Proceedings of the 10th Brazilian Congress of Thermal Sciences and Engineering -- ENCIT 2004
Braz. Soc. of Mechanical Sciences and Engineering -- ABCM, Rio de Janeiro, Brazil, Nov. 29 -- Dec. 03, 2004

Paper CIT04-0XXX

Turbulence Modelling in Hydrocyclone Flow

Capela Moraes, C.A.
PETROBRAS Research Center.

Atila P. Silva Freire
Mechanical Engineering Program (PEM/COPPE/UFRJ)
C.P. 68503, 21945-970, Rio de Janeiro, Brazil
atila@mecanica.coppe.ufrj.br

Abstract. Flow in a hydrocyclone cannot be suitably described using isotropic turbulence models since streamline curvature and rotation of fluid particles impose anisotropy to turbulence.
Main fluid dynamic phenomena in hydrocyclone flow are determined by inertia forces due to radial equilibrium between pressure and centrifugal forces and can be essentially approached through an ideal flow modeling concept. Further improvement in modeling this flow implies consideration of viscosity and turbulence. Models using the concept of turbulent viscosity tend to be too diffusive preventing the capture of the so called cyclonic effect (increasing of rotation velocity for smaller distances from the axis). This feature can be captured using more robust models known as Complete Reynolds Stress Models which require seven additional differential equations to solve the problem.
This paper presents a modified k-ε turbulence model that can be adjusted to reproduce the performance of the RSM in describing velocity profiles in the hydrocyclone operating without an air core. Using only two additional equations this model requires roughly one third of the computational resources required by RSM.

Keywords. *Turbulence, hydrocyclone,* ***modified k-ε turbulence model***

1. Introduction

The problem of a turbulent jet impinging orthogonally onto a surface has been dealt with by the present authors in two recent publications (Guerra e Silva Freire (2003, 2004)). Those publications analyzed both the velocity and the temperature fields with an emphasis on a description of the inner layers of the flow. At the time, both works specifically analyzed the existence of the so called universal law of the wall, in view of its relevance for the calculation of the wall shear stress and of the wall heat transfer. For wall jets, we cannot positively say that the log-law is a well established concept. In fact, several authors (Patel (1962), Tailland and Mathieu (1967), Ozarapoglu (1973), Irwin (1973)) have reported a large range for values for the log-law constants. This certainly raises some important questions as to the validity on the use of the log-law for the estimation of surface friction and the heat transfer coefficient.

Almost at the same time however, other researchers (Özdemir and Whitelaw (1992)) showed that for an oblique impinging jet the law of the wall could be observed for both the velocity and the temperature fields. More than that, these authors proposed a functional behavior for the log-law parameters that resorted to a scaling procedure based on the stream-wise evolution of the flow by the maximum jet velocity. In fact, Narasimha et al. (1973) were the first to acknowledge that the traditional use of the nozzle diameter as the reference scaling for wall jet flows was not appropriate. They proposed a scaling length that would take into consideration the flow evolution.

The purpose of the present work is to carry out further investigations on scaling laws governing the motion of a orthogonal jet impinging onto a surface. Here, for the first time, we will present data for the longitudinal turbulent intensities. The law of the wall, for both the velocity and the temperature fields, will also be investigated under the light of some new data.

Thus, at this point, it important to make it clear to the reader that other authors have specifically studied the role of the scaling laws in wall jet flows. That is the case of the work of Wygnanski et al. (1992) where the relevance of the wall to the evolution of the large coherent structures in the flow was studied. Here, in the impinging jet, the problem is further complicated by a deflection of the streamlines and by the presence of a stagnation point.

A turbulent jet impinging orthogonally onto a surface is a geometrical arrangement commonly used in industry to promote high rates of heat exchange. The studies have concentrated on the investigation of different features of the phenomenon because of the several important aspects associated to the problem. These studies normally solve for the velocity and the temperature fields in regions around but not at the stagnation point.

In fact, a question that has been the object of many investigations is the behavior of the heat transfer coefficient at the stagnation point. Cases where the Reynolds number is low enough so that the flow can be rendered laminar, asymptotic methods can be used to find analytical solutions in all flow regions except near the stagnation point, which presents a strong singularity. Consequently, even for this simple flow condition, calculation of the heat transfer coefficient at the stagnation point is very difficult. The result is that a severe lack of information on the flow behaviour in the stagnation region exists. The reason for this is clear, due to the small scales that define this region, the placement of dedicated instrumentation is always very difficult.

The flow structure of an impinging jet produced by a nozzle can be highly complex due to the ambient fluid entrainment, flow separation, interaction of the flow with the impingement or confining walls, and generation of vortices. In this work, we will provide experimental data on turbulent semi-confined and unconfined impinging jets.. Once understood the flow structure, one can foresee how this dynamics affects the heat transfer process. Results are presented for the turbulent characteristics of a round jet. The work includes measurements for the radial mean and instantaneous velocity profiles and pressure distributions. The velocity field was measured through a hot-wire system.

In regard to the behavior of the heat transfer coefficient at the stagnation point (Lee and Lee (1999,2000), Kendoush (1998), Nishino et al. (1996)). If the Reynolds number is low enough so that the flow can be rendered laminar, then asymptotic methods can be used to find analytical solutions in all flow regions but near the stagnation point, where a strong singularity is present. Thus, even for this simple flow condition, calculation of the heat transfer coefficient at the stagnation point is very difficult to achieve.

For turbulent flows, the correct description of the flow field is greatly complicated by the necessary specification of turbulence models that can capture all relevant characteristics of the problem. Frequently, turbulence models of the eddy viscosity type are used together with some heat transfer analogy consideration for the description of the temperature field (see, e.g., Behnia et al. (1998, 1999), Gibson e Harper (1997)). This leads to a serious difficulty at the stagnation point where the Reynolds analogy between eddy-diffusivity and eddy-viscosity breaks down. Indeed, when the equations of motion are integrated to the wall and the hypothesis of a constant turbulent Prandtl number is used, the calculated heat transfer rates at the stagnation point are observed to exceed by much the actual values.

Despite the critics of many researchers, the use of wall functions to by-pass the difficulties involved with the modeling of low Reynolds number turbulence is still an attractive means to solve problems in a simple way. Cruz and Silva Freire (1998) have proposed an alternative approach where new wall functions are used to describe the velocity and temperature fields in the wall logarithmic region. As the stagnation point is approached, these functions reduce to power-law solutions recovering Stratford's solution. The paper of Cruz and Silva Freire resorted to Kaplun limits for an asymptotic representation of the velocity and temperature fields. Results were presented for the asymptotic structure of the flow and for the skin-friction coefficient and Stanton number at the wall.

$$\tilde{U}_i = U_i + u_i \qquad \textbf{(1-1)}$$

Navier-Stokes equations are as follows:

$$\frac{\partial \tilde{U}_i}{\partial x_i} = 0 \qquad \textbf{(1-2)}$$

$$\frac{\partial \tilde{U}_i}{\partial t} + \tilde{U}_j \frac{\partial \tilde{U}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \tilde{P}}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial \tilde{U}_i}{\partial x_j}\right) + \tilde{F}_i \qquad \textbf{(1-3)}$$

taking the time average of these equations we obtain the so called Reynolds equations:

$$\frac{\partial U_i}{\partial x_i} = 0 \qquad \textbf{(1-4)}$$

$$U_j \frac{\partial U_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu \frac{\partial U_i}{\partial x_j} - \overline{u_i u_j}\right) + F_i \qquad \textbf{(1-5)}$$

where P is the averaged pressure field and Fi is the averaged body force vector field. The capital letters and the overbar on the correlation of fluctuating velocities mean that these quantities are time averaged values.

Since these equations are only four and there are ten unknowns (P, U_i, $\overline{u_i u_j}$) – remembering that the Reynolds tensor $\overline{u_i u_j}$ is symmetrical – it is necessary to model the six elements of the Reynolds or turbulent tensor.

Trying to get a transport equation for the $\overline{u_i u_j}$ through algebraic manipulations and averaging process results in other unknown terms which have higher order correlations of fluctuating velocities. These new unknowns also require additional equations that will again introduce new higher order correlations terms in a endless process that is known as the closure problem of Reynolds equations for turbulent flow. To brake up this process and get a closed system of equations for turbulent flow it is necessary to model the unknown terms.

The goal of the turbulence models is to provide additional equations which together with equations (1-4) and (1-5) form a closed system of equations for turbulent flow modelling.

1.2. Turbulent Viscosity Concept

The simplest approach to model turbulent stresses that appear in equation (1-5) was suggested by Boussinesq who, for some specific kinds of flow, postulate a relationship between the Reynolds stresses and the local deformation rate of the mean flow similar to that observed between the stress tensor and the deformation rate in a laminar flow of a Newtonian fluid. The expression below, proposed by Komolgorov, is a generalisation of the Boussinesq hypothesis:

$$-\overline{u_i u_j} = -\frac{2}{3} k \delta_{ij} + \nu_T \left(\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \quad \textbf{(1-6)}$$

where $k = \frac{1}{2}\left(\overline{u_1^{\,2}} + \overline{u_2^{\,2}} + \overline{u_3^{\,2}} \right)$ is the kinetic energy of the turbulence.

The parameter νT is called turbulent viscosity and is a property of the flow field whereas the molecular viscosity is a fluid property.

The expression (1-6) implies that the main axis of the turbulent tensor have the same direction of the main axis of the mean flow deformation tensor which can only be a reasonable proposal if the turbulence is isotropic.

In the flow in hydrocyclones the spiralled stream lines (highly curved) imply a high degree of anisotropy. In such flow conditions, it is expected that the deformation rate of the mean flow has different values for azimuthal, radial and axial directions. The same can be said on the turbulence fluctuations themselves. Bradshaw [1973] has shown that for shear flow on a concave surface, in the direction of the main flow, if it is kept the turbulent viscosity concept, the model must be of the form:

$$\overline{uv} = \nu_T \left(\frac{\partial U}{\partial y} + \alpha \frac{\partial V}{\partial x} \right) \quad \textbf{(1-7)}$$

where the value of the factor α is around 10. Hoffmann at al. [1985] and Launder at al. [1977] analysed turbulent flow on concave surfaces and realised great differences comparing to the flow on plane surfaces and even to the flow on convex surfaces. On concave surfaces, the turbulence is directly modified by the curvature of the streamlines and indirectly by the appearance longitudinal vortices on the surface (Gortlër vortices) (see Schlichting [1968]).

Despite these limitations flow models based on the turbulent viscosity concept have been largely employed, with some adaptations tailored for the specific application, even in cases of complex flows, with reasonable success.

1.3 Two equation k-ε turbulence model and its problems in describing hydrocyclone flow

This model is one of the most used in engineering applications and it uses two transport differential equations one for the turbulent kinetic energy $k = \frac{1}{2}\left(\overline{u_i u_i}\right)$ and another one for the rate of turbulent kinetic energy dissipation ε.

The exact expression for the turbulent kinetic energy can be obtained starting from instantaneous momentum equations written in terms of decomposed velocity (equation 1-1), and multiplying each component of these equations by its corresponding velocity fluctuation (u_i). The equations obtained should be added and the resultant equation should be averaged to give the total kinetic energy expression. From this expression it should be subtracted the mean flow kinetic energy. This final expression represents the kinetic energy of the turbulence fluctuations.

The turbulent kinetic energy equation can be written as follows:

$$\frac{\partial k}{\partial t} + C_k = D_k + P_k + \varepsilon \quad \textbf{(1-8)}$$

Where C, D and P stand for turbulent energy convection, diffusion and production terms, respectively, and ε is the dissipation of the turbulent energy. The expressions for these terms are shown below:

$$C_k = U_j \frac{\partial k}{\partial x_j} \tag{1-9}$$

$$D_k = -\frac{\partial}{\partial x_j}\left[\overline{u_j\left(\frac{u_i u_i}{2}+\frac{p}{\rho}\right)} - \nu \frac{\partial k}{\partial x_j}\right] \tag{1-10}$$

$$P_k = -\overline{u_i u_j}\frac{\partial U_i}{\partial x_j} \tag{1-11}$$

$$\varepsilon = -\nu\overline{\left(\frac{\partial u_i}{\partial x_j}\frac{\partial u_i}{\partial x_j}\right)} \tag{1-12}$$

The terms on the left member of equation (1-8) represent the local variation and the convection (by the mean flow) of the turbulent energy. The first part of the diffusion term (on the right member of equation (1-10)) represents the diffusion of the turbulent energy due to the turbulent velocity fluctuations and the second part is the diffusion by molecular transport, and it is important only in regions with low turbulence intensity. The production term (1-11) represents the transference of energy from the mean flow to the turbulence. Finally the dissipation term (1-12) represents the viscous dissipation of turbulent energy by the smaller eddies.

In order to close the system of equations for the calculus of the turbulent flow it is easy to realise that only the terms on the left member of equation (1-8) do not need modelling. The diffusion, production and dissipation terms need some kind of modelling, since in the form shown on equations (1-10), (1-11) and (1-12) they would introduce other unknowns into the system of equations.

Relating to the diffusion term, it is necessary to model only the first part (turbulent diffusion). This is done considering that the correlation of the turbulent quantities can be represented by the product of a diffusivity coefficient and the gradient of turbulent energy:

$$-\overline{u_j\left(\frac{u_i u_i}{2}+\frac{p}{\rho}\right)} \approx \nu_T \frac{\partial k}{\partial x_j}$$

The production term also presents a problem because of the complete Reynolds stress tensor $\overline{u_i u_j}$. This term is modelled by the Kolmogorov equation (1-6), with turbulent viscosity coefficient modelled by:

$$\nu_T = c_\mu \frac{k^2}{\varepsilon}$$

Where c_μ is a empirically determined parameter

Considering the above models, the equation of transport of turbulent kinetic energy takes the form:

$$\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j}\left[(\nu+\nu_T)\frac{\partial k}{\partial x_j}\right] + \nu_T\left[\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right]\left[\frac{\partial U_i}{\partial x_j}\right] - \varepsilon \tag{1-13}$$

Modelling of dissipation term ε requires somewhat arbitrary assumption of a turbulent length scale. This suggests that a more general approach for the model would be obtained if we consider ε as a new variable subject to the transport phenomena of the flow.

The transport equation for the dissipation ε can be obtained through the following steps: subtracting the mean momentum equation (1-5) from the total (or non-averaged) momentum equation (1-3). The resultant equation is differentiated with respect to x_k and multiplied by $\nu(\partial u_i / \partial x_j)$. The obtained equation is then time-averaged. (For detailed deduction see for instance Sloan at al. [1986]).

The final expression for the dissipation equation takes the form:

$$\frac{\partial \varepsilon}{\partial t} + C_\varepsilon = D_\varepsilon + P_\varepsilon + d_\varepsilon \quad \textbf{(1-14)}$$

Where:

$$C_\varepsilon = U_j \frac{\partial \varepsilon}{\partial x_j} \quad \textbf{(1-15)}$$

$$D_\varepsilon = -\frac{\partial}{\partial x_j}\left[\overline{\nu u_j \frac{\partial u_i}{\partial x_m}\frac{\partial u_i}{\partial x_m}} + \frac{2\nu}{\rho}\overline{\frac{\partial p}{\partial x_i}\frac{\partial u_j}{\partial x_i}} - \nu \frac{\partial \varepsilon}{\partial x_j} \right] \quad \textbf{(1-16)}$$

$$P_k = -2\nu \frac{\partial U_i}{\partial x_j}\left(\overline{\frac{\partial u_i}{\partial x_m}\frac{\partial u_j}{\partial x_m}} + \overline{\frac{\partial u_k}{\partial x_i}\frac{\partial u_k}{\partial x_j}} \right) - 2\nu \overline{\frac{u_j \partial u_i}{\partial x_m}} \frac{\partial^2 U_i}{\partial x_j \partial x_m} - 2\nu \overline{\frac{\partial u_i}{\partial x_j}\frac{\partial u_i}{\partial x_m}\frac{\partial u_j}{\partial x_m}} \quad \textbf{(1-17)}$$

$$d_\varepsilon = -2\overline{\left(\nu \frac{\partial^2 u_m}{\partial x_j \partial x_m} \right)^2} \quad \textbf{(1-18)}$$

The convective term and the last term in the brackets of the diffusion term Dε are the only ones which do not need to be modelled. All the other terms, involving averages of correlations of derivatives of fluctuating quantities, have to be modelled in order to yield, (together with the mean velocity equations and the turbulent kinetic energy equation), a closed system.

Taking into account the necessity of modelling the terms above mentioned, the transport equation for the dissipation ε takes the form:

$$\frac{\partial \varepsilon}{\partial t} + U_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\varepsilon} + \nu \right) \frac{\partial \varepsilon}{\partial x_j} \right] + c_{\varepsilon 1}\frac{\varepsilon}{k}P_k - c_{\varepsilon 2}\frac{\varepsilon^2}{k} \quad \textbf{(1-19)}$$

where Pk is the production term for the kinetic energy given by equation (1-11) and the tensor $\overline{u_i u_j}$ is given again by equation (1-6).

The first term on the right member of equation (1-19) is the usual model for a diffusion of a quantity, making use of the gradient of the quantity being diffused and of a diffusion coefficient. The other two terms, production and dissipation terms, deserve some comments.

Let's take the production term. It seems reasonable to suppose that production of kinetic energy should be balanced by the production of dissipation in order to avoid an unlimited grow of the turbulent kinetic energy. Making use a dimensional analysis it is easy to observe that the rate do production of turbulent energy (per unit of mass of fluid) has the units of L2T-3 whereas the rate of production of dissipation (also per unit of mass) has the units of L2T-4. The natural time scale to relate both quantities is given by: $T = k/\varepsilon$. So it follows:

$$P_\varepsilon \propto \frac{\varepsilon}{k}P_k$$

The other term, which could be understood as the destruction of dissipation, must have a form that annihilates dissipation, i. e. become infinitely large, when the energy of the turbulence approaches zero, in order to avoid negative values of turbulent kinetic energy. Another desirable feature of the term of destruction of dissipation is that it must increase as the dissipation itself increases. Dimensional analysis similar to the one described in the previous paragraph leads to:

$$d_\varepsilon \propto \frac{\varepsilon}{k}\varepsilon$$

The two-equation model for high values of the Reynolds number can be summarised as follows:

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=\frac{\partial}{\partial x_j}\left[\nu_T\frac{\partial k}{\partial x_j}\right]+\nu_T\left[\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i}\right]\left[\frac{\partial U_i}{\partial x_j}\right]-\varepsilon \tag{1-20}$$

$$\frac{\partial \varepsilon}{\partial t}+U_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\varepsilon}+\nu\right)\frac{\partial \varepsilon}{\partial x_j}\right]+c_{\varepsilon 1}\frac{\varepsilon}{k}P_k-c_{\varepsilon 2}\frac{\varepsilon^2}{k} \tag{1-21}$$

where:

$$\nu_T=c_\mu\frac{k^2}{\varepsilon} \tag{1-22}$$

The k-ε model as above described has five empirical constants. The determination of these constants, in the standard model (Launder and Spalding [1974]), was based on experiments performed for some kinds of flow conditions, hindering the expected generality of the model.

The values of the empirical constants for the standard model are:

$$c_\mu=0.09,\ c_{\varepsilon 1}=1.44,\ c_{\varepsilon 2}=1.92,\ \sigma_k=1\ \text{ and }\ \sigma_\varepsilon=1.3$$

This model can be successfully applied to regions of the flow with high values of the Reynolds number but it cannot be applied, in its standard form, near walls, where viscous effects become dominant and some kind of law of wall approach has to be used.

Despite its good performance for many engineering applications the k-ε model still has some limitations when applied to hydrocyclone flow description, due to the accentuated curvature of the stream lines and the presence of strong body forces (due to rotation).

Fluid particles' path within a hydrocyclone shows a spiral form with a very small radius of curvature. Furthermore, the fluid particles are subject to centrifugal acceleration which is a function of the azimuthal component of the velocity. The azimuthal velocity has also a turbulent fluctuation which generates a fluctuation on the centrifugal acceleration. This latter fluctuation affects the transport of turbulent quantities.

Let's discuss further about the limitations of the k-ε model aiming its utilisation in hydrocyclone's flow simulation.

Starting with a general approach, it should be emphasised that this model, as still based on the turbulent viscosity concept (Boussinesq hypothesis) lacks capacity to deal with the anisotropic flow conditions which takes place within a hydrocyclone, as was already mentioned.

Bradshaw, in his comprehensive study on the effect of streamline curvature on the turbulent flow [1973], concluded that this curvature has a strong effect on the processes by which the Reynolds stresses are generated and maintained and that empirically adjusted shear flow models are unlikely to catch that effect.

This author even suggested that linear relations between the Reynolds stress and the mean rate of strain, even with an amplification factor for some direction of strain, as in equation (1-7), are not reliable in the presence of large curvatures of the streamlines. That suggestion is based on the fact that the curvature effects are attributed to changes in higher-order structure parameters of the flow and in consequence it should not be expected that the full effects on the local Reynolds stresses appear as soon as the curvature is imposed.

Launder et al. [1977] proposed a modified k-ε model with a correction on the ε equation based on a non-dimensional parameter called Richardson number, which can be interpreted as the ratio between centrifugal force (generated by streamline curvature) and inertia force. For more details see Launder, Priddin and Sharma [1977] and Bradshaw [1969]. That correction gives better results in the presence of curvature, but for complex geometry of the flow it is difficult to interpret suitably the Richardson number and the form of correction to be persued.

Let's examine closely the problem of a surface curvature that produces streamline curvatures. If we consider the general Reynolds stress transport equation, the turbulent stress production term (from mean flow deformation) can be written as:

$$P_{\overline{u_i u_j}} = -\left(\overline{u_i u_k} \frac{\partial U_j}{\partial x_k} + \overline{u_j u_k} \frac{\partial U_i}{\partial x_k} \right)$$

Considering a bi-directional flow (shearing flow on a curved surface) Bradshaw [1973] concluded, as was already mentioned, that even small curvatures in the flow yields noticeable alterations on the turbulent stresses. In this case, the production term above assumes the form:

$$P_{\overline{uv}} \cong -\left(\overline{v^2} \frac{\partial U}{\partial y} + \overline{u^2} \frac{\partial V}{\partial x} \right)$$

and

$$P_{\overline{v^2}} \cong -2\overline{uv} \frac{\partial V}{\partial x}$$

Even if the surface curvature is very small, and generates a main direction gradient of the normal component of the velocity very small compared to the normal gradient of the main component of the velocity, let's say: $\frac{\partial V}{\partial x} \approx 0.02 \frac{\partial U}{\partial y}$, its influence on the turbulent stress production term might be strong.

Near the wall $\overline{u^2}$ is much greater than $\overline{v^2}$ (see Schlichting [1968]) and it makes bigger the influence of $\frac{\partial V}{\partial x}$ on the production of $\overline{uv}$. Furthermore, from the production term of $\overline{v^2}$ it is clear that for a positive value of the gradient $\frac{\partial V}{\partial x}$, which occurs for a concave surface, there is an increase in the turbulent stress $\overline{v^2}$.

From the above considerations, it is verified that the influence of $\frac{\partial V}{\partial x}$ on $\overline{uv}$ is ten to fifteen times greater than the influence of $\frac{\partial U}{\partial y}$. Of course, this different degree of influence cannot be generated by an expression like the Boussinesq's":

$$-\overline{uv} = \nu_T \left(\frac{\partial U}{\partial y} + \frac{\partial V}{\partial x} \right)$$

1.4 Corrections aiming taking into account curvature of streamline and rotation

Due to the complexity of turbulent flows, most of the technical literature on turbulence focuses on simple shear layers flows, i.e. flows with only one significant rate of strain. The presence of curved streamlines originates extra rates of strain that cannot be dealt with using the linear approach given by the Boussinesq relation – equation (1-6) –, as was already above commented. It was mentioned, that even small extra rates of strain, caused by mild streamline curvature, cannot be treated by a theory based on a zero order tensor turbulent viscosity.

Therefore, turbulence models based on turbulent viscosity concept, which were developed aiming the study of flows with only one significant rate of strain, are not optimised for the solution of flows with more complex strain conditions.

It seems advantageous for engineering purposes, however, to extend the range of application of the simplest turbulence differential models to include flow fields with curved streamlines. Therefore, many researchers have worked on developing correction factors to be employed on models based on turbulent viscosity concept. The performance of these adapted models in predicting the effect of streamline curvature and/or swirl on the turbulent stresses vary from reasonably well to fairly poor; and the desired generality of any of them has never been achieved.

Adaptation of the models based on the turbulent viscosity

Prandtl as far as in 1929 (see reference in Bradshaw [1973] or So [1975]) has proposed a factor based on a dimensionless curvature parameter for correcting the mixing length. Bradshaw [1969] has defined a curvature Richardson number from a analogy with the buoyancy Richardson number which is non-dimensional parameter used by some authors in a correction function on the dissipation rate equation in the k-ε model, for some kinds of flows.

The deficiencies of the k-ε model are generally attributed to the lack of physical grounds of the modelled dissipation (ε) equation, and in this equation the source terms are the ones which requires the bolder assumptions.

The approach to correct the k-ε model for the effects of curvature of the streamline or for the effects of swirl is one of two main types:

correction of the source terms of the ε equation (most of proposals actuate specifically on the term which represents the destruction of dissipation) – usually modifying the coefficient $c_{\varepsilon 2}$ in equation (1-21)

and corrections on the coefficient c_μ for the calculus of turbulent viscosity ν_T equation (1-22)

It is reasonable to expect that the correction factor directly on the eddy viscosity makes the effects of curvature be instantaneously imposed on the flow, while the correction applied on the equation for the rate of dissipation of turbulent energy will impose the correction to all the effects present in a transport equation and so they should be more suitable for complex flows.

Correction for curvature of the streamlines

From the analogy between curvature and buoyancy (Bradshaw [1969]) it is defined the gradient Richardson number for curvature (Bradshaw[1969]):

$$Ri_T = \frac{k^2}{\varepsilon^2} \frac{2U_s}{R_c^{\,2}} \frac{\partial (R_c U_s)}{\partial R_c} \qquad \textbf{(1-23)}$$

where R_c is the local radius of curvature of the streamline and U_s is the mean velocity along the streamline.

This parameter is used in a correction factor which multiplies the coefficient $c_{\varepsilon 1}$ in the destruction of dissipation of the ε equation:

Thus, this term takes the form:

$$\text{“Destruction of dissipation”} \equiv c_{\varepsilon 2}\left(1 - c_c Ri_T\right)\frac{\varepsilon^2}{k} \qquad \textbf{(1-24)}$$

where $c_c = 0.2$ (Launder, Priddin and Sharma[1977]).

Corrections for swirling flows

The proposal above described is recommended for flows with curved streamlines without swirl. For the case of strongly swirling flows with recirculation, however, correction analysis is more complex, as discussed below:

Stabilising effect of the swirl

An analogous to the curvature Richardson number proposed to represent the swirl effect is proposed by Launder at al. [1977] and takes the form:

$$Ri_{gs} = \frac{k^2}{\varepsilon^2}\frac{W}{r}\left(\frac{\partial W}{\partial r} + \frac{W}{r}\right) \quad \textbf{(1-25)}$$

The correction is introduced again as a modification of the dissipation term of ε equation, thus:

$$\text{“Destruction of dissipation”} \equiv c_{\varepsilon 2}\left(1 - c_s Ri_{gs}\right)\frac{\varepsilon^2}{k} \quad \textbf{(1-26)}$$

where $c_s = 0.002$.

The above expression for the Richardson number (1-25) results in a positive value of this parameter in the case of positive gradient of the angular momentum of the fluid particle (as is the case in solid body rotation) and, therefore, according to (1-21) it results in a reduction of destruction of dissipation, i.e., it results in a decrease in turbulent viscosity.

To evaluate the numerical variation of coefficient $c_{\varepsilon 2}$ due to the proposed correction let's consider an inviscid analytical model for hydrocyclone's flow, whose velocity profiles on the axial plane (U for axial component and W for azimuthal component) are given by the following equations (see articles by Bloor and Ingham [1973-1983]):

$$U = \frac{1}{2}B\, r^{-1/2}\left(3\alpha^* - \frac{5r}{x}\right) \quad \textbf{(1-27)}$$

$$W_{non\,\mathrm{dim}} = \frac{1}{R'}\frac{\gamma\left(\frac{4}{5},\frac{2}{5}C\, r'^{5/2}\right)}{\gamma\left(\frac{4}{5},\frac{2}{5}C\right)} \quad \text{where} \quad \gamma(n,y) = \int_0^y t^{n-1}\exp(-t)\,dt \quad \textbf{(1-28)}$$

where r and x are the radial and axial coordinates respectively (cylindrical coordinate system), and B, C and α* are parameters related to geometry and flow conditions of the hydrocyclone;

The expression for the gradient Richardson number for swirl, with time scale based on the mean flow, is as follows:

$$Ri_{gs} = \frac{2\frac{W}{r}\left(\frac{\partial W}{\partial r} + \frac{W}{r}\right)}{\left(\frac{\partial U}{\partial r}\right)^2 + \left[r\frac{\partial}{\partial r}\left(\frac{W}{r}\right)\right]^2} \quad \textbf{(1-29)}$$

It should be noted that the denominator of above expression (1-29) has dimension of reciprocal of time squared. In equation (1-30) this mean time scale was substituted by the turbulent time scale (k/ε).

Velocities profiles obtained from expression (1-27) and (1-28) for an arbitrarily chosen hydrocyclone geometry and flow conditions are presented below. These velocities are expressed in meters/second in a profile distributed over the non-dimensional radius (r/Rc – where Rc is the nominal radius of the hydrocyclone):

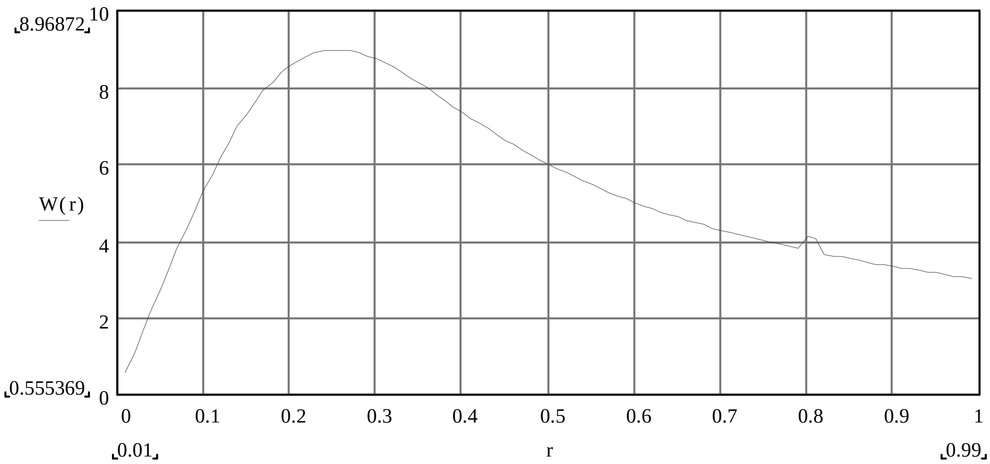

Figure 1-1 – Tangential Velocity

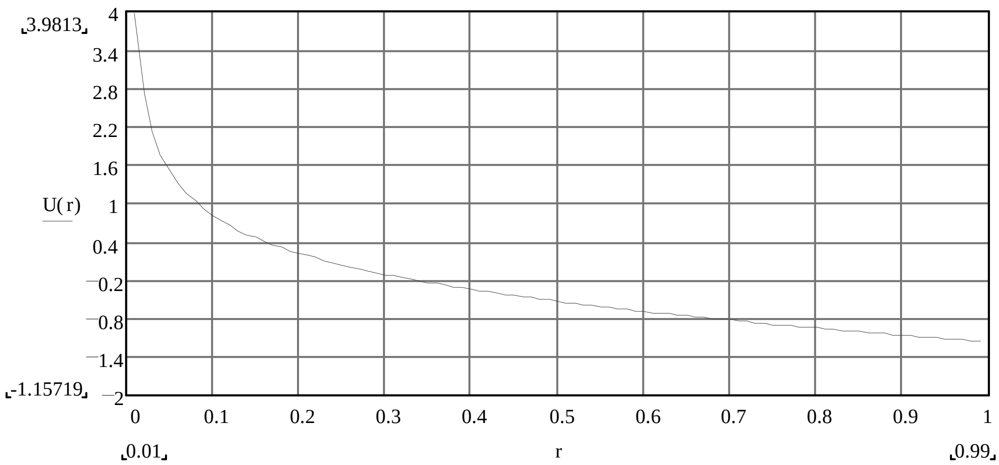

Figure 1-2 – Axial Velocity

Figure 1-1 shows that the inlet swirl velocity is 3 m/s (near the wall – $r = 1$) and it reaches a maximum of about 3 times this value at about 25% of the radius from the axis. From figure 1-2 it can be seen that the axial velocity near the axis (reverse flow) tends to infinity. This flow condition can be characterised as a strong swirl flow.

The distribution of the gradient Richardson number, given by expression (1-29), for these conditions is represented by the curve below:

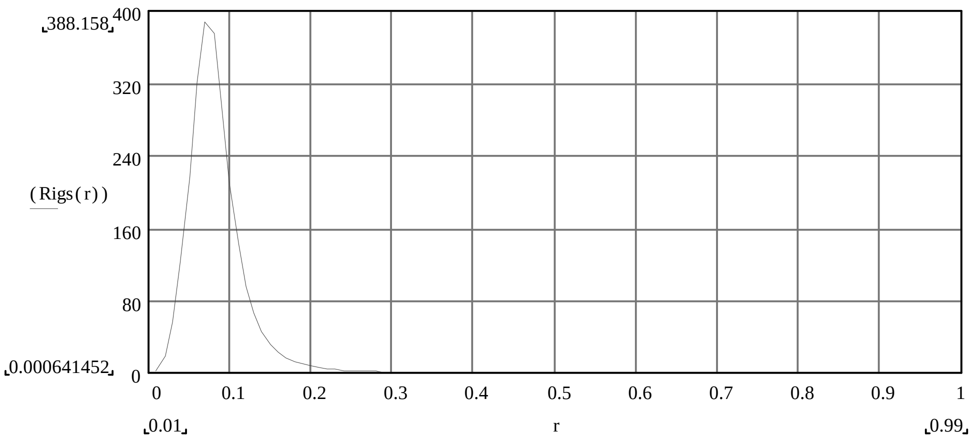

Figure 1-3 – Gradient Richardson number distribuition

It can be noted that Richardson number reaches very high values, in the region of solid body rotation, i.e. for $0 \le r \le 0.25$. These high values are due to the strong swirl of the flow. For even stronger swirl flows, the peek of the curve tends to even higher values.

The variation in corrected coefficient $c_{\varepsilon 2}$ (the term between parenthesis in equation 26 times $c_{\varepsilon 2}$) is given by the curve below (it should be noted that $c_{\varepsilon 2} = 1.92$ for the original – with no correction – κ-ε model):

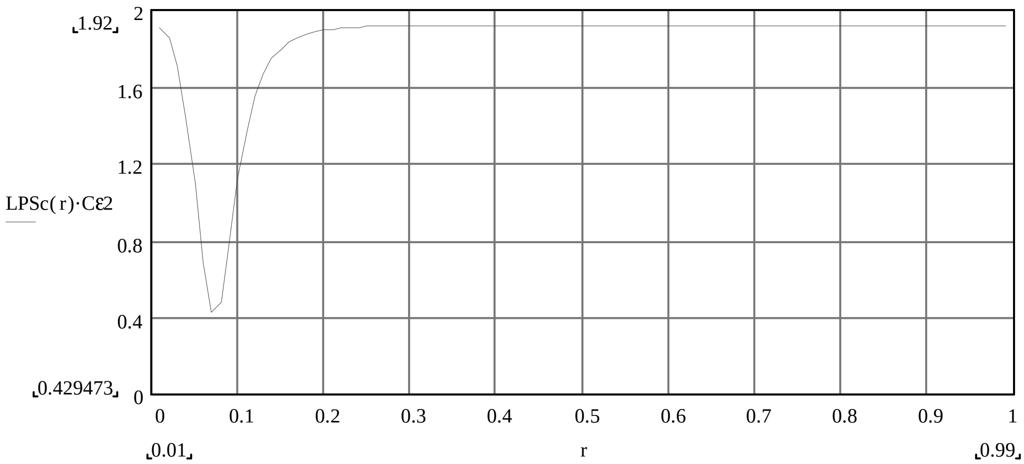

Figure 1-4 – Distribution of corrected $c_{\varepsilon 2}$ coefficient

Figure 1-4 above shows that, for the hydrocyclone and flow conditions considered in the above example, the range of variation of $c_{\varepsilon 2}$ coefficient is: $0.43 \le c_{\varepsilon 2} \le 1.92$, in the region of solid body rotation, it means that, in this region, the destruction of dissipation is substantially reduced (to less than ¼ of its normal value) due to the stabilisation effect of the swirl on turbulence. In the rest of the flow region, including the approximate free vortex outer region, no correction is obtained. This last result is due to the fact that, in latter region, the numerator of equation (1-29) and equation (1-30) become null.

Destabilising effect of the swirl

According to Sloan at al. [1986], Rodi claims that the behaviour of the flux Richardson number, instead of the gradient Richardson number, is more consistent with the destabilising effects of rotation in swirling jets and over spinning surfaces.

An expression for the flux Richardson number in terms of mean quantities is (see abovementioned reference):

$$Ri_{fs} = \frac{2\frac{W}{r}\left(\frac{\partial W}{\partial r} - \frac{W}{r}\right)}{\left(\frac{\partial U}{\partial r}\right)^2 + \left[r\frac{\partial}{\partial r}\left(\frac{W}{r}\right)\right]^2} \qquad \textbf{(1-30)}$$

The correction proposed by Rodi (see reference in Sloan at al. [1986]) has the objective of increasing turbulence (turbulent length scale or turbulent viscosity) with swirl. In opposition to the previously seen correction factor which acts on the destruction of dissipation term of dissipation equation, Rodi proposes the use of the flux Richardson number in a correction factor applied on the production term of dissipation equation ε. This corrected term takes the form:

Normal production of dissipation = $c_{\varepsilon 1}\frac{\varepsilon}{k}P_k$

Corrected production of dissipation = $c_{\varepsilon 1}\left(1+0.9Ri_{fs}\right)\dfrac{\varepsilon}{k}P_k$

Where P_k is the production of kinetic energy term (equations 1-11 and 1-6).

Analogously to what was done to the coefficient $c_{\varepsilon 2}$ for stabilizing effect, let's also analyse the numerical variation of corrected coefficient $c_{\varepsilon 1}$ due to the proposed correction for de-stabilizing effect. To do this let's consider the same conditions and velocity profiles used in the previous analysis – equations (1-27 and 1-28). We get the following results:

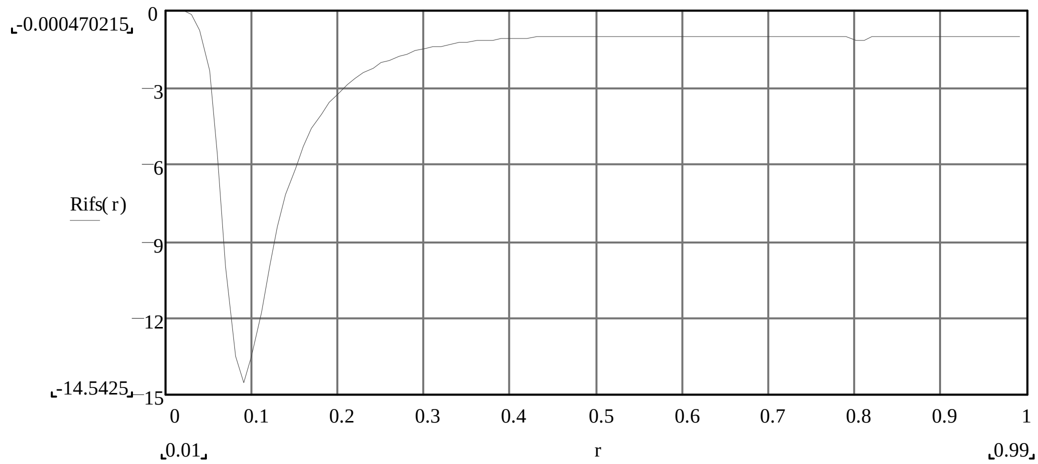

Figure 1-5 – Flux Richardson number distribution

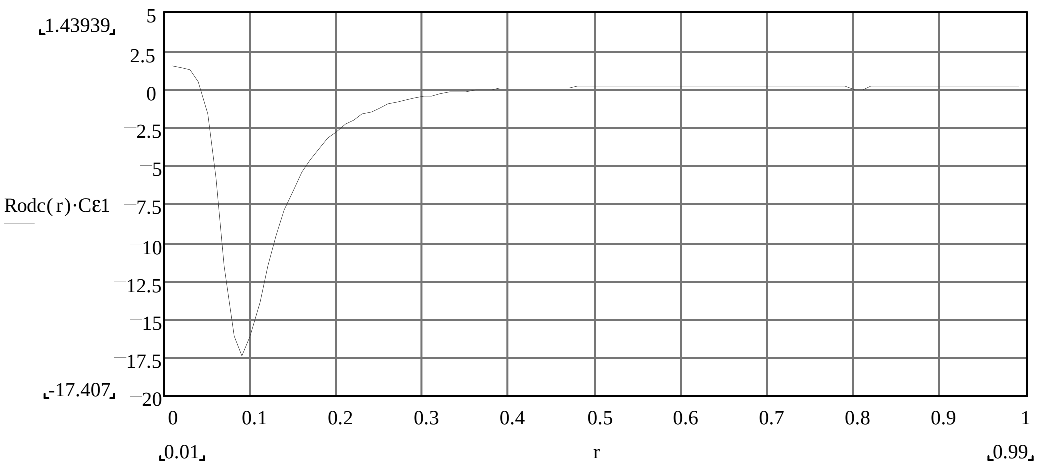

Figure 1-6 – Distribution of the correction on $c_{\varepsilon 1}$ coefficient

It can be observed from figure 1-5 that the flux Richardson number assumes negative values all over the domain of flow in the hydrocyclone. Figure 1-6 shows that the value of coefficient $c_{\varepsilon 1}$ is very much affected by the corrected form proposed by Rodi, particularly in the forced vortex region. In this region, this coefficient changes from the normal value $c_{\varepsilon 1} = 1.44$ (assumed by Launder and Spalding [1974]) to negative valuse of magnitudse more than ten times its recommended magnitude. It is difficult to interpret the consequences of this abrupt and very large change in the coefficient of the "production of dissipation" term in the dissipation equation. Not only the magnitude but also the sign inversion implies a strong locally non-equilibrium condition between turbulent energy production and dissipation in this region of the flow. Convergence of simulated solution by such a model can only be achieved if the transport terms in both k and ε equations were significant enough to compensate this local non-equilibrium imposed by the correction.

Another feature of the correction proposed by Rodi, contrarily to the previous one, is the extension of its influence to the free vortex region. Figure 1-6 shows that for free-vortex like profile ($r > 0.5$), the corrected coefficient is essentially constant $c_{\varepsilon 1} = 0.144$, i.e., about 10% of the normal value. This means a lower production of dissipation and, in consequence, higher turbulence than resulted by the non-corrected k-ε model.

2.0 Proposal of a modified k-ε turbulence model for hydrocyclone flow simulation

2.1 Model description

Analyzing, from a physical stand point, which should be the influence of the mean flow on turbulence in a hydrocyclone it is easy to conclude that there is a preferential direction for the vorticity that is the axial direction. We can say that this feature poses a constraint to turbulence which turns it "less chaotic". It seems to be a reasonable conclusion the consideration that turbulent eddies – at least the biggest ones – tend to align in a preferential direction turning turbulence in a hydrocyclone more ordered and so strongly anisotropic, resulting in the appearance of what was called by Lesieur [1997] as "coherent structures" and approaching "bidimensional turbulence", both statements usually considered heretic in the context of a chaotic system.

The above described phenomenon implies that no isotropic turbulent model can suitably describe hydrocyclone flow.

Simulation of the flow in hydrocyclones using two-equation turbulence models can capture reasonably well the shape of the axial and radial components of mean velocity profiles, but the azimuthal profile usually tends to the rigid body rotation shape which means lower rotation speed for smaller distances from de axis instead of the expected Rankine vortex shape profile which possesses the characteristic free-vortex shape in the out of core region.

We can analyze the reasons for this behavior of the above mentioned models. Cyclonic effect, i.e. the increasing in the magnitude of azimuthal component of velocity for smaller distances from the axis is a phenomenon characteristic of fluids of low viscosity. This phenomenon is essentially determined by inertial forces as can be seem when studying this flow considering an ideal (inviscid) approach which results in a free-vortex behavior for the azimuthal component and conducts to infinite rotational velocity on the axis. This non physical behavior is prevented in the non ideal fluid due to the action of viscosity which attenuates the velocity, imposing a rigid body pattern near the axis in order to assure the boundary condition of zero azimuthal velocity on it. Keeping the same set of flow conditions (boundary and initial conditions) but increasing molecular viscosity it turns more difficult to establish free-vortex like profile. In fact the increasing of viscosity results in an increasing in the rigid body rotation region and even to the suppression of free-vortex region, for some flow conditions.

From the above considerations we can conclude that the utilization of a turbulence model based on the concept of turbulent viscosity – which corresponds to an increase in the coefficient of molecular viscosity – will mimic the behavior of a much more viscous fluid than the actual one (since turbulent viscosity is orders of magnitude greater than molecular viscosity) and so preventing the formation of free-vortex pattern. In other words, these models are too diffusive to correctly represent the physics of the actual flow. Although increased diffusivity is a general characteristic of turbulent flows, for the present case of "less chaotic" turbulence this feature is not be desirable. To avoid this problem we can use a complete Reynolds Stress Model with the penalty of increasing computational cost.

To keep the advantage of computational simplicity of the k-ε two-equation model and simultaneously to avoid the behavior above described we propose to actuate on generation terms of ε equation taking into account some physical features of hydrocyclone flow. Our proposal is to impose two opposite corrections actuating in different regions of the flow domain. Of course this is a disturbing procedure since it will impose a sudden non equilibrium condition in the turbulent variables but as far as we can do it based on the physics of the present flow it may conduct to better results than the ones obtained with the non modified model.

The correction proposed here is based on a proposal from Launder et al. [1977] using a non-dimensional parameter called Richardson number for curvature and rotation (Bradshaw [1969]), as discussed in the previous section. This parameter can be regarded as expressing a relation between centrifugal forces due to rotation and inertia forces. The attempting of application of this kind of correction for hydrocyclones has not been successful so far since it is somewhat arbitrary to define what kind of correction is required. It is difficult, in such a complex flow, to know if turbulence is to be amplified or attenuated. Our proposal tries to shed some light on how to make these kinds of corrections based on the physics of the problem, considering particularly the equilibrium between radial pressure gradient and centrifugal forces.

The shape of the azimuthal velocity profile in a hydrocyclone is previously known from experimental data and it corresponds, as abovementioned, to a core region of "rigid body" like profile and an outer region of "free vortex" behavior. We also know that in the region with "rigid body" behavior when a fluid particle (turbulent "lump" of fluid) is project radially outwards it finds a region of higher pressure than the one that can be resisted by its centrifugal force. This happens because in this outer region the average rotation velocity is higher and so it is the centrifugal force which is in equilibrium with radial pressure. Thus, in the "rigid body" region, the mentioned fluid particle will face a restoring force opposing its radial movement or we can say that turbulence is attenuated. For other hand, in the region of "free vortex" behavior, same reasoning shows us that the non equilibrium between a projected fluid particle velocity and its neighbors' velocity will keep or even increase turbulence intensity.

The effects described above can be achieved through suitable corrections on source terms of ε equation in the k-ε turbulence model, as was already mentioned. There it follows the proposed modification:

$$\frac{\partial k}{\partial t}+U_i\frac{\partial k}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_k}+\nu\right)\frac{\partial k}{\partial x_j}\right]+P_k-\varepsilon \qquad \textbf{(2-1)}$$

$$\frac{\partial \varepsilon}{\partial t}+U_i\frac{\partial \varepsilon}{\partial x_i}=\frac{\partial}{\partial x_i}\left[\left(\frac{\nu_T}{\sigma_\varepsilon}+\nu\right)\frac{\partial \varepsilon}{\partial x_j}\right]+c_{\varepsilon 1}\frac{\varepsilon}{k}P_k-c_{\varepsilon 2}\frac{\varepsilon^2}{k} \qquad \textbf{(2-2)}$$

where $\nu_T=c_\mu\dfrac{k^2}{\varepsilon}$ and $P_k=\nu_T\left[\dfrac{\partial U_i}{\partial x_j}+\dfrac{\partial U_j}{\partial x_i}\right]\left[\dfrac{\partial U_i}{\partial x_j}\right]$

with $c_\mu=0.09$ and $c_{\varepsilon 1}$ and $c_{\varepsilon 2}$ area parameters which are going to vary increasing or decreasing source terms so as to implement the desired corrections in following form:

If we call azimuthal velocity component of V_θ and r the axial coordinate (zero on the axis) we propose:

If $\dfrac{\partial V_\theta}{\partial r}\geq 0$ then $c_{\varepsilon 2}=1.92(1-c_c Ri)$ **(2-3)**

and $c_{\varepsilon 1}=1.44$.

Where Richardson's number is given by: $Ri=\dfrac{k^2}{\varepsilon^2}\dfrac{V_\theta}{r}\left(\dfrac{\partial V_\theta}{\partial r}+\dfrac{V_\theta}{r}\right)$ **(2-4)**

If $\dfrac{\partial V_\theta}{\partial r}<0$ then $c_{\varepsilon 1}=1.44(1+c_f Ri)$ **(2-5)**

and $c_{\varepsilon 2}=1.92$.

Where Richardson's number is given by:

$$Ri=\frac{k^2}{\varepsilon^2}\frac{V_\theta}{r}\left(\frac{\partial V_\theta}{\partial r}-\frac{V_\theta}{r}\right) \qquad \textbf{(2-6)}$$

The first one of the above corrections acts only in the regions where azimuthal velocity increases with radial distance from the axis and it reduces the parameter which is responsible for the destruction of dissipation rate, thus increasing dissipation and so reducing turbulence. This, as mentioned, is in accordance with the physics of the problem.

The second correction has an opposite effect but actuates only in the regions where azimuthal velocity component decreases for higher values of the distance for the axis. This behavior is also in accordance with the physics of the problem.

The values of the new parameters c_c and c_f can be optimized so as to give the model a good performance for hydrocyclone flow. This optimization can be made using experimental data or, in its absence, from data extracted from results of simulation using higher order turbulence models. This second option was tested in the following section.

2.2 Implementation in CFD of the modified model and some comparative results

To evaluate the performance of the modified k-ε model comparing to the non-modified one we implemented this model in a CFD code (it was choose the CFX-5.6) comparing the same results of both models and results obtained from SSG Reynolds Stress Model available in this code.

Since CFX internally calculates velocity gradients using a Cartesian mesh we have to adapt the equations for the correction above described, that are in cylindrical coordinates, in the variables used by CFX. That can be done in the following form:

$$r = \sqrt{X^2 + Y^2} \quad \text{and} \quad \theta = arctan(Y/X) \qquad \textbf{(2-8 a, b)}$$

$$X = r\,cos\,\theta \quad \text{and} \quad Y = r\,sin\,\theta \qquad \textbf{(2-9 a,b)}$$

Azimuthal velocity can be expressed as:

$$V_\theta = V\,cos\,\theta - U\,sin\,\theta \qquad \textbf{(2-10)}$$

Where U and V are functions of X and Y, which are retangular coordinates. It is also necessary to calculate an expression for $\partial V_\theta / \partial r$ in function of retangular coordinates

This expression assumes the following form

$$\frac{\partial V_\theta}{\partial r} = \frac{1}{X^2+Y^2}\left[X^2\frac{\partial V}{\partial X} - \frac{X^3 V}{X^2+Y^2} + XV - XY\frac{\partial U}{\partial X} + \frac{X^2 YU}{X^2+Y^2}\right] + \frac{1}{X^2+Y^2}\left[XY\frac{\partial V}{\partial Y} + \frac{Y^3 U}{X^2+Y^2} - YU - Y^2\frac{\partial U}{\partial Y} - \frac{XY^2 V}{X^2+Y^2}\right] \qquad \textbf{(2-11)}$$

Considering the above expression the above described modified model was implemented into CFX code so as to let us adjust the parameters c_c and c_f to best fit the results obtained using SSG Reynolds Stress Model.

To make the simulations we choose a hydrocyclone with the geometrical characteristics shown below and the flow was considered single phase water flowing into the domain with an average velocity on the entrance of 1.324 m/s.

Dimensional Characteriscs of the Hydrocyclone (mm)	
Cylindrical head diameter(*D*)	76,2
Feed pipe diameter (D_I)	21,3
Overflow diameter (D_o)	25,9
Underflow diameter (D_u)	12,4
Cone angle (θ)	11,3°
Vortex finder length (*l*)	30,5
Lenght of cylindrical head (*H*)	38,1
Total lenght of the hidrociclone (*L*)	381,0

Table 2-1 – Dimension of the hydrocyclone

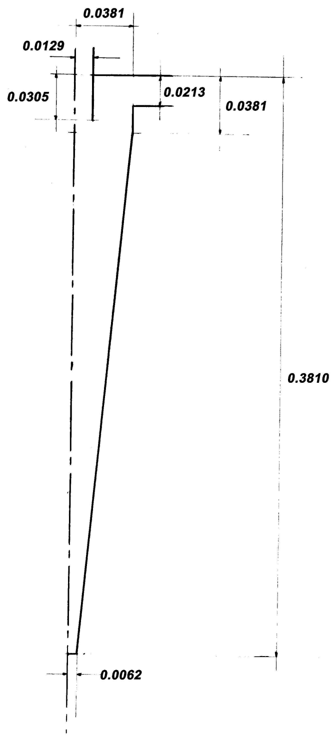

Figure 2-1 – Half axial plane section of the hydrocyclone (dimensions in meters)

For all the simulations it was used a hexahedral mesh of about 200,000 nodal points which projection on domain surface is shown in figures xxxx and xxxx below. To avoid imposition of unrealistic boundary conditions at hydrocyclone exits we considered as part of flow domain for the purpose of simulation two regions which can provide free development of flow conditions ("far field conditions"). These regions are shown on figure xxx.

Figure 2-2 – General view of surface mesh on domain

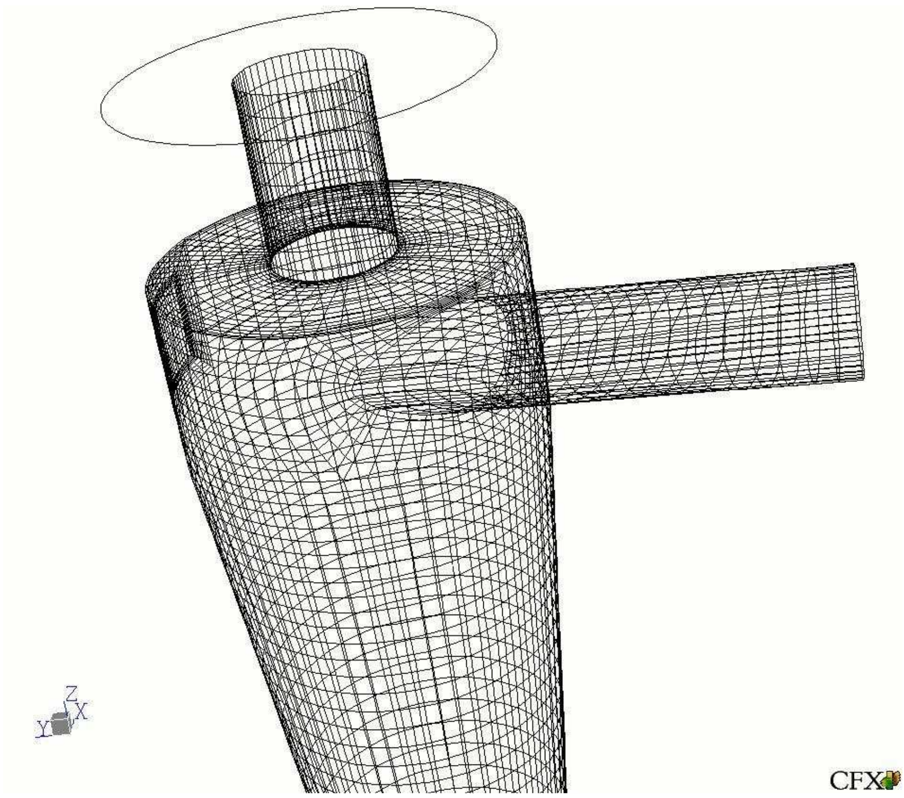

Figure 2-3 – Close detail of surface mesh

In order to have a completely defined modified model we have to determine the values of the parameters C_c and C_f from equations 2.3 and 2.5. In the search of values of parameter C_f we started from a maximum value of 0.9 which is the one recommended by Rodi (see reference in Sloan et al. [1986]) for simulation of flows with curvature of stream line where there is an amplification of turbulence (jets on concave surfaces). But even considering values orders of magnitude lower than this value we could not get solutions since the problem turned unstable and an explosion of numerical values of mean variables occurred. In other words, any reduction of rate of turbulent energy dissipation conducts to an increase in turbulence energy that can produce, due to the coupling of the equations, an "explosive" behavior for the mean variables.

However, a conceptual analysis of the flow field in a hydrocyclone seems to indicate that a correction aiming to increase turbulence, even in the "free-vortex" region, is not suitable to properly describe the flow. In this region flow is essentially irrotational, that means that turbulence cannot be generated there, it can only be transported in this region being generated in regions where rotational effects are prevalent (core region and walls). Thus it seems that in turbulent variable equations the terms of generation of turbulent energy for the non-modified model (with its constants optimized for other kinds of flow) are strong enough to increase turbulence in the "free-vortex" region without any need for a complementary amplification.

Relating to the correction in the "rigid body" behavior region we started from a value 0.002 for the parameter C_c, which was proposed by Launder (see reference in Sloan et al. [1986]) for attenuation of turbulence in cases of stream line curvature and rotation, obtaining good results. An even better agreement between the results with the modified model and SSG Reynolds Stress model was obtained when the value of C_c reached 0.004. For both cases, the value of parameter C_f was fixed 0.000 – which means no correction for "free-vortex" region.

As can be seen from figures xxx and xxx, modified k-ε turbulence model produces a better agreement with SSG Reynolds Stress model than the conventional k-ε turbulence model, since the latter resulted in a "rigid body" profile on whole domain while the former resulted in a Rankine like profile as was also obtained with the SSG model. Furthermore, modified model overestimated the maximum value of azimuthal velocity in about 10% related to SSG model while the non-modified model underestimated this value in about 15%.

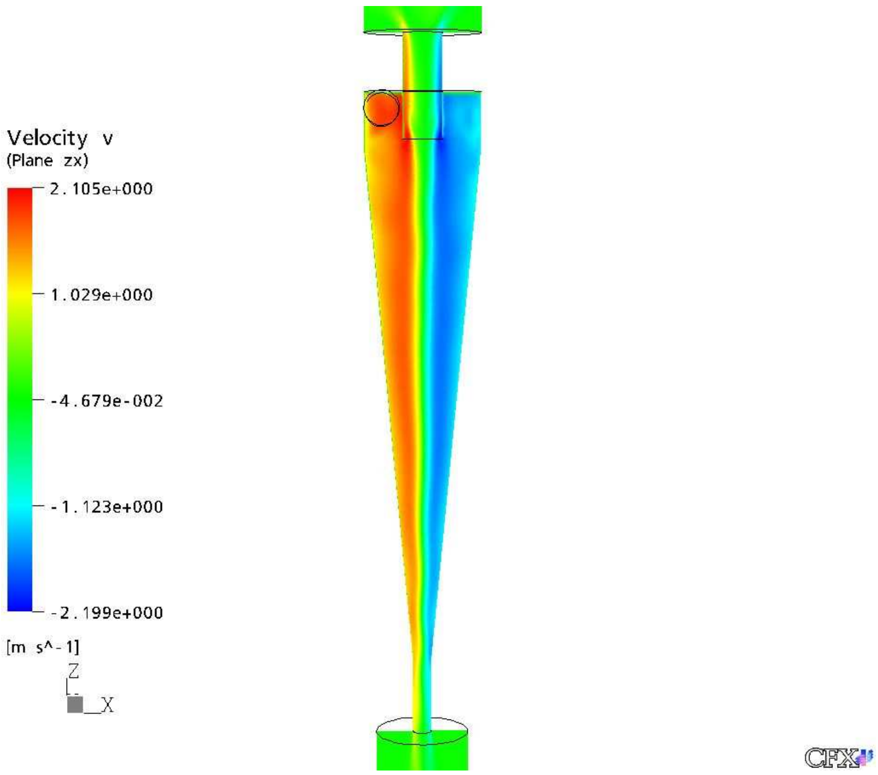

Figura 2-4 – Azimuthal component distribution on the axial plane of the hydrocyclone SSG Reynolds Stress Model

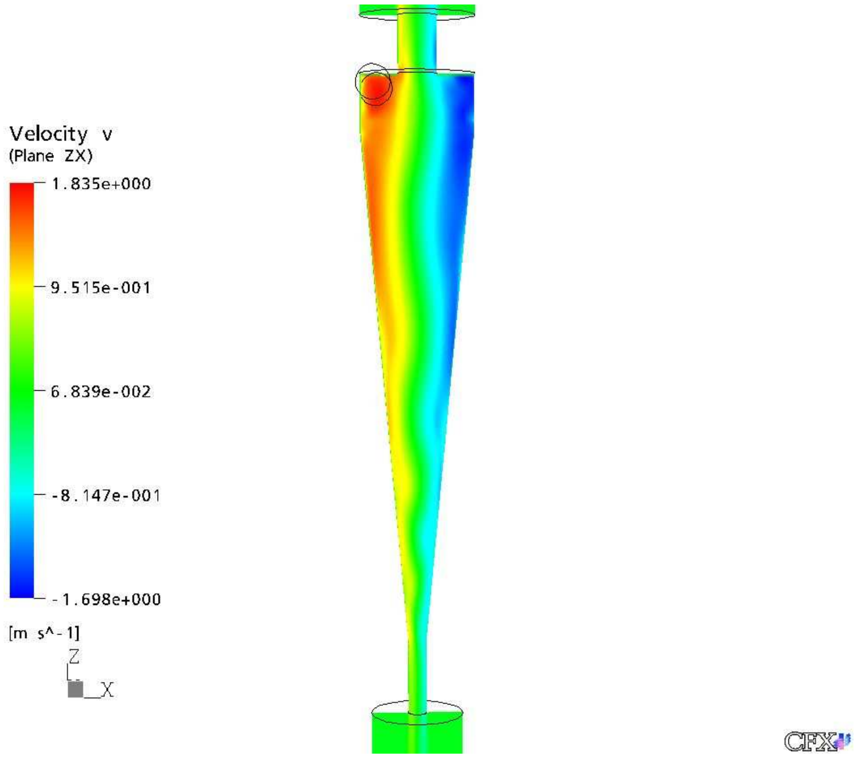

Figura 2-5 – Azimuthal component distribution on the axial plane of the hydrocyclone Conventional *k*-ε model

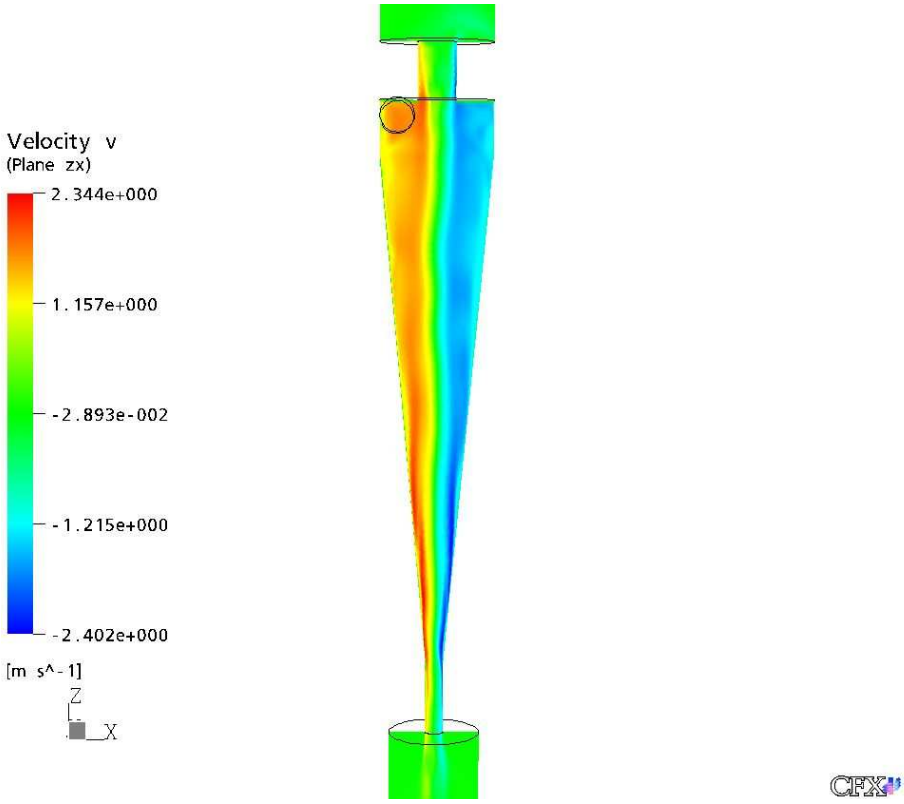

Figura 2-6 – Azimuthal component distribution on the axial plane of the hydrocyclone modified *k*-ε model ($C_c = 0.002$)

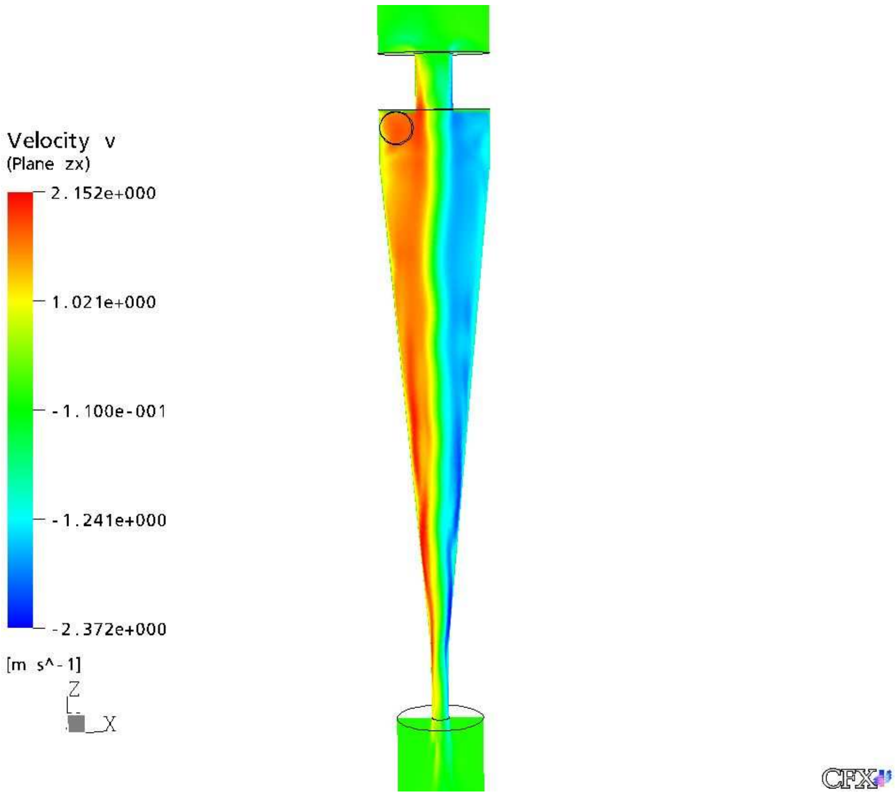

Figura 2-7 – Azimuthal component distribution on the axial plane of the hydrocyclone modified *k*-ε model ($C_c = 0.004$)

Total CPU time for both k-ε models was about 80,000 seconds, to simulate 1,300 variable time steps in steady state mode while for the Reynolds Stress model this total time was about 200,000 seconds. These figures were obtained in a P-IV, 1.8 GHz, serial processing unit (one CPU).

It is presented below the radial distribution of azimuthal velocity profiles obtained from simulations using the abovementioned models for three different distances (z) from the smaller cross section of the cone (underflow). The axial plane chosen to plot these profiles was the one normal to the feed entrance pipe, although it can be said that the flow was pretty much axissimetric.

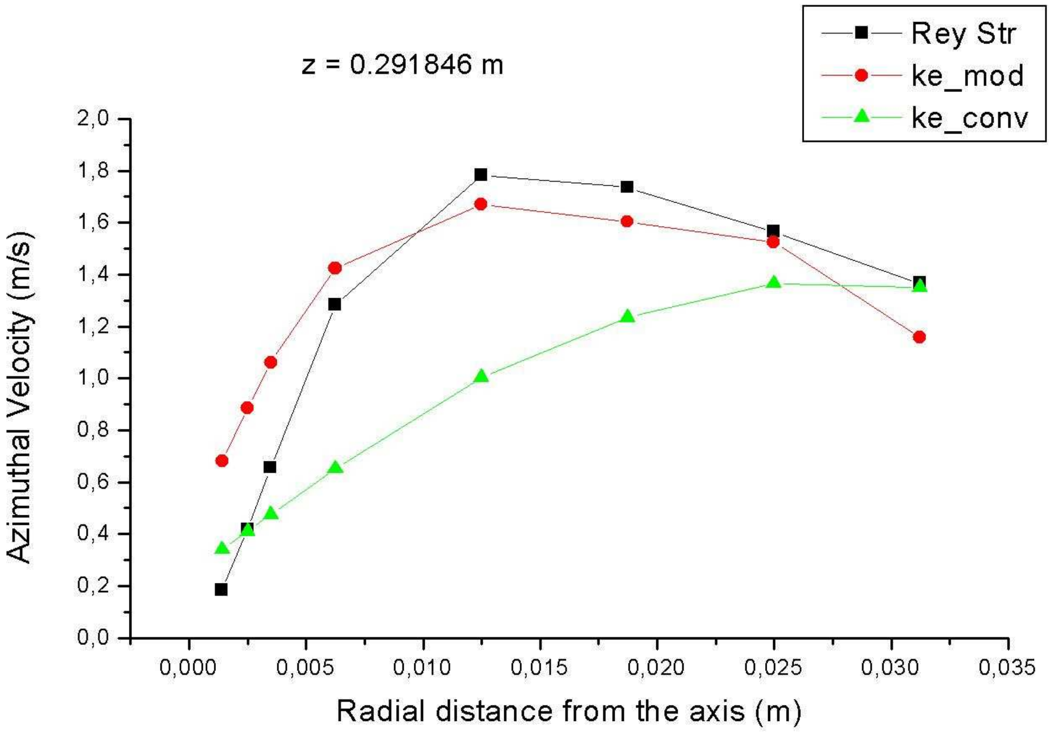

Figure 2-8 – Radial profile of azimuthal velocity component for z = 0.291846 m

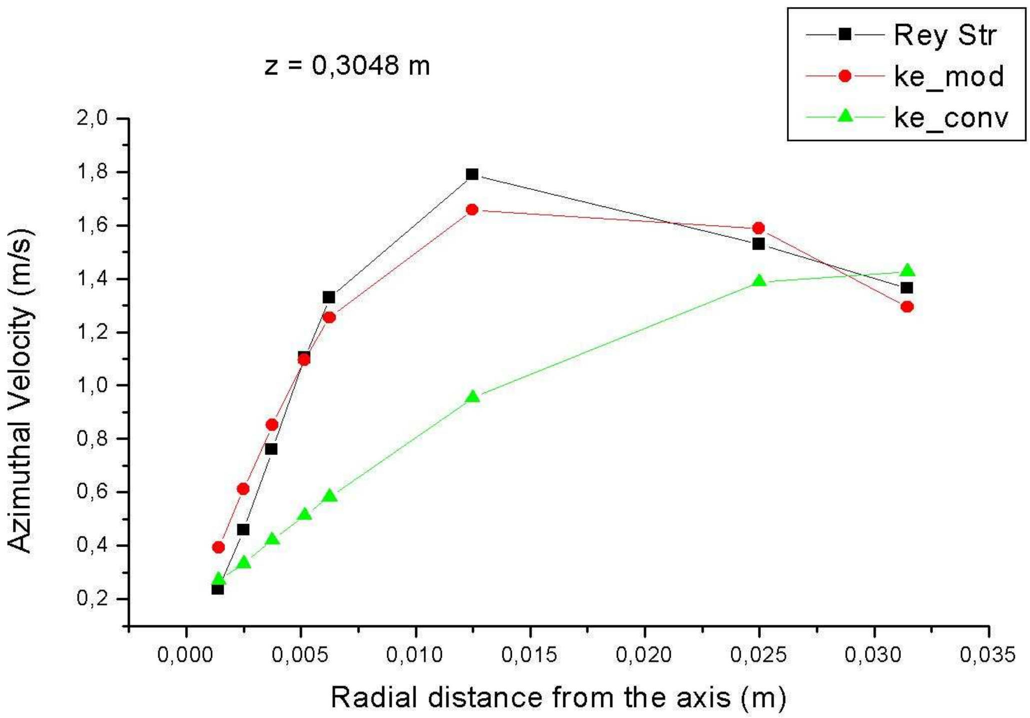

Figure 2-9 – Radial profile of azimuthal velocity component for z = 0.3048 m

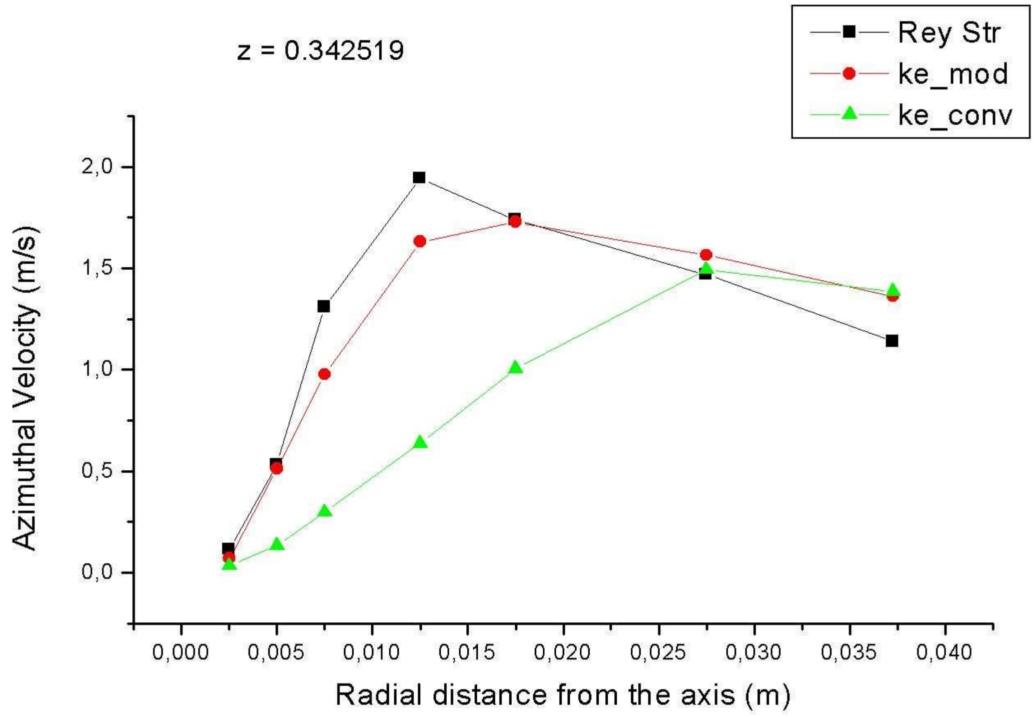

Figure 2-10 – Radial profile of azimuthal velocity component for z = 0.342519 m

3. References

ABCM, COPPE/UFRJ & IME/RJ (Editors) – First Spring School on Transition and Turbulence - Mini Courses – 21-25/Sept./1998 – Rio de Janeiro/Brazil

Bloor, M.I.G.; Ingham, D.B. – Theoretical of the Fluid Mechanics of Hydrocyclone Flow – Filtration and Separation, July-August/1984

Bloor, M.I.G.; Ingham, D.B. – Theoretical Aspects of Hydrocyclone Flow – Progress in Filtration and Separation, Vol. 03, 1983

Bradshaw, P. – Effects of Streamline Curvature on Turbulent Flow – Advisory Group For Aerospace Research and Development (AGARD) – North Atlantic Treaty Organization – AGARDograph no.169 – 1973

Bradshaw, P. – The Analogy between Streamline Curvature and Buoyancy in Turbulent Shear Flow – Journal of Fluid Mechanics, vol. 36, part 1, pp. 177-191, 1969

Capela Moraes, C.A. – An investigation of the flow in a Hydrocyclone Operating without an Air Core – D.Sc. Thesis – Federal University of Rio de Janeiro – November 2003

Hoffmann, P.H.; Muck, K.C. and Bradshaw, P. – The effect of concave surface curvature on turbulent boundary layers – Journal of Fluid Mechanics, vol.161, 1985

Launder, B.E.; Priddin, C.H. and Sharma, B.I. – The Calculation of Turbulent Boundary Layers on Spinning and Curved Surfaces – Journal of Fluids Engineering, March, `1977

Launder, B.E. and Spalding, D.B. – The Numerical Computation of Turbulent Flows – Computer Methods in Applied Mechanics and Engineering 3 – North-Holland Publishing Company – 1974

Schlichting, H. – Boundary-Layer Theory – McGraw Hill Book Company – 1968

Sloan, D.G.; Smith, P.J. and Smoot, L.D. – Modelling of Swirl in Turbulent Flow Systems – Progress in Energy and Combustion Science, vol.12, 1986

So, Ronald M.C. – A turbulence Velocity Scale for Curved Shear Flows – Journal of Fluid Mechanics, vol.70, 1975