Influence of turbulent inlet conditions on the flow inside a bulb turbine draft tube using Large-Eddy Simulations

P Véras\(^1\), G Balarac\(^{1,2}\), O Métais\(^3\), D Georges\(^3\), A Bombenger\(^4\), C Séguin\(^\text{f}^4\)

\(^1\) Université Grenoble Alpes, CNRS, Grenoble-INP, LEGI, 38000 Grenoble, France
\(^2\) Institut Universitaire de France (IUF)
\(^3\) Université Grenoble Alpes, CNRS, Grenoble-INP, GIPSA-lab, 38000 Grenoble, France
\(^4\) GE Renewable Energy, 82 Avenue Léon Blum, 38100 Grenoble, France

E-mail: pedro.veras@univ-grenoble-alpes.fr

Abstract.

The head losses inside the draft tube of a bulb turbine can represent an important portion of the total energy losses due to the low heads at which these machines operate. However, the complexity of the flow makes it a numerical challenge to simulate. Previous studies have shown that Large Eddy Simulations (LES) based on mean inlet conditions improve flow prediction inside the draft tube’s cone but diverge further downstream [4]. Moreover, uncertainties and the lack of detailed experimental data at the inlet have been identified as the main reason for these discrepancies and precise measurements close to the walls are necessary to correctly validate the numerical simulations [8, 4]. Thanks to detailed experimental measurements, the objective of this paper is to enhance the accuracy of previous results by performing LES computations to numerically simulate the flow inside the draft tube of a bulb turbine, and to investigate the influence of inlet conditions using an innovative approach. A two-criteria based mesh adaptation along with an element-wise masking strategy is used to assure a good spatial discretization level while reducing the computational cost. Partial experimental data imposed at the inlet are often not sufficient to achieve a proper downstream flow prediction with a LES. The real challenge thus consists in the economical generation of proper mean and fluctuating inlet flow fields. We first show that a simple homogeneous and isotropic synthetic turbulence field added to the mean experimental profiles may improve the prediction of the downstream flow, but this is only achieved through empirical adjustments. Therefore, we also investigate the use of machine learning procedure to automatically generate proper inlet mean and fluctuating fields.

1. Introduction

The draft tube of a hydraulic turbine is a divergent shaped equipment responsible for efficiently converting the residual kinetic energy leaving the runner into pressure, increasing the turbine’s effective head and enhancing its performance [1]. The head losses inside this equipment can represent an important portion of the total energy losses, specially in the case of bulb turbines, which operate at low heads. Therefore, correctly predicting the flow behavior inside the draft tube is very important in terms of competitiveness.

Performing numerical simulations are a good way of evaluating the flow inside a draft tube, specially because they are less expensive to conduct than experiments and give access to a larger
database. However, the complexity of the flow inside a draft tube makes it a real challenge to simulate. Compared to steady and unsteady Reynolds Averaged Navier-Stokes (RANS) simulations, Large-Eddy Simulations (LES), give more accurate results for the flow and head losses inside the draft tube [2, 3]. Recently, Wilhelm et al. [4] obtained a good agreement between numerical and experimental results and conducted a detailed energy balance analysis to identify the main head loss mechanisms inside the draft tube.

One difficulty pointed out by Wilhelm et al. [4], though, is to impose realistic inlet conditions on the numerical domain, either due to a lack of information coming from experiments and/or due to simplifications aiming to reduce the computational cost of the simulations. Nevertheless, they have a major impact on the flow behavior inside the draft tube [5–9] and should therefore be carefully specified. Moreover, turbulent inlet conditions also have an impact on the flow, but they are even more difficult to impose in unsteady draft tube simulations and sophisticated methods to generate the fluctuations have therefore to be employed [10, 11].

Starting from the results of a detailed experimental campaign conducted by GE Renewable Energy, LES computations were performed at LEGI (Laboratory of Geophysical and Industrial Flows) in order to investigate the flow behavior inside the draft tube of a bulb turbine. After presenting the numerical methodology, we study the influence of different turbulent inflow conditions and numerical domains on the flow behavior inside the draft tube. Finally, a simple diffuser case is considered to investigate the application of Artificial Neural Networks (ANN) as an optimization tool for inlet boundary conditions.

2. Numerical methodology

All LES computations are performed in YALES2 incompressible fractional-step solver using finite-volume formulation with 4th order space and time numerical schemes [12]. The $\sigma$ turbulence model [13] is used to model the effect of sub-grid scales (SGS) on the flow and the time step is calculated to ensure a CFL smaller than 0.9 in the whole domain. Simulations are run up to the velocity profiles stabilization of and flow statistics convergence.

2.1. Experimental configuration and numerical domain

The test-rig studied in this work is shown on Fig. 1. The draft tube geometry is similar to those used in real bulb turbines and identical to that studied in Wilhelm et al. [4]. The working fluid is air, but the Mach number is lower than 0.15 (incompressible flow) and the Reynolds number is of the order of $1.5 \times 10^6$. Since accurate measurements at the boundaries are crucial when evaluating head losses [4], guides vanes and runner are replaced by a grid and a set of 34 fixed blades upstream the draft tube, reducing thus uncertainties and improving flow characterization. Experimental results are available at six locations in the test-rig, as shown in Fig. 1. Five-holes pressure probes measure velocity and pressure profiles from the wall all the way up to the center of the flow, while hot-wires give accurate velocity magnitude and turbulence intensity distribution in the near-wall region, since accurately predicting the flow behavior in this location is essential to evaluate the head losses inside a draft tube [9].

![Figure 1. Test-rig domain.](image1)

![Figure 2. LES domains: (a) non-extended; (b) extended.](image2)
The first numerical domain considered in this work is shown in Fig. 2(a) and consists in the draft tube portion of test-rig, including a small part of the hub, and a straight extension positioned at its exit, which reduces the influence of the outlet conditions on the flow inside the draft tube. The second domain, shown in Fig. 2(b), is used when investigating the influence of the synthetic turbulence field and is identical to the first one, except for a straight extension positioned at its inlet.

2.2. Mesh adaptation strategy

Mesh dependency studies are conducted using a two-criteria based automatic adaptation strategy proposed by Benard et al. [14] and implemented in YALES2 through MMG3D library [15]. The first criterion, $Q_{C1}$, is defined in Eq. (1) and minimizes the discretization error of the mean velocity gradients to ensure the correct resolution of the LES mean velocity field.

$$Q_{C1} = \Delta^2 \max_{i=1,2,3} \left\{ \left| \frac{\partial^2 u_i^*}{\partial x_i^2} \right| \right\}$$

where $\Delta$ is the local mesh size, $u_i$ is the mean velocity field and $x_i$ the direction.

The second criterion, $Q_{C2}$, is defined in Eq. (2) and ensures that more than 80% of the total turbulent kinetic energy is explicitly resolved, guaranteeing thus the validity of LES approach on the domain [16].

$$Q_{C2} = \frac{E_{sgs}}{E_{sgs} + E_R} \leq 0.2$$

where $E_{sgs}$ and $E_R$ are, respectively, the SGS and resolved turbulent kinetic energy.

Results are considered independent of the mesh after three adaptation steps. Figure 3 shows that most of the refinement occurs in the center region, where the swirl and hub create a large vortical structure and therefore high velocity gradients. A few layers of prismatic elements later transformed into tetrahedrons are used close to the walls to reduce the overall computational cost. Since velocity gradients are important close to the walls, a masking strategy is implemented to avoid adapting these elements. Three wall refinement levels (maximum $y^+$ equal to 70, 150 and 220) were investigated and Duprat et al. [17] wall-layer model was used. The impact of these different wall refinements was found to be negligible and the coarser grid was finally used. The final adapted mesh is composed by 8.7 million elements.

![Figure 3](image)

Figure 3. Detail of the automatic mesh adaptation process near the hub. (a) Original mesh; (b) $Q_{C1} = 20$; (c) $Q_{C1} = 10$; (d) $Q_{C1} = 5$.

2.3. Boundary conditions

Rotating phase-averaged velocity inlet conditions are commonly used in unsteady draft tube simulations [11, 4] since velocity distribution is far from uniform in azimuthal direction due to the small number of blades in the runner. To mitigate this effect, a large number of blades is used in the test-rig and, as a consequence, mean experimental axisymmetric velocity profiles (shown normalized by the averaged inlet axial velocity, $V_{b,in}$, in Fig. 4 and the non-dimensional height, $h/H$) are used as inlet conditions. Thanks to hot-wires measurements, we have access
to some information regarding the turbulence intensity at the draft tube’s inlet, which can then be used to estimate the turbulence kinetic energy, \( k \). Therefore, an important aspect of this work is the investigation of turbulent inlet conditions, a common difficulty in draft tube numerical simulations as this quantity is normally unknown. YALES2 has a built-in capability of generating and injecting a synthetic homogeneous isotropic turbulence (HIT) field to the inlet. The method, described in Fig. 5, depends on four parameters: fluctuation levels, \( u' \), a characteristic length, \( l_e \), a convection (injection) speed, \( \bar{U} \), and a profile of \( k \). Synthetic HIT fields are generated in cubes with sides equal to \( 4l_e \) using a method similar to those explained in Kraichnan [18] and Smirnov et al. [19]. These cubes are later joined together to cover the whole injection area and the turbulence intensity distribution is scaled based on the profile of \( k \). Finally, the synthetic turbulence is injected through the domain’s inlet at a velocity equal to \( \bar{U} \).

3. Influence of the incoming turbulence field

3.1. Non-extended domain

We first consider the flow simulation inside the domain shown in Fig. 2(a). Comparisons of the normalized mean axial velocity, \( V_z \), and turbulence intensity, \( Tu \), profiles are shown in Fig. 6 and Fig. 7, respectively. Results of simulations without any synthetic turbulence show that the grid adaptation strategy is very effective in capturing the flow characteristics of the central vortex (near \( h/H = 1 \)). However, starting at station R3, the boundary layer thickens faster than expected and, as a consequence of the lower \( V_z \) in this region, the axial velocity increases in the intermediate portion of the flow. These are characteristics of laminar flows and so we inject a simple synthetic HIT field, as explained in section 2.3. At \( Tu \) levels comparable to the experiments, influence on \( V_z \) is found to be negligible, probably due to the rapid dissipation of the synthetic turbulence downstream. Increasing \( Tu \) improves the results close the walls but has little effect on the intermediate region of the flow. Finally, if \( l_e \) is also increased, \( V_z \) results in the intermediate region are improved, but the excessive mixing quickly dissipates the central vortex and the wall boundary layer is too thin.

![Figure 4. Normalized inlet profiles.](image)

![Figure 5. Scheme of the synthetic HIT field injection method.](image)

![Figure 6. Normalized mean axial velocity profiles using non-extended domain.](image)
Figure 7. Turbulence intensity profiles using non-extended domain.

Figure 8 shows the evolution of total pressure distribution inside the draft tube and normalized by the dynamic pressure, $q$, at the inlet. Cases without turbulence or too low $T_u$ present the same discrepancies near the walls and in the intermediate region as discussed in Fig. 6. Increasing $T_u$ and $l_e$ allows to improve the shape of the total pressure profile, but also shifts it to lower values as compared with the measurements.

Figure 8. Normalized total pressure profiles using non-extended domain.

3.2. Extended domain

More sophisticated synthetic turbulence generation methods, such as Smirnov et al. [19], Davidson [20] and Jarrin [21] try to recreate spatial and temporal correlations. However, when the real structure of the turbulent eddies is not accurately reproduced, a transition region is observed before the synthetic turbulence redevelops to a more physical state [21]. The problem is that even for fairly simple flows, this transitional region can correspond to a large portion of the numerical domain [22]. To overcome this difficulty in our simulations, we add a straight extension upstream of the real draft tube inlet (see Fig. 2(b)). The same axisymmetric boundary conditions, including synthetic turbulence characteristics, are imposed at this new upstream inlet position. The difference is that the extension provides more space for the synthetic turbulence to develop before entering the draft tube. Figure 9 shows the comparison of normalized total pressure evolution inside the draft tube using both numerical domains.

The positive effect of injecting a synthetic turbulence field is still present, both close to the walls and in the intermediate region of the flow. Furthermore, adding an extension has a significant effect on the total pressure shifting observed in Fig. 8. Numerical results are now at the same level as the experimental measurements and the turbulence intensity profiles shown in Fig. 10 are found to be closer to experiments at the first stations inside the draft tube.

Finally, Figure 11 compares the normalized mean axial velocity profiles obtained using both domains. Similar to total pressure results, the positive effect of adding a synthetic fluctuation
field is still present, i.e., increased speed close to the walls and improvements in the intermediate region. Although the addition of a fictitious inlet extension helps for the development of the synthetic turbulence field, it however modifies the mean flow and thus the inlet conditions at the real inlet of the draft tube, as shown in Fig. 12. To remedy this defect, we investigate the use of machine learning algorithms as an alternative to adjust the inlet conditions at the fictitious inlet to obtain the proper experimental profiles at the real draft tube inlet.

4. Application of ANNs
We now investigate the use of Artificial Neural Networks as an optimization tool for inlet boundary conditions. Several works have been devoted to draft tube geometry optimization [23–26]. Galvan et al. [27] applied Evolutionary Algorithms (EAs) to optimize inlet conditions...
and Taheri [11] used ANNs to rebuild the turbulence field at the draft tube's inlet of a non-
extended domain. ANN applications to LES are limited because they require a fair amount of
different simulations obtained by varying the flow parameters to yield proper results. However, a
recent work investigated the use of sophisticated neural networks to generate synthetic turbulent
inflow conditions in a Direct Numerical Simulation (DNS) [28].

4.1. Case study
To demonstrate the viability of the ANN approach to optimize the inlet flow conditions in a
draft tube, we here consider a simplicity geometry consisting in a simple conical diffuser previously
studied by Clausen et al. [29]. We furthermore choose the RANS (Reynolds Averaged Navier-
Stokes) approach which allows multiple flow simulations at a reduced cost as compared with
LES. The flow geometry consists in a 0.51 m long conical diffuser, with a 0.26 m inlet diameter
and a 20° opening angle. The working fluid is air and a rotating honeycomb screen upstream
the diffuser creates a swirl that prevents the boundary layer from detaching. Other works used
this case to test their numerical methodology before simulating actual draft tubes [7, 30, 31]
and Duprat et al. [32] investigated the effect of turbulent inlet conditions using LES.

4.2. Methodology
RANS simulations are performed with ANSYS CFX 18.1 using $k - \omega$ SST turbulence model
[33]. Similarly to the previous section, two flow domains are considered: a first one with no
inlet extension and a second one with an upstream flow extension (Fig. 13). An average static
pressure equal to 0 is imposed at the outlet and no-slip conditions with automatic wall treatment
[34] are imposed at the walls. A block-structured hexahedral mesh is used and $y^+$ values are
lower than 2. Geometry axisymmetry is used.

The ANN is implemented in Python 3.7 using TensorFlow library. Its structure consists of
four Multilayers Perceptrons (MLPs), one for each velocity component plus another for $k$. The
number of layers and neurons depends on the profiles considered, the number of points used to
describe them and the hyper-parameters used to control the training. All cases use the Mean
Squared Error (MSE) as loss function however.

4.3. Results and discussion
The main objective consists in using the ANN approach to generate the proper mean velocity
profiles at the upstream inlet of the extension allowing for a correct prediction of the downstream
mean field development in the diffuser. In a first phase, ANN have to be trained. During
the training, ANN inputs are mean axial $V_z$ and tangential $V_u$ velocity profiles from RANS

Figure 12. Normalized mean velocity profiles at IN station.

Figure 13. Numerical domains used in the conical diffuser study: (a) non-extended; and (b) extended.
simulations with inlet extension at all eight stations inside the diffuser (S0 to S7) and outputs are the corresponding velocity and turbulent kinetic energy profile at the extension’s inlet. A database of 300 simulations is used to train the ANN. In a second phase, once training is achieved, the experimental measurements at the eight station S0 to S7 are used as inputs and the ANN returns the best inlet conditions that should be imposed at the extension. The corresponding predicted inlet conditions at the beginning of the extension are shown in Fig. 14. These are compared with the corresponding inlet conditions for the RANS simulation without extension. Note that, in this latter case, only the mean axial and tangential velocity components are available and the radial component profile is reconstructed using Payette’s [7] methodology. $\epsilon$ is obtained using Armfield’s et al. [35] method.

Figure 14. Inlet conditions imposed at both conical diffuser domains (with and without inlet extension).

Comparisons of the simulations using ANN inlet conditions and the non-extended case are shown in Figs. 15 (axial velocity profiles) and 16 (tangential velocity profiles). Both extended and non-extended results agree very well with the measurements, indicating that the ANN method is a very promising tool to predict proper inlet conditions at the upstream extension.

Figure 15. Comparison between the experimental and numerical mean axial velocity profiles.

Figure 16. Comparison between the experimental and numerical tangential velocity profiles.
5. Conclusions
In this paper, the impact of inlet boundary conditions on the flow inside the draft tube of a bulb turbine has been investigated using Large-Eddy Simulations. Different turbulent inlet conditions have been tested and we analyzed their influence on the accuracy of the numerical results as compared to experimental data. An automatic mesh adaptation strategy ensured a good flow resolution near the central region of the draft tube while Duprat’s et al. [17] wall-layer model was used to reproduce the flow behavior close to the walls.

Initial simulations, imposing only mean axisymmetric velocity profiles as inlet conditions, resulted in a fast thickening of the boundary layer and an excessive axial $V_z$ velocity in the intermediate region of the flow. Injecting a simple HIT field improved mean axial velocity results, but only when its intensity was significantly higher than experimental measurements. Moreover, total pressure results were compromised by intense synthetic turbulence fields, as they were shifted to lower values when compared with the experiments. These issues could be related to the development of the synthetic turbulence inside the numerical domain and thus an extension was added upstream of the inlet to give it more space to develop before entering the draft tube. The extension improved total pressure results while keeping positive effects near the walls and in the intermediate region. However, the addition of a fictitious inlet extension also modified mean flow and thus the velocity profiles at the real draft tube’s inlet.

The ongoing part of this study is the investigation of ANN as an optimization tool to define the correct boundary conditions to be imposed at the fictitious inlet to obtain a good match with the experimental observations downstream in the diffuser. A simplified study using Clausen’s [29] conical diffuser and RANS simulations has been conducted, using an ANN composed by four MLPs, one for each mean velocity component and for the turbulent kinetic energy $k$. A database consisting on the results of 300 simulations was used to train the network and the numerical results agreed very well with a reference non-extended case and the experiments, despite the presence of an extension upstream. The present results are very promising and the next step will consist in applying this machine learning methodology to the LES of bulb turbine draft tube.

Acknowledgments
The authors would like to thank the Chaire Hydro’Like for funding P. Véras and GENCI (Grant 2A00611) for providing the computing resources.

References
[1] Gubin M F 1973 Draft tubes of hydroelectric stations. (Published for the Bureau of Reclamation, US Dept. of the Interior and National Science Foundation, Washington, DC by Amerind Pub. Co.)
[2] Gehrer A, Benigni H and Köstenberger M 2004 Unsteady Simulation of the Flow Through a Horizontal-Shaft Bulb Turbine Proceedings of the 22nd IAHR Symposium on Hydraulic Maschines and Systems, Stockholm, Sweden
[3] Jošt D and Škerlavaj A 2014 Efficiency prediction for a low head bulb turbine with SAS SST and zonal LES turbulence models IOP Conference Series: Earth and Environmental Science 22 (IOP Publishing)
[4] Wilhelm S, Balarac G, Metais O and Séguoin C 2016 Analysis of head losses in a turbine draft tube by means of 3D unsteady simulations Flow Turbul Combust 97 1255-80
[5] De Henau V, Payette F A, Sabourin M, Deschénes C, Gagnon J M and Gouin P 2010 Computational study of a low head draft tube and validation with experimental data IOP Conf. Series: Earth and Environmental Science 12 (IOP Publishing)
[6] Guénette V, Houde S, Ciocan G D, Dumas G, Huang J and Deschénes C 2012 Numerical prediction of a bulb turbine performance hill chart through RANS simulations IOP Conf. Series: Earth and Environmental Science 15 (IOP Publishing)
[7] Payette F 2008 Simulation de l’écoulement turbulent dans les aspirateurs de turbines hydrauliques: impact des paramètres de modélisation Master’s thesis Laval University
[8] Brugiere O 2015 Fiabilité et évaluation des incertitudes pour la simulation numérique de la turbulence: application aux machines hydrauliques Ph.D. thesis Grenoble University
[9] Wilhelm S 2017 Étude des pertes de charge dans un aspirateur de turbine bulbe par simulations numériques instationnaires Ph.D. thesis Grenoble University
[10] Duprat C 2010 Simulation numérique instationnaire des écoulements turbulents dans les diffuseurs de centrales hydroélectriques en vue de l’amélioration des performances Ph.D. thesis Grenoble INP
[11] Taheri A 2015 Detached eddy simulation of unsteady turbulent flows in the draft tube of a bulb turbine Ph.D. thesis Laval University
[12] Moureau V, Domingo P and Vervisch L 2011 Design of a massively parallel CFD code for complex geometries CR Mécanique 339 141-8
[13] Nicoud F, Todd H B, Cabrit O, Bose S and Lee J 2011 Using singular values to build a subgrid-scale model for large eddy simulations Phys Fluids 23
[14] Benard P, Balarac G, Moureau, V, Dobrzynski C, Lartigue G and D’Angelo Y 2016 Mesh adaptation for large-eddy simulations of porous media Int. J. Numer. Meth. Fluids 81 719-40
[15] Dobrzynski C and Frey P 2008 Anisotropic Delaunay Mesh Adaptation for Unsteady Simulations Proceedings of the 17th International Meshing Roundtable, Pittsburgh, USA, 177-94
[16] Pope S B 2000 Turbulent Flows (Cambridge University Press)
[17] Duprat C, Balarac G, Metais O, Congedo P M and Brugiere O 2011 A wall-layer model for large-eddy simulations of turbulent flows with/out pressure gradient Phys Fluids 23
[18] Kraichnan R H 1970 Diffusion by a random velocity field The Physics of Fluids 13 22-31
[19] Smirnov A, Shi S and Celik I 2001 Random flow generation technique for large eddy simulations and particle-dynamics modeling J. Fluid Eng. 123 359-71
[20] Davidson L and Billson M 2007 Hybrid LES-RANS using synthesized turbulent fluctuations for forcing in the interface region Int. J. Heat Fluid Flow 27 1028-42
[21] Jarrin N 2008 Synthetic inflow boundary conditions for the numerical simulation of turbulence Ph.D. thesis University of Manchester
[22] Poletto R, Craft T and Revell A 2013 A new divergence free synthetic eddy method for the reproduction of inlet flow conditions for LES Flow Turbul Combust 91 519-39
[23] Eisinger R and Ruprecht A 2002 Automatic shape optimization of hydro turbine components based on CFD Task Quarterly 6 101-11
[24] Marjavaara B D and Lundström T S 2006 Redesign of a sharp heel draft tube by a validated CFD-optimization Int. J. Numer. Meth. Fluids 50 911-24
[25] Marjavaara B D, Lundström T S, Goel T, Mack Y and Shyy W 2007 Hydraulic turbine diffuser shape optimization by multiple surrogate model approximations of Pareto fronts J. Fluid Eng. 129 1228-40
[26] McNabb J, Devals C, Kyriacou S A, Murry N and Mullins B F 2014 CFD based draft tube hydraulic design optimization IOP Conf. Series: Earth and Environmental Science 22 (IOP Publishing)
[27] Galvan S, Reggio M and Guibault F 2015 Numerical optimization of the inlet velocity profile ingested by the conical draft tube of a hydraulic turbine J. Fluid Eng. 137 071102
[28] Fukami K, Nabae Y, Kawai K and Fukagata K 2019 Synthetic turbulent inflow generator using machine learning Phys. Rev. Fluids 4 064033
[29] Clausen P D, Koh S G, Wood D H 1993 Measurements of a swirling turbulent boundary layer Exp. Therm. Fluid Sci. 6 39-48
[30] Menter F R, Kuntz M and Langtry R 2003 Ten years of industrial experience with the SST turbulence model Turbulence Heat and Mass Transfer 4 625-32
[31] ANSYS® CFX Release 18.1 2018 ANSYS CFX Solver theory guide Ansys Inc., Canonsburg, PA, USA.
[32] Clausen P D, Koh S G, Wood D H 1993 Measurements of a swirling turbulent boundary layer Exp. Therm. Fluid Sci. 6 39-48
[33] Menter F R, Kuntz M and Langtry R 2003 Ten years of industrial experience with the SST turbulence model Turbulence Heat and Mass Transfer 4 625-32
[34] ANSYS® CFX Release 18.1 2018 ANSYS CFX Solver theory guide Ansys Inc., Canonsburg, PA, USA.
[35] Arnaud S W, Cho N H and Fletcher, C A J 1990 Prediction of turbulence quantities for swirling flow in conical diffusers AIAA 28 453-60