CHAPTER 7

Numerical Simulation of Heat Transfer from Impinging Swirling Jets

B. Sunden¹, J. Larocque² & Z. Wu¹
¹Department of Energy Sciences, Lund University, Sweden.
²Institut National Polytechnique de Grenoble, France.

Abstract

This chapter presents a brief review of studies on swirling and non-swirling impinging jets. Also, a numerical investigation of heat transfer from impinging swirling jets is described. Various turbulence models are used. Comparison with experimental data is provided. Generally, the V2f model was found to perform better than other models, but for small nozzle-to-plate distances, it was hard to maintain good or sufficient accuracy. The difference in performance, in terms of the Nusselt number distribution on the impingement wall, between swirling and non-swirling impinging jets was clearly identified. The magnitude of the swirling motion has great influence on the flow field and the formation of recirculating zones and accordingly the heat transfer process.

Keywords: Swirling jet, heat transfer, CFD, turbulence modeling

1 Introduction

Impinging jets occur in many engineering applications of cooling, e.g. gas turbine combustors and turbine blades. Swirling jets appear in combustors to promote flame stabilization by enhanced mixing of fuel and oxidant [1, 2]. By adding swirl to a jet, mixing properties are enhanced and the swirling motion is beneficial for controlling jet flows in gas turbines, diesel engines or flames in combustion chambers when enhanced dilution and rapid mixing is desired. Usually, the result is a reduction in the flame length and a stabilization of the flame. Swirling impinging jets have been studied to a less extent, whereas swirling flow, in general, has been considered extensively, see, e.g. [3–5].
Impinging jets have been widely used for heat transfer augmentation in a variety of engineering applications such as cooling of hot steel plates, drying of papers and films, cooling of turbines blades, combustor walls and electronic components. Figure 1 introduces the necessary vocabulary used for a free impinging jet. The impinging jet can be divided into several regions (i.e. the free jet, the stagnation impingement zone, the thin boundary layer along the wall, i.e. the wall jet) in which one can use the appropriate system of equations in the analysis. In an effort to increase the mixing in such flows, swirling impinging jets might be considered. Actually, the main source of the turbulence in swirling flows is the additional shear of the jet. The additional tangential velocity component of the shear layer increases the spreading and the mass entrainment. The mixing is substantially increased both by the large coherent structures of the jet and by a higher level of turbulence.

The swirling velocity component may have a large influence on the flow. The swirling motion causes a centrifugal force that is directed away from the centerline. Hence, close to the centerline an area of low pressure arises, as the fluid is forced to move away from the centerline. The intensity or strength of the swirl becomes weaker with larger distance downstream of the nozzle. Therefore, the pressure drop becomes smaller and the axial variation of the pressure increases. The combination of radial and axial pressure gradients has a direct influence on the flow field.

It has been observed (e.g. [4, 5]) that even at low amount of swirl, the spreading rate of a swirling jet is higher than that of the non-swirling counterpart. The mechanism of jet spread in the non-swirling case is dominated by coherent structures and the turbulent mixing at the interface of the jet and the ambient fluid. In the case of swirling jets, there is an additional component coming from the centrifugal forces, which are acting in the sense of increasing the spreading rate of the jet. With increasing spreading rate, the entrainment increases as well. The addition of

Figure 1: Regions in an impinging jet.
swirl to the jet increases substantially the turbulence levels and, therefore, decreases the potential core length. Besides, the swirling effect diminishes as the nozzle-to-plate distance increases. The results by Rodriguez and El-Genk [6] show that the swirling impinging jet flow field transfers to that of a non-swirling jet beyond a few jet diameters, which is consistent with the recent similarity theory developed by Semaan et al. [7].

Various numerical simulations of non-swirling impinging jets have been studied in the literature [8]. Jet impingement is often used as a benchmarking flow for improving turbulence models due to the following difficulties associated with flow patterns in impinging jets, as stated in Hofmann et al. [9]:

1) Entrainment of fluid from the environment and prediction of the jet spreading angle.
2) Relaminarization near the stagnation point.
3) Large acceleration of the flow, followed by a deceleration.
4) Laminar-turbulent transition in the wall jet.
5) Different curve characteristics of radial heat transfer evolution, depending on flow velocities and nozzle-to-plate distances.

A comparison of different numerical simulation techniques for impinging jet flow and heat transfer is provided in Table 1 based on the work by Zuckerman and Lior [10]. Even with high-resolution grids, the various implementations of the $k-\epsilon$, $k-\omega$, algebraic stress model and Reynolds stress model (RSM) present large errors compared with experimental results, whereas the shear-stress transport (SST) and V2f models have better predictive abilities for impinging jet and are recommended as the best compromise between computational cost and accuracy, as shown in Table 1. The long computational time of the direct numerical simulation (DNS) limits its application to low Reynolds numbers. However, for laminar impingement, the DNS approach offers little improvement in accuracy over other numerical models or analytic solutions. Compared with DNS, large eddy simulation (LES) can be applied for relatively high Reynolds numbers, e.g. the non-swirling and swirling impinging jets with Reynolds numbers of 21,000 and 23,000 and four different swirl numbers were investigated via LES by Uddin et al. [11]. Numerical investigations on swirling impinging jets are relatively scarce, e.g. [12–14] for laminar flow and [6, 11, 15–17] for transition and turbulent flow, mainly focusing on the flow field. The present work will further evaluate the performance of various turbulence models for swirling impinging jets in terms of the heat transfer characteristics on the impingement wall. Recently, Brown and coworkers [18–22] experimentally investigated the effect of swirl on the heat transfer characteristics of jet impingement cooling and showed both enhanced and diminished performance.

In this investigation, the swirling jets are circular jets with a circumferential or spiraling motion around the main axial flow direction. Usually, a rotating motion is induced upstream at the nozzle exit to provide the swirl. First, a brief presentation of a swirling impinging jet and a description of physical phenomena occurring in this kind of jet are given. Then, a numerical approach based on computational fluid dynamics (CFD) simulations is presented. Some turbulence models are
applied. Numerical tests have been carried out to understand and verify the physics of swirling jets, and comparison with available experimental data is carried out to check the accuracy and capabilities of the computational method in prediction of the heat transfer and flow characteristics of swirling impinging jets.

The level of swirl is usually characterized by the dimensionless swirl number $S$. There are several ways of defining the swirl number. Here, we present two:

1) The ratio of the axial flux of angular momentum $G\theta$ and axial flux of axial momentum $G$, normalized by the jet radius. This is currently the way $S$ is defined commonly in the literature

\[
S = \frac{G\theta}{RG} = \frac{2\pi \int_0^\infty \rho U_{ax} U_{tan} r^2 dr}{R 2\pi \int_0^\infty \rho U_{ax}^2 r dr}
\] (1)

Table 1: Comparison of common CFD turbulence models used for impinging jets [10].

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Computational cost (computation time required)</th>
<th>Heat transfer coefficient prediction</th>
<th>Ability to predict secondary peaks in Nu</th>
</tr>
</thead>
<tbody>
<tr>
<td>$k-e$</td>
<td>Low cost</td>
<td>Poor: Nu errors of 15%–60%</td>
<td>Poor</td>
</tr>
<tr>
<td>$k-\omega$</td>
<td>Low–moderate</td>
<td>Poor–fair: Nu errors of at least 10%–30%</td>
<td>Fair: may have incorrect location or magnitude</td>
</tr>
<tr>
<td>Realizable $k-e$ and other $k-e$ variations</td>
<td>Low</td>
<td>Poor–fair: Nu errors of at least 15%–30%</td>
<td>Poor–fair: may have incorrect location or magnitude</td>
</tr>
<tr>
<td>Algebraic stress model</td>
<td>Low</td>
<td>Poor–fair: Nu errors of at least 10%–30%</td>
<td>Poor</td>
</tr>
<tr>
<td>Reynolds stress model (full SMC)</td>
<td>Moderate–high</td>
<td>Poor: Nu errors of 25%–100%</td>
<td>Fair: may have incorrect location or magnitude</td>
</tr>
<tr>
<td>SST</td>
<td>Low–moderate</td>
<td>Good: typical Nu errors of 20%–40%</td>
<td>Fair</td>
</tr>
<tr>
<td>V2f</td>
<td>Moderate</td>
<td>Excellent: Nu errors of 2%–30%</td>
<td>Excellent</td>
</tr>
<tr>
<td>DNS/LES time-variant models</td>
<td>Extremely high</td>
<td>Good–excellent</td>
<td>Good–excellent</td>
</tr>
</tbody>
</table>
If the upper limit of the integration is set to R instead of infinity and if constant parameters (volume mass, axial and tangential velocity) are assumed, a simple linear relation between $S$ and $U_{\tan}$ exists, namely

$$S = \frac{2}{3} \frac{U_{\tan}}{U_{ax}}$$

(2)

2) From a technical point of view, in cases where swirl is generated by guide vanes, the swirl number can more conveniently be expressed by the vane angle, i.e.

$$S = a = \arctan \left( \frac{U_{\tan}}{U_{ax}} \right)$$

(3)

It has been shown experimentally that the behavior of swirling flows depends on the method with which the swirl is generated. This dependence is due to the different axial and tangential velocity profile distributions associated with the different swirl generation methods. Even if the axial flux and swirl numbers are the same, different configurations of the flow field could be obtained by changing the inlet velocity profiles. To find out the influence of the swirl velocity profile on the wall heat transfer, four different profiles for the tangential velocity components are considered, see Table 2, based on [23]. Of course, a free vortex could also be included but it was deemed sufficient to consider those in Table 2.

### 2 Governing Equations

Although in swirling jets three directions of motion ($x$, $r$ and $\theta$) are important, axisymmetry is assumed for all circular jets, even for a jet with swirl. This is of course an important limitation, as in practice many swirling jets are not axi-symmetric. However, in case of axi-symmetry, a description in two coordinates $x$ and $r$ is possible, which makes a numerical simulation much simpler. Note, however, that the third velocity component is also calculated. With this approach, the global flow

---

**Table 2: Different profiles for the tangential velocity.**

<table>
<thead>
<tr>
<th>Profile</th>
<th>$U_{\tan}$</th>
<th>Constant</th>
</tr>
</thead>
<tbody>
<tr>
<td>Constant angle</td>
<td>$C_1$</td>
<td>$C_1 = U_{\tan}$</td>
</tr>
<tr>
<td>Forced vortex</td>
<td>$C_2 r$</td>
<td>$C_2 = \frac{4}{3} \frac{U_{\tan}}{R_0}$</td>
</tr>
<tr>
<td>Rotating tube</td>
<td>$C_3 r^2$</td>
<td>$C_3 = \frac{5}{3} \frac{U_{\tan}}{R_0}$</td>
</tr>
<tr>
<td>Rankine vortex</td>
<td>$\frac{C_4}{r} \left( 1 - \exp \left( -\frac{r^2}{R_0^2} \right) \right)$</td>
<td>$C_4 = 1.812 U_{\tan} R_0$</td>
</tr>
</tbody>
</table>
behavior like spreading, velocity decay and entrainment, as a function of the swirl velocity can be investigated.

The flow is also assumed to be incompressible due to low Mach number and the buoyancy forces due to gravity are neglected. Steady state is considered. With these assumptions one has:

$$\frac{\partial}{\partial t} = 0 \text{ and } \frac{\partial}{\partial \theta} = 0$$  \hspace{1cm} (4)

The governing equations are written in a general form in cylindrical coordinates (see Fig. 2) based on the study by Facciolo [24]. The radial, azimuthal and axial directions are denoted by \((r, \theta, x)\) and the corresponding velocity components are \((w, v, u)\), respectively. It is assumed that the rotation is along the axial direction and hence the rotation vector can be written as \(\vec{\Omega} = \Omega e_x\).

With these assumptions, the conservation equation of mass (continuity equation) becomes

$$\frac{\partial w}{\partial r} + \frac{w}{r} + \frac{\partial u}{\partial x} = 0$$  \hspace{1cm} (5)

The conservation of momentum (Navier–Stokes equations) can be written as

$$w \frac{\partial w}{\partial r} + u \frac{\partial w}{\partial x} - \frac{v^2}{r} = -\frac{1}{\rho} \frac{\partial p}{\partial r} + \nu \left( \frac{\partial^2 w}{\partial r^2} + \frac{1}{r} \frac{\partial w}{\partial r} - \frac{w}{r^2} + \frac{\partial^2 w}{\partial x^2} \right) - 2\Omega v$$  \hspace{1cm} (6)

$$w \frac{\partial v}{\partial r} + u \frac{\partial v}{\partial x} + \frac{vw}{r} = \nu \left( \frac{\partial^2 v}{\partial r^2} + \frac{1}{r} \frac{\partial v}{\partial r} - \frac{v}{r^2} + \frac{\partial^2 v}{\partial x^2} \right) + 2\Omega w$$  \hspace{1cm} (7)

$$w \frac{\partial u}{\partial r} + u \frac{\partial u}{\partial x} = -\frac{1}{\rho} \frac{\partial p}{\partial x} + \nu \left( \frac{\partial^2 u}{\partial r^2} + \frac{1}{r} \frac{\partial u}{\partial r} + \frac{\partial^2 u}{\partial x^2} \right)$$  \hspace{1cm} (8)
The appearance of several terms due to the rotation effects is evident:

- The term \(-\frac{\nu^2}{r}\) is the centrifugal force, which gives the effective force in the \(r\)-direction resulting from fluid motion in \(\theta\)-direction.
- The term \(\pm \frac{vw}{r}\) is the Coriolis force, which is the effective force in the \(\theta\)-direction when there is a flow in both the \(r\) and \(\theta\)-directions.
- The Coriolis terms are \(-2\Omega v\) and \(+2\Omega w\).

The energy equation is given as follows:

\[
\rho c_p \left( \frac{w}{r} \frac{\partial T}{\partial r} + u \frac{\partial T}{\partial x} \right) = \lambda \left[ \frac{1}{r} \frac{\partial}{\partial r} \left( r \frac{\partial T}{\partial r} \right) + \frac{\partial^2 T}{\partial x^2} \right] 
\]  

(9)

3 Numerical Approach

In this case, the finite volume method has been used to solve the governing equations. Four different turbulence models are applied, see below. Non-uniform grids are used and the grid arrangement as well as the number of grid points has been varied to make sure that decent numerical results are obtained. The SIMPLE algorithm has been chosen to handle the pressure–velocity coupling, and the PRESTO algorithm has been applied for discretization of the pressure field. The commercial computer code FLUENT was adopted as the solver in this work.

3.1 Turbulence and turbulence modeling

Most swirling flows are turbulent and it is then common to use the Reynolds’ decomposition to obtain the equations for the mean flow and temperature fields. Then, some models to describe the turbulence field is needed. In this work, some turbulence models are applied. These are briefly described here. Equations are omitted but these can be found elsewhere, e.g. in references [25–27].

3.1.1 The RNG \(k-\epsilon\) model

Compared with the standard \(k-\epsilon\) model, the RNG \(k-\epsilon\) model effectively has some differences in treating the coefficients in the turbulent dissipation equation. The RNG procedure systematically removes the small scales of motion from the governing equations by expressing their effects in terms of larger scale motions and a modified viscosity. The model contains a strain-dependent correction term in a coefficient in the turbulent dissipation equation. In addition, an option is available in FLUENT to modify the turbulent viscosity properly to improve the calculation of swirling flows. This means that the turbulent viscosity for non-swirling flows is multiplied by a function which depends on the swirl number and a swirl coefficient.
3.1.2 The SST $k-\omega$ model
The SST $k-\omega$ model has been developed to effectively blend the robust and accurate formulation of the $k-\omega$ model in the near wall region with the free-stream independence of the $k-e$ model in the far field. No wall functions are used in conjunction with this model.

3.1.3 The RSM model
The two models above assume that the turbulent viscosity is isotropic, or in other words that the ratio between the Reynolds stress tensor and mean rate of deformation tensor is the same in all directions. This assumption is known to fail in many categories of flow and inaccurate flow predictions occur. A remedy to this can be to solve transport equations for the Reynolds stresses themselves instead, i.e. to apply the so-called RSM models. Such models are based on six transport equations, one for each Reynolds stress plus an equation for the turbulent dissipation. An RSM model has also been considered in this work.

3.1.4 The V2f model
Another more recent model has also been applied. The so-called V2f model [28, 29] is a four-equation model based on transport equations for the turbulence kinetic energy ($k$), its dissipation rate ($\varepsilon$), a velocity variance scale ($\overline{v^2}$) and an elliptic relaxation function ($f$).

The main feature of the V2f model is its use of the velocity scale $\overline{v^2}$ instead of the turbulent kinetic energy $k$ for evaluating the eddy viscosity. This model provides also an evaluation of a time scale required in the calculation of the turbulent viscosity. This model does not require the use of wall functions because the V2f model is valid down to a wall. Actually, the near wall treatment is included in the turbulence model. Some investigations have shown that this model provides very good performance for impinging jets.

3.1.5 Near wall treatment
Turbulent flows are significantly affected by the presence of walls. The near wall modeling highly impacts the fidelity of numerical solutions. In general, two methods exist namely, the wall function approach and the low Reynolds number approach. For the RNG $k-\varepsilon$ model and the RSM model, wall functions are used in this case but not for the SST $k-\omega$ model and the V2f model.

4 Results and Discussion

4.1 Basic test case – reveal of physical influence of swirl
As a first calculation test case, the physical influence of swirl on the heat transfer distribution on an impingement wall was considered. Figure 3 shows the geometry selected for a laminar case, which is identical to that of Shuja et al. [12] studied numerically. However, Shuja et al. mainly considered the flow field and not the wall heat transfer.
In Fig. 3, the swirling jet impinges on a solid surface (Wall 2). The following flow data were used: $U_{ax} = 0.03$ m/s ($Re = 48$), $U_{swirl} = 0–0.03$ m/s. The nozzle radius is: $R_o = 0.0127$ m.

Figure 4 shows some results for the local Nusselt number for various strengths of the swirling motion. The region with the highest $Nu_D$ moves radially and the heat transfer is suppressed near the stagnation point with increasing swirl intensity. The overall observation is that the swirling motion has no enhancement effect for this case. The flow fields calculated in this work agreed with those of Shuja et al. for those cases where direct comparisons could be carried out.
4.2 Turbulent swirling impinging jet

Figure 5 shows the geometry for this case. In this case, the turbulence model was an RSM to handle anisotropic effects, and the wall function approach with non-equilibrium wall function was employed. The pressure–velocity coupling was handled by the SIMPLE algorithm and the PRESTO scheme was applied to discretize the pressure. The jet velocity at the nozzle exit is 30 m/s corresponding to a Reynolds number $Re = 19,000$.

In Fig. 6, the impact of the swirling motion on the local Nusselt number distributions is depicted. The higher Nusselt number region moves radially and heat transfer is suppressed near the stagnation point with increasing swirl intensity. For $S = 0$ or 5, the shape of the Nu distribution in the region $0 < r < 0.006$ m was almost uniform. The results for $S = 0$ correspond well with the results for other similar geometries of impinging jets, see, e.g. [30, 31]. In general, there also exist spread in such experimental data but the physical behavior is consistent. In Fig. 6, one observes that the distribution shows a peak (saddle shape) at $r \sim 0.004$ m for $S = 10$. For all conditions of swirl intensity, Nu decreases monotonically in the radial direction, for fluids being heated by the surface flowing outward and a thermal boundary layer develops on the surface in the outer region. From these results, it is clear that the stretching of the radial width of the jet is linked with the radial distance of the maximum local heat transfer coefficient. The stretching of the radial width of the jet, which leads to a decrease in the axial velocity, is the predominant factor in the formation of the characteristic radial heat transfer distribution of a swirling impinging jet. The region where the heat transfer characteristics of a swirling impinging jet is different from those of an axial impinging jet is at approximately $r/d < 1$. Instantaneous velocity and temperature fields (not shown) in the stagnation region are subjected to the recirculation zone, and

![Figure 5: A geometry of a turbulent swirling jet. Left-hand vertical line is the impingement wall ($Re = 19,000$).](image-url)
The behavior of this zone is changed by the swirl intensity. For $S = 10$, stationary recirculation zones are formed. Fluid flows after impinging into the stagnation region along the impinging plate and heat transfer near the stagnation point are suppressed. For $S = 5$, it is not possible to observe a recirculation zone. Increasing swirl intensity brings about enhancement of radial momentum transport, and adversely reduces axial momentum transport. The presented results confirm the findings by Nozaki et al. [32] that for a swirling impinging jet, the heat transfer process is divided into two modes, i.e. a suppression mode and an enhancement mode. Overall, the swirl effect is similar for laminar and turbulent cases as revealed by comparing Figs 4 and 6.

### 4.3 Calculations vs. experiments by Bilen et al. [33]

The geometry is similar to that presented in Fig. 5 but the following experimental set up data were used by Bilen et al.: $Re = 20,000$ and 40,000 (Reynolds number based on the bulk velocity at the nozzle exit), $D = 0.015$ m (diameter of the nozzle), $H/D = 8$ (impingement distance from the heated plate to the nozzle exit), $a = 0^\circ$, 22.5$^\circ$ and 50$^\circ$ (corresponding to swirl numbers 0, 0.3, and 0.89). In the numerical calculations, various profiles for the axial and swirl velocity were used. It was found that a uniform value of the axial velocity profile at the nozzle exit was most proper. For this case (large H/D), it was found that the swirl profile had a minor influence. Further details can be found in Larocque [34].

Figure 7 shows the results for the local Nusselt number distribution at $S = 0.3$. Four turbulence models were used. For weak swirl, the SST models still provides
good performance, whereas the RSM and RNG overpredict the heat transfer in the stagnation point region. One can also observe a decrease in the Nusselt number at the stagnation point compared with the case without swirl. The V2f model overpredicts the heat transfer at the stagnation point but is closer to experimental data for $r > 0.025$ m than the RSM or RNG models. Overall, it seems that the V2f and the SST models perform reasonably well. This might be an indication that the wall function approaches for RSM and RNG are not appropriate.

Figure 8 shows the corresponding results for the local Nusselt number at $S = 0.89$. For high swirl, all models predict the saddle shape for the radial Nusselt number distribution and one can observe that the experimental data are between the overprediction by the RSM and RNG models and the underprediction of the SST model. Consequently, one may deduce that the physical phenomena like heat transfer suppression in the stagnation point and the displacement of the maximum Nusselt number location are captured by the numerical models. The V2f model gives a very good prediction for high swirl. This is the most accurate model for this case.

4.4 Calculations vs. experiments by Huang and El-Genk [35]

Again, the considered geometry is similar to that one in Fig. 5. The experimental data for this case were taken for: $Re = 17,600$ (Reynolds number based on the
bulk velocity at the nozzle exit), $D = 0.0127$ m (diameter of the nozzle), $H/D = 1$ (impingement distance from the heated plate to the nozzle exit), $\alpha = 15^\circ$ and $30^\circ$ (the swirl angle).

Figure 9 depicts the local Nusselt numbers vs. radial distance for this case. When the impingement distance is small ($H/D = 1$), the RSM and RNG models predict an overly suppress around the stagnation point compared with the experimental data. The SST model still underpredicts the experimental data. One can also observe that the difference in the results between a swirl angle of $15^\circ$ or $30^\circ$ is indeed small. In this case, it was found that the swirl profile has some influence on the results. A non-constant profile for swirl velocity cancels the heat transfer suppression at the stagnation point and provides better prediction in this region. The V2f model gives a high underprediction and seems not to be accurate for this case of a small impingement distance ($H/D = 1$). The influence of the swirl profile (Table 2, further details can be found in [34]) might be due to differences in turbulence production provided by the velocity vector of a constant swirl profile and that of, e.g. a Rankine vortex. Further studies are needed to clarify this. For all cases presented, there are also several interesting results on the local flow structure, but in this chapter these are not presented.
5 Conclusions

A numerical investigation of heat transfer and fluid flow for swirling impinging jets was presented. A literature review was also provided.

The behavior of the recirculation zone in swirling impinging jets plays a major role in the heat transfer mechanism, and this transfer is highly sensitive to the swirl intensity. The heat transfer at the stagnation point is generally decreased and a maximum Nusselt number occurs at a certain distance from the jet centerline.

It was found that although the V2f model seemed to be inaccurate for small impinging distance, it provided good performance for large impinging distances.

For a small impinging distance, the swirl velocity profile was found to be important and had an influence on the flow and especially on the formation of the recirculation zone.

Acknowledgement

Financial support from the Swedish Energy Agency (STEM) is gratefully acknowledged.
Nomenclature

\begin{itemize}
\item $C_1, C_2$ coefficients
\item $C_3, C_4$ coefficients
\item $D, d$ jet diameter
\item $e$ unit vector
\item $f$ relaxation function
\item $G$ momentum
\item $h, H$ distance from jet nozzle to plate
\item $k$ turbulent kinetic energy
\item $Nu$ Nusselt number
\item $p$ pressure
\item $R, R_0$ radius of the jet
\item $Re$ Reynolds number
\item $r$ radial coordinate
\item $S$ swirl number
\item $T$ temperature
\item $t$ time
\item $U$ axial approach velocity
\item $u$ velocity component (axial)
\item $v$ velocity component (azimuthal)
\item $w$ velocity component (radial)
\item $x$ coordinate
\item $y$ coordinate
\item $z$ axial direction
\end{itemize}

Greek symbols

\begin{itemize}
\item $\alpha$ swirl angle
\item $\varepsilon$ dissipation of turbulent kinetic energy
\item $\theta$ azimuthal coordinate direction
\item $\lambda$ thermal conductivity
\item $\nu$ kinematic viscosity
\item $\omega$ turbulent vorticity
\item $\Omega$ rotation magnitude
\item $\rho$ density
\end{itemize}

Subscripts

\begin{itemize}
\item \textit{ax} axial direction
\item \textit{tan} tangential direction
\item \textit{\theta} azimuthal direction
\end{itemize}
References


