3 \) and for different inlet velocity ratios in the range of \( u_{\phi i}/u_{r i} < 2.5 \). The ordinate axis has been dimensionless by the radius of the nozzle inlet cross section. It is interesting to note that for \( u_{\phi i}/u_{r i} < 1.65 \), the boundary layer thickness is a linear function of the nozzle length but for \( u_{\phi i}/u_{r i} > 1.65 \) it is a nonlinear function of that. Fig. (6b) is plotted in a similar manner to Fig. (6a).
except for the ordinate axis which has been dimensionless by the local radius of the nozzle cross section. The first part of the curves is flat but after \( u_{\phi} / u_{r_i} > 1.65 \) the boundary layer thickness reaches a minimum value at the middle of the nozzle length. The reason is that the centrifugal force causes the radial boundary layer be fattened.

Fig. (7a) represents the ratio of the tangential velocity boundary layer thickness to the radius of the nozzle inlet cross section for \( Re = 3 \times 10^3 \), \( \theta_o = 40^\circ \) and different inlet velocity ratios in the range of \( 1 \leq u_{\phi} / u_{r_i} \leq 2.5 \). The tangential boundary layer thickness reaches a maximum value at a distance of 80% of the nozzle length from the inlet. In addition, the mean value of the tangential boundary layer thickness in the second part, i.e. from the nozzle inlet up to the maximum point, is smaller than that the first part. It means that the maximum tangential shear stress occurs near to the nozzle inlet. Fig. (7b) is similar to Fig. (7a) except for the normalization parameter which is the local radius of the nozzle cross section. The trend of this figure indicates that the tangential boundary layer thickness is almost a linear function of the nozzle length and does not change significantly with the velocity ratio.

Figure (8) plots the normalized pressure distribution, \( (p - p_o)/(p_o - p_i) \), against the \( r \)-direction for \( Re = 3 \times 10^3 \), \( \theta_o = 40^\circ \) and different velocity ratios in the range of \( u_{\phi} / u_{r_i} \leq 1.65 \). We can see the trend of normalized pressure distribution is different for swirling flow and non-swirling flow significantly. The pressure loss being 63 atm across the nozzle for \( u_{\phi} / u_{r_i} = 1.65 \). Near the outlet region, the rate of change of the pressure loss is much higher than that of the inlet one. Since the nozzle length is very short, hence the contribution of the skin friction is insignificant and the most contribution belongs to the change of momentum forces. On the other hand when the velocity ratio goes to zero, the pressure loss in the nozzle is very small. Consequently, the rate of momentum change is caused by \( r \)-component and becomes very small through the nozzle in comparison with the case that the velocity ratio is large. In this case, the rate of momentum change is caused by both \( r \)- and \( \phi \)-components.

Figure (9) states pressure coefficient, \( C_p = \frac{dp}{dr} D / \rho V^2 \), of the swirling flow under the previous conditions except for the velocity ratio in the range of \( u_{\phi} / u_{r_i} \leq 2.5 \). Since the length of the pressure-swirl atomizer is very short, so the shear force is not important and the pressure gradient should control the process. The pressure coefficient increases with velocities ratio. According to the previous discussion, the frictional force and momentum changes in the \( r \)-direction have small effects on pressure loss. Therefore, the pressure coefficient changes are mainly due to the effect of the rotational motion effects.
Figures (10a-c) predict the deviation of real swirl number to the ideal one against the r-direction for different nozzle angle in the range of \(20^\circ < \theta < 60^\circ\), Reynolds number, \(10^1 < Re < 5 \times 10^3\) and the inlet velocity ratio of \(1 < u_{\phi}/u_r < 2.5\) respectively. The ideal swirl number is defined for potential flow with a uniform radial velocity and free vortex profile for tangential velocity without boundary layer. The swirl number across the nozzle cross section is defined by the ratio of axial component of angular velocity momentum to radial velocity momentum about z-axis as following.

\[
S = \oint \frac{(u_r \cos \theta)u_\theta r \sin \theta dA}{r \sin \theta \int_0^R u_r^2 \cos \theta dA}
\]  

(22)

where \(dA = r^2 \cos \theta \tan \theta / \cos \theta d\theta\).

In Fig. (10), the ratio of real flow swirl number to the potential flow swirl number is nearly one. This small deviation is due to the small contribution of shear stress. The shear stress inversely depends on the minimum boundary layer thickness. According to Fig. (5-7), the radial velocity boundary layer thickness is much less than the tangential one in each cross-section and it decreases with increasing the nozzle angle, Reynolds number and inlet velocities ratio. So, the swirl number deviation increases with increasing these three variables due to the shear stress.

V. Conclusions

A steady incompressible turbulent viscous flow is simulated in a convergent nozzle with axial and tangential velocity components under different conditions. The governing boundary layer equations are solved by integration method. The results show that the radial and tangential boundary layer thicknesses depend on the ratio of \(u_{\phi}/u_r\), Reynolds number and nozzle angle. The peak of the radial and tangential boundary layer thicknesses are located about at \(z/L \approx 0.2\) and \(z/L \approx 0.8\) from the nozzle inlet respectively. It is found that the large ratio of \(u_{\phi}/u_r\) decreases the radial boundary layer thickness due to centrifugal force. The pressure loss in the nozzle is very large and it is caused by the momentum changes. The contribution of the shear stress is not significant on the pressure loss because of the short length of the nozzle.

References


![Fig. (1) Schematic diagram of flow geometry through a convergent nozzle](image)

Fig. (1) Schematic diagram of flow geometry through a convergent nozzle
Fig. (2) Friction coefficient along the nozzle wall

Fig. (3) Swirl number along the nozzle wall

Fig. (4) Comparison of the radial boundary layer thicknesses
Fig. (5) Radial boundary layer thickness for different Reynolds number

Fig. (6) Radial boundary layer thickness for different nozzle inlet velocities ratio
Fig. (7) Tangential boundary layer thickness for different velocities ratio

Fig. (8) Pressure distribution along the nozzle wall
Fig. (9) Pressure coefficient along the nozzle wall
Fig. (10) Swirl number deviation from potential flow along the nozzle