A benchmark test case for swirling flows: design of the swirl apparatus, experimental data, and numerical challenges

R F Susan-Resiga\textsuperscript{1,2}, C Popescu\textsuperscript{1}, R Szakal\textsuperscript{1}, S Muntean\textsuperscript{2,1}, A Stuparu\textsuperscript{1}
\textsuperscript{1} Politehnica University Timisoara, Romania
\textsuperscript{2} Romanian Academy – Timișoara Branch, Romania
romeo.resiga@upt.ro

Abstract. We introduce a new benchmark test case for assessing the robustness and accuracy of various numerical methodologies for simulating swirling flows in pipes and step diffusers. First we present the design and detailed geometry of the swirl generator and downstream test section. The swirl is generated with fixed plane blades, parallel to the symmetry axis, having a lean angle with respect to the meridian half-plane, in a mildly convergent pipe. The swirling flow is further ingested by a step diffuser with two cylindrical segments of 100 mm and 120 mm in diameter, respectively. This particular setup has a relatively simple geometry and it produces a swirl with pronounced axial deficit in the central region. Second, we present the methodology for axisymmetric swirling flow computation which account for the blades via blade body forces defined such as the flow tangency condition is enforced. Third, LDV measurements are presented in two survey sections corresponding to the two cylindrical pipe segments, and the accuracy of the numerical results is assessed. We conclude that this test case is suitable to define a new application challenge for assessing the ability of various numerical methodologies to accurately reproduce the turbulent swirling flows, with particular relevance for quasi-three-dimensional design and optimization of turbo-machines.

1. Introduction
For the past decade our efforts have been focused both on exploring the fundamentals of the turbomachinery swirling flows and on developing and testing control and stabilization techniques. In doing so we have designed a specific swirl apparatus with a swirl generator [1] that includes both upstream stay vanes producing a free-swirl rotation and a downstream runner, rotating at runaway speed, that generates a total pressure deficit near the hub (where the runner blade works as a turbine) and a corresponding excess near the shroud (where the runner blade works as a pump). This particular design insured that when the swirling flow evolves further downstream in a convergent-divergent test section [2] it will develop the self induced instability with precessing helical vortex specific to the discharge cone of Francis turbines operated at partial discharge [3]. Both the geometry of the swirl apparatus and the experimental data for LDV measurements on three survey sections and for unsteady pressure on the conical diffuser wall are public (https://openfoamwiki.net/index.php/Sig_Turbomachinery_/Timisoara_Swirl_Generator) for testing various numerical models and codes on a challenging three-dimensional unsteady flow problem. This experimental facility was replicated by another research group [4] and led to remarkable new experimental findings pertaining to vortex filament dynamics.

However, for preliminary design and optimization of hydraulic turbo-machines, or for exploring the potential benefits of novel technologies aimed at providing flexible, efficient and smooth operation, simple models that capture the main flow characteristics are still required. For example, axisymmetric swirling flow models that account for the stationary and rotating blades in a simplified
approach (the so-called through-flow methods) have the ability to quickly assess various design options and/or development in comparison with the full 3D unsteady flow analysis. Even if ultimate accuracy is not the selling point of such methods, certain robustness must be insured for reliable use. In particular, a robust modeling and computation of turbulent axisymmetric swirling flow is still necessary, in spite of numerous efforts to tackle this apparently simple problem.

The present paper introduces a novel swirl apparatus developed at the Politehnica University Timișoara, using a simple yet not trivial geometry of the swirl generator. Moreover, we opted for a step diffuser instead of a conical one in the present investigations, although the geometry of the test section could be easily modified while keeping the same swirl generator. Experimental data and numerical computations are presented in order to underline the open issues in numerical modeling.

Section 2 of the paper presents the geometry of the swirl apparatus, Section 3 summarizes the basic numerical procedures employed, and Section 4 presents a comparison of numerical results against experimental data. Section 5 summarized the conclusions.

2. The swirling flow apparatus

The swirling flow apparatus includes an upstream swirl generator and a downstream test section. The hydraulic passage shown in figure 1 has an inlet annular section with pipe diameter of 150 mm and a hub diameter of 111.6 mm, where the incoming flow from a reservoir has a typical velocity of 4 m/s. Straight blades, with a lean angle, are extended from the annular cylindrical part to the convergent section of the pipe, such that the combination between the radial velocity towards the axis and the lean angle generates circumferential velocity resulting in a swirling flow. This swirl further evolves in a step diffuser with a first cylindrical section of 100 mm in diameter and 105 mm in length, followed by a second cylindrical part of 120 mm in diameter and 140 mm in length. Finally, the flow discharges into a downstream reservoir through a 160 mm diameter pipe. Two survey sections, equipped for both non-invasive velocity and unsteady wall pressure measurements are located at 45 mm downstream the beginning of the cylindrical test sections. The overall length of 520 mm shown in figure 1 is actually used for the computational domain, as the 160 mm diameter pipe extends further downstream.

![Figure 1. Meridian cross-section of the swirl apparatus. The flow goes from left to right.](image)

A three-dimensional view of the swirl apparatus is shown in figure 2. One can notice the 36 leaned blades of the swirl generator, as well as the optical windows and the pressure sensors installed at both survey sections. The picture of the actual swirl apparatus, installed in a closed-loop test rig, with the LDV system on is also shown in figure 2. The pipe and hub profiling of the convergent section is omitted from this paper, but it is available on demand.
The swirl generator blades inserted into the hub are shown in figure 3, while the cross-section geometry of the bladed region is shown in figure 4. Note that although there are straight blades, the lean angle between the blade and the meridian plane varies with the radius, $\lambda(R)$. The blade lean is therefore better defined using the virtual tangent cylinder with radius $R_*=39.46\,mm$. As a result, the lean angle can be computed as

$$\tan \lambda(R) = \frac{R_*}{\sqrt{R^2 - R_*^2}}. \quad (1)$$

At the hub radius $R_{\text{hub}} = 55.8\,mm$ the lean angle is $\lambda_{\text{hub}} = 45^\circ$, to equation (1), while at the pipe wall, $R_{\text{pipe}} = 75.0\,mm$, the lean angle decreases to $\lambda_{\text{pipe}} = 31.74^\circ$. Obviously, in the convergent part of the pipe the lean angle is computed at any radius in the bladed region using equation (1).
3. The axisymmetric swirling flow computation

The so-called throughflow method, which stands for axisymmetric turbomachinery calculations, [5] remains the backbone of modern turbomachinery design process [6]. It can be applied though the whole hydraulic passage of general radial-axial turbomachines, including bladed and bladeless regions [7], or to specific sections of interest such as the trailing edge of the runner blades in a Francis turbine [8]. Depending on the solver employed, the flow turning in the bladed regions is modeled either via a kinematic flow tangency condition [7] or using a blade body force [9] as a source term in the momentum equations such that the absolute or relative flow tangency on the blade surface is achieved. Since for the present numerical simulations we are using the axisymmetric flow solver from the FLUENT 16.2 expert code, a blade body force is added as a source term in the momentum equations in order to account for the flow turning in the bladed region. This model is particularly well suited for large number of blades, as it is the case here, when the infinite number of blades assumption is reasonably reproduced.

The axisymmetric swirling flow equations, as employed in the Finite Volume solver of FLUENT can be summarized as

$$\frac{\partial (\beta \rho)}{\partial t} + \nabla \cdot (\beta \rho \mathbf{V}) = 0,$$

$$\frac{\partial (\beta \rho \mathbf{V})}{\partial t} + \nabla \cdot (\beta \rho \mathbf{V} \mathbf{V}) = -\beta \nabla p + \beta \rho \mathbf{f}_B + \text{viscous terms},$$

where $\beta(R) = 1 - \frac{N_{\theta} \delta_{B}}{2\pi R_s}$ is the blade blockage coeff.

The geometrical parameters for the present swirl generator, see figure 3, are $N_B = 36$ , $\delta_B = 1$ mm , and $R_s = 39.46$ mm, resulting in $(N_B \delta_B)/(2\pi R_s) = 0.1452$. The derivation of blade blockage coefficient is given in Appendix A. The above equations are written in cylindrical coordinates for the axial, $V_z$, radial, $V_r$, and circumferential, $V_\theta$, velocity components. Since the blades are parallel to the symmetry axis, there will only be radial and circumferential components for the blade body force $\mathbf{f}_B$,

$$f_{B\theta} = -V_z \frac{\partial V_r}{\partial Z} + V_r \frac{\partial V_z}{\partial R} - \frac{V^2}{R} \tan \lambda \tan \lambda, \quad f_{Br} = f_{B\theta} \tan \lambda \quad \text{and} \quad f_{Bz} = 0.$$  

Note that the circumferential velocity component is computed from the flow tangency condition on the blades,

$$V_\theta = -V_r \tan \lambda(R).$$

The full derivation of the blade body force given in Appendix B, showing how the circumferential body force acceleration, $f_{B\theta}$, is obtained by evaluating the circumferential projection of the left-hand side of the momentum equation (2) with the flow tangency condition (4) embedded.

The source term in the momentum equation that accounts for the blade body force is implemented via a User Defined Function (UDF) given in the Appendix C.

The inlet boundary condition corresponds to a discharge (axial) velocity of 3.8 m/s, for a volumetric flow rate used in the experiments of 0.03 m$^3$/sec. On the outlet section a radial equilibrium condition for the pressure is prescribed.

An important issue in axisymmetric swirling flow computations is the turbulence model. FLUENT offers a plethora of turbulence models [10], including for axisymmetric swirling flow computations. However, it seems that some of them have implementation issues. For example, although the vanishing radial and circumferential velocity is a mandatory condition in the axis, some models lead to finite (and wrong) circumferential velocity for vanishing radius. Moreover, the computed results are surprisingly widespread when simply switching from one turbulence model to another, most of them in flagrant disagreement with the experimental data. The comparison between the RNG $k-\varepsilon$ model...
and the Reynolds stress model, both implemented in Fluent, for swirling pipe flows simulation [11] conclude that the RNG $k-\varepsilon$ model better agrees with experimental data, although both models are finally judged as rather inadequate for developing pipe flows with swirl. Other investigations attempt to modify the plain $k-\varepsilon$ model in order to predict the axisymmetric swirling flow in the discharge cone of hydraulic turbines operated at part load [12]. It not the purpose of this paper to present a comprehensive survey of the literature on turbulence modeling for axisymmetric swirling flows, but we consider that identifying a robust, reliable and accurate model to be used in hydraulic machines throughflow computations is an open challenge awaiting for a proper answer.

We found that the standard $k-\varepsilon$ turbulence model, with enhanced wall function, provided the numerical results in best agreement with experimental data for axial and circumferential velocity profiles. Thereby, we present only these results in the next section. The numerical computation have been performed with unsteady axisymmetric swirling flow solved, and coupled velocity-pressure fields. Although the unsteady solver is employed, the solution reaches a steady configuration. Note that the investigated swirling flow does not present significant instabilities in the form of a distinct precessing helical vortex. There are only mild fluctuations, not presented in this paper, associated with an eccentric straight vortex filament near the axis.

4. Numerical results and comparison with experimental data
Numerical results presented in this section are obtained using the Fluent 16.2 expert code, with the unsteady axisymmetric swirling flow solver and the standard $k-\varepsilon$ turbulence model, implemented and tested in [14]. The computational domain corresponds to the meridian half-plane geometry as shown in figure 1, with boundary conditions given in Section 3. The momentum equations are altered by inserting a source term in the radial and circumferential projections using the UDF code given in the Appendix C, and by specifying the “porosity” through the blade blockage coefficient.

Figure 5. The flow direction at the trailing edge of the blades, according to equation (4). The solid line is $\tan(\lambda)$, computed with equation (1). The circles are numerical data for the velocity components ratio $-V_\theta/V_r$. Numerical simulations are performed without and with the blade body force activated. When the body force source terms are activated, one expects that the flow tangency condition on the blades, equation (4), be satisfied. In order to check this we plot in figure 5 the numerically obtained data for $-V_\theta/V_r$ at the blade trailing edge, together with the corresponding theoretical values from the blade geometry, equation (1). One can see that the blade body force correctly enforces the flow tangency condition. The small discrepancy near the hub can be attributed to the flow detachment from the hub at the blade trailing edge.

Figure 6 shows the streamlines and the total pressure map without accounting for the blades, in the lower half plane, and with blade body force source terms, in the upper half plane. When there is no swirl, the flow detaches at the junction between different diameter sections, with a long reattachment length. The first swirl effect is to shorten the reattachment length due to the centrifugal force, as expected. In both cases the flow detaches from the hub immediately after leaving the upstream annular segment. While the no-swirl flow has a velocity deficit near the axis in the wake of the hub, this velocity deficit is much more pronounced in the presence of the swirl, eventually leading to a
recirculation region. The total pressure map, in conjunction with the streamline pattern, reveals the evolution of the main flow (large total pressure) distinct from detachment and recirculation regions.

![Diagram of total pressure map and streamline pattern](image)

**Figure 6.** Meridian flow field without (lower half plane) and with (upper half plane) blade body force.

The comparison between the measured and computed velocity profiles in the two survey sections from figure 1 are presented in figures 7 and 8. On the survey section of the 100 mm diameter pipe, figure 7, the numerical data for the axial velocity are in very good agreement with the measurements, except a flattening of the velocity profile near the wall; the wake velocity distribution is correctly predicted. In the wake region, with axial velocity deficit, there is a solid body rotation of the flow from the axis up to a radius of approximately 30 mm. In the remaining annular section up to the pipe wall, with 50 mm radius, i.e. in the main stream, the computed circumferential component under-predicts the measured values. This is most probably the result in the simplifications assumed when deriving the blade body force expressions (3). For example, the boundary layers on the blades, which effectively thicken the blade through the displacement thickness, are not accounted for. Moreover, the blade wakes mixing downstream the trailing edge is also not accounted for, so far. Once again, the numerical results are obtained with the standard $k-\varepsilon$ turbulence model and other models show either worse agreement with experimental data, or are simply wrong.

In the survey section of the 120 mm diameter pipe, figure 8, the axial velocity deficit in the wake is still correctly predicted, but the numerical results overshoots the experiment close to the wall, in the main stream. The computed circumferential velocity under-predicts now even the solid-body rotation in the wake region most likely due to the over-diffusive characteristics of the standard $k-\varepsilon$ turbulence model. A first attempt to refine the grid (from 85631 cells to 342524, i.e. each cell split in four) did not change the solution numerical solution. Further investigations should aim at clarifying these discrepancies.

5. Conclusions
The paper presents a new swirl apparatus with a simple geometry that can be easily reproduced in a computational environment. Although details of the geometry are provided, the full geometry information can be available on demand.

The large number of blades used in the swirl generator makes this setup particularly suited for axisymmetric swirling flow modeling. The blade body force expression is derived for the particular blade geometry, and the source code of the user-defined function is included. Numerical results obtained with an axisymmetric swirling flow solver and the standard $k-\varepsilon$ turbulence model are compared against experimental data for axial and circumferential velocity profiles.

Testing other turbulence models, available in the Fluent expert code, led us to the conclusion that the results are widespread, most of the time in severe disagreement with experiments or even simply
wrong due to the inconsistent implementation of the axis boundary conditions. As a result, this issue should be further clarified since simple, robust and accurate axisymmetric swirling flow models are required for practical applications development. It is our hope that by defining this new benchmark test case, other research groups could contribute to such developments.

**Figure 7.** Axial and circumferential velocity components in survey section 1 in figure 1.

**Figure 8.** Axial and circumferential velocity components in survey section 2 in figure 1.
Acknowledgements

The authors affiliated with the Romanian Academy – Timișoara Branch have been supported by the research program „Unsteady Hydrodynamics of Helical Vortex Flows”. The design and manufacturing of the test section, as well as the LDV investigations, were supported by the project PN-II-ID-PCE-2012-4-0634. The first author was also supported by a mobility grant of the Romanian Ministry of Research and Innovation, CNCS-UEFISCDI, project number PN-III-P1.1-MC-2018-2568, within PNCDI III.

Appendix A. The blade blockage coefficient

The swirl generator has \( N_B \) blades of constant thickness \( \delta_B \). At a radius \( R \) the blades block a fraction of the flow passage, and the corresponding blockage coefficient is defined as

\[
\beta(R) = 1 - \frac{N_B \delta_B(R)}{2\pi R}.
\]

where the blade thickness in the circumferential direction is

\[
\delta_b(R) = \delta_B / \cos \lambda(R).
\]

Using the equation (1) the cosine of the lean angle is obtained as

\[
\frac{1}{\cos \lambda(R)} = \sqrt{1 + \tan^2 \lambda(R)} = \sqrt{1 + \left( \frac{R^2 - R_0^2}{R^2 - R_c^2} \right)} = \frac{R}{\sqrt{R^2 - R_c^2}}
\]

and finally the blade blockage coefficient has the expression

\[
\beta(R) = 1 - \frac{N_B \delta_B}{2\pi \sqrt{R^2 - R_c^2}} = 1 - \frac{N_B \delta_B \tan \lambda(R)}{2\pi R_c}.
\]

The dimensionless coefficient \( N_B \delta_B / (2\pi R_c) \) encapsulates all the geometrical parameters of the blades. The first expression in equation (8) is also valid for radial blades, when \( R_c \) vanishes and the blade blockage coefficient becomes \( \beta(R) = 1 - \frac{N_B \delta_B}{(2\pi R)} \).

Appendix B. The blade body force

The blade body force acceleration \( \vec{f}_B \) is computed from the convective acceleration, taking into account the flow tangency condition (4). Expressing the momentum equations in cylindrical coordinates gives the circumferential component of the blade body force as

\[
f_{B\phi} = V_r \frac{\partial V_\phi}{\partial Z} + V_\phi \frac{\partial V_r}{\partial R} + \frac{V_r^2}{R} - \frac{V_\phi}{R}.
\]

with \( V_r \), \( V_\phi \) and \( V_\theta \) the axial, radial and circumferential components of the superficial velocity in porous formulation. Inserting \( V_\phi(Z,R) = -V_r (Z,R) \tan \lambda(R) \) in equation (9) gives

\[
f_{B\phi} = -\left[ V_r \frac{\partial V_\phi}{\partial Z} + V_\phi \frac{\partial V_r}{\partial R} + \frac{V_r^2}{R} \left( 1 + \frac{R}{\tan \lambda} \frac{\partial \tan \lambda}{\partial R} \right) \right] \tan \lambda.
\]

Using equation (1), we have

\[
\frac{\partial (\tan \lambda)}{\partial R} = \frac{d}{dR} \left( \frac{R_{c_{\phi}}}{\sqrt{R^2 - R_0^2}} \right) = \frac{RR_c}{(R^2 - R_c^2)^{3/2}} = \frac{R}{R^2 - R_c^2} \tan \lambda,
\]

and as a result the expression in the round brackets of equation (10) becomes

\[
1 + \frac{R}{\tan \lambda} \frac{\partial \tan \lambda}{\partial R} = 1 - \frac{R^2}{R^2 - R_0^2} = -\frac{R_c^2}{R^2 - R_c^2} = -\tan^2 \lambda.
\]

As a result we recover the expression for \( f_{B\phi} \) in equation (3), and \( f_{B\phi} \) follows from the condition that the blade body force be normal to the blade surface, [5].
Appendix C. The User Defined Function implementation

The commercial code developers usually provide a facility that allows customization of the available solvers [13] using the so-called User Defined Functions (UDF). Such a UDF is employed in the paper to implement the source terms in the circumferential and radial momentum equations, as listed above.

The code below includes two procedures. The first one, named \texttt{blade\_body\_force}, computes the source terms in the momentum equations. The axial and radial velocity components are recovered via the functions \( C_U(c,t) \) and \( C_V(c,t) \), respectively, and the gradient components for the radial velocity are obtained as \( C_V_G(c,t)[0] \) and \( C_V_G(c,t)[1] \), respectively. The fluid density is obtained as \( C_R(c,t) \). The UDF code is self-explanatory and it can be easily followed in conjunction with equations (3) for the blade body force components, and the momentum equation from (2). Note the last two lines that actually select the circumferential and radial momentum equation projections for the corresponding source terms. The second one, named \texttt{blockage}, is a function that sets the blade blockage coefficient for “porosity” in the porous flow computation dialog box.

Both procedures are activated only in the bladed region, shown in figure 1.

```c
#include "udf.h"
#define RT 0.03945656 /* tangent circle radius */
#define NB 36 /* number of blades */
#define TN 0.001 /* blade thickness */
#define TWOPI 6.28318530717959 /* 2*PI */

DEFINE_SOURCE(blade_body_force,c,t,dS,eqn) {
  real x[ND_ND],R,DENS,TANLEAN,VA,VR,DVRDZ,DVRDR,BBFA,BETA,DELTA;
  Delta=NB*TN/TWOPI/RT; /* blade blockage parameter */
  C_CENTROID(x,c,t); /* get cell centroid radius */
  TANLEAN=RT/sqrt(R*R-RT*RT); BETA=1.0-Delta*TANLEAN;
  VA = C_U(c,t); /* get axial velocity */
  VR = C_V(c,t); /* get radial velocity */
  DVRDZ = C_V_G(c,t)[0]; /* dVr/dZ */
  DVRDR = C_V_G(c,t)[1]; /* dVr/dR */
  BBFA = VA*DVRDZ+VR*DVRDR-VR/R*TANLEAN*TANLEAN;
  DENS = C_R(c,t); /* get fluid density */
  BBFA = DENS*BBFA*TANLEAN;
  dS[eqn] = 0.0; /* explicit implementation */
  if(eqn==EQ_Z_MOM){return BBFA;} /* circumferential bbf */
  if(eqn==EQ_Y_MOM){return BBFA*TANLEAN;} /* radial bbf */
}

DEFINE_PROFILE(blockage,t,iv) {
  /* porosity of the media is defined as the ratio of the volume occupied by the fluid to the total volume */
  /* Thread *t - pointer to thread, int iv - variable index */
  real x[ND_ND]; /* cell centroid coordinates */
  real R; /* cell centroid radius */
  begin_c_loop(c,t) {
    C_CENTROID(x,c,t); /* get cell centroid coordinates */
    R=x[1]; /* get cell centroid radius */
    C_PROFILE(c,t,iv)=1.0-NB*TN/TWOPI/sqrt(R*R-RT*RT);
  }
  end_c_loop(c,t)
}
```
References

[1] Susan-Resiga R F, Muntean S, Tănăsă C and Bosioc A I 2008 Hydrodynamic design and analysis of a swirling flow generator 4th German-Romanian Workshop on Turbomachinery Hydrodynamics

[2] Bosioc A I, Susan-Resiga R F and Muntean S 2008 Design and manufacturing of a convergent-divergent test section for swirling flow apparatus 4th German-Romanian Workshop on Turbomachinery Hydrodynamics

[3] Bosioc A I, Susan-Resiga R F, Muntean S and Tănăsă C 2012 Unsteady pressure analysis of a swirling flow with vortex rope and axial water injection Journal of Fluids Engineering – Transactions of the ASME 134 081103

[4] Skripkin S G, Tsoy M A, Kuibin P A and Shtork S I 2017 Study of pressure shock caused by a vortex ring separated from a vortex rope in a draft tube model Journal of Fluids Engineering – Transactions of the ASME 139 081103

[5] Damle S V, Dang T Q and Reddy D R 1997 Throughflow method for turbomachines applicable for all flow regimes Journal of Turbomachinery 119 256

[6] Horlock J H and Denton J D 2005 A review of some early design practice using computational fluid dynamics and a current perspective Journal of Turbomachinery 127 5

[7] Bosman C and El-Shaarawi M A I 1977 Quasi-three-dimensional numerical solution of flow in turbomachines Journal of Fluids Engineering – Transactions of the ASME 99 132

[8] Susan-Resiga R F, Muntean S and Popescu C 2016 Swirling flow computation at the trailing edge of radial-axial hydraulic turbines 28th IAHR Symposium on Hydraulic Machinery and Systems – IOP Conf. Series Earth and Environmental Science 49 082012

[9] Pacciani R, Rubechini F, Marconcini M, Arnone A, Cecchi S and Daccà F 2016 A CFD-based throughflow method with an explicit body force model and an adaptive formulation for the S2 streamsurface Proc. IMechE Part A: J Power and Energy 230 16

[10] ANSYS 2015 ANSYS Fluent Theory Guide Release 16.2

[11] Escue A and Cui J 2010 Comparison of turbulence models in simulating swirling pipe flows Applied Mathematical Modelling 34 2840

[12] Foroutan H and Yavuzkurt S 2016 An axisymmetric model for draft tube flow at partial load Journal of Hydrodynamics B 28 195

[13] ANSYS 2015 ANSYS Fluent Customization Manual Release 16.2

[14] Lilley D G and Rhode D L 1982 A Computer Code for Swirling Turbulent Axisymmetric Recirculating Flows in Practical Isothermal Combustor Geometries NASA CR3442