DESIGN AND CFD ANALYSIS OF CENTRIFUGAL COMPRESSOR FOR A MICRO GAS TURBINE

R. Aghaei and M. Soltani
Niroo Research Institute of IRAN

A. M. Tousi
Department of Aerospace Engineering, Amir Kabir University of Technology – Tehran, Iran

Abstract:

Micro turbine centrifugal compressors have been designed and simulated three-dimensionally using the commercial CFD-code FLUENT 6. A model only one impeller channel was used to compare 1D design data which obtained from CCD code, written and developed by authors. The numerical results in respect to performance data showed quite good agreement with experimental data (& 1D design) at and near the operating point of best efficiency. The often used model for turbomachinery, known as 'frozen-rotor'-model only yields satisfying results for efficiency and pressure ratio, at and near the point of best efficiency. For this case the static pressure shows a nearly uniform circumferential distribution at the inlet of the diffuser, which numerically leads to more homogenous flow rates through the single vane channels, and thus to a more realistic time- averaged flow distribution. Unsteady simulation for this purpose, i.e. modeling the rotor rotation relative to the diffuser, can be carried out and would certainly provide a flow field solution of higher quality. The intention of the paper is to show how good compressor can be designed and modeled with CFD steady models and to explain reasons for discrepancies between experiment (1D design) and CFD.

1. Introduction

The centrifugal compressor has very large field of applications in the compression of gases in the process industry there are innumerable combinations of applications, gases and pressure possible. The gases to be compressed may be air, oxygen, hydrocarbons, refrigerants or almost any other industrially used gas, whereas the pressure levels vary from 50 kpa for vacuum blowers up to 50 Mpa in barrel compressors and the pressure ratio may go up to 10.

Microturbines are small gas turbines. In a micro turbine, a centrifugal compressor compresses the inlet air that enters axially, but is then turned radially out from the impeller. The static pressure rises due to the decrease of the relative velocity, but also due to the centrifugal forces. After the impeller, the flow enters a radially diffuser. The compressor needs a gathering body, volute, to assemble the flow to the next stage or to the outlet piping.

The first part of the paper deals how the design of a centrifugal compressor starts with the definition of the initial parameters, which are used to calculate the one- dimensional flow in the corresponding program. This program, developed by authors, and test results from previous design to give preliminary results about the compressor performance.

The radial flow turbine- driven compressor is quite similar in terms of design and volumetric flow to automobile, truck, and other small reciprocating engine turbochargers. It is required, that the high efficiency range of compressor is as wide as possible [12].

In second part of paper three dimensional design and CFD analysis of centrifugal compressor will be described. The initial parameters of compressor impeller geometry and flow are used in the three-dimensional design process which is used to define the whole compressor wheel geometry.

The compressor flow is calculated with three-dimensional CFD- program (FLUENT 6). Results of CFD- calculation and one-dimensional calculations are compared, and the design procedure repeated, if necessary. The high complexity of the flow, especially in the rotating impeller, makes the CFD modeling very difficult. There has been done much research in analyzing the flow and the different viscous phenomena, but there are still no programs that can calculate the time dependent flow explicit [10].

2. One–dimensional flow design

The design of centrifugal compressor impeller usually assumes tow distinct stages. A preliminary design, making use of one-dimensional analysis based on previous experience, is sketched out to specify the inlet and outlet blade angles and the “skeletal” dimensions. This is followed by a detailed design in which the complete blade and channel geometry is specified and then subsequently refined by means of successive aerodynamic and stress analyses [9].

At first, the design team gathered data from literature and visited to other universities and research institute and was convinced that the design of a centrifugal compressor starts with the definition of the initial parameters, which are used to calculate the one-dimensional flow in the corresponding program. Gradually, we were able to have design parameters from our own developed code for centrifugal compressor design (CCD). This code uses empirical data from literature and test results from previous design (such Whitfield (1990) [7] and Japikse (1996) [8]) to give preliminary results about the compressor performance as well as some impeller and diffuser geometrical parameters.

The input data includes compressor pressure ratio, inlet pressure (both in total magnitude), rotating speed (rad/sec), and mass flow rate. The program includes variables that can later be altered: Specific speed, different geometrical ratios, slip factor, inlet flow angles, etc.

The main calculation results are:

_ Aerothermodynamics properties at different position of impeller wheel

_ Main geometry parameters, compressor specific speed and power consumption.

_ Isentropic Efficiency and Pressure ratio in each point, Pressure recovery in the diffuser.

_ Velocity triangles, Deviation angle and Relative Mach number at inlet and outlet.

This program able to calculate the off-design performance and produces the compressor performance parameters for further studies on the performance map.

2.1. Basic equations for one-dimensional design

A further important step is necessary before design optimization calculation can be conducted, that is a complete description of the flow process, on a one-dimensional basis, in term of the basic governing equations. The basic governing equations are presented here, but only for mixed out flow, from Japikse (1996) [8]. To compute the exit mixing process correctly, a control volume must be setup. This was a problem in the early Johnston and Dean (1965), and Dean, Wright and Runstadler (1970) derivations but was corrected in Japikse (1985). Using the proper control volume shown in Fig.1. [8]

Mixed-out state equations (m):

Momentum:

$$\int_0^\tau \left(\oint\!\!\oint_{c.v.} \rho(t)C(t)[C(t).dA] \right) dt = \int_0^\tau \sum F(t)dt$$

Mass:

$$\int_0^\tau \left(\oint\!\!\oint_{c.v.} \rho(t)C(t).dA \right) dt = m$$

Energy:

$$\int_0^\tau \left(\oint\!\!\oint_{c.v.} \Delta h_0 [\rho(t)C(t).dA] \right) dt = \sum_i P_i$$

Where: $C(t) = C_m(t)\hat{i}_m + C_\theta(t)\hat{i}_\theta$ and $dA = dxrd\ \theta\hat{i}_m$

And: P_i includes all parasitic power terms (-) and the overall shaft power (+)

Where $\hat{i}_m$ and $\hat{i}_\theta$ are unit vectors in the meridional and tangential directions.

Hence: $C(t).dA = \pm C_m rdAdx$

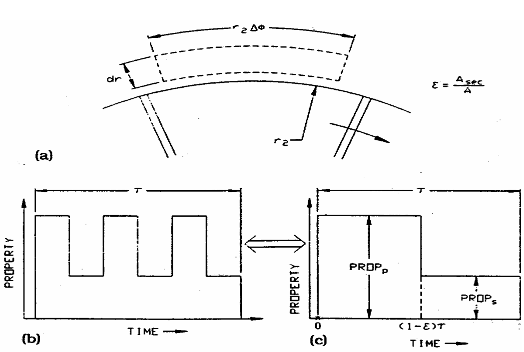

Fig.1: These figures are equivalent for classical tow-zone modeling (Japikse 1996) [8]

The control volume (see Fig.1.a) has an infinitesimal radial extent (dr), hence there are no shear stresses on the sides or ends of the control volume. The depth of the control volume is b_2 and gradients in the X- direction are not considered. Hence, integration is very, very small but finite angle. Hence there are no variations in the ϕ direction to be considered. However, the velocities are time-dependent entering the control volume (Fig.1.b and 1.c). Integration will be over a period τ and hence, for example, C_m is C_{m2p} during the time interval 0 to $(1-\varepsilon)\tau$ and it is C_{m2s} during the time interval $(1-\varepsilon)\tau$ to τ. Parameters on the r+dr side are space and time averaged by definition, hence yielding the so-called mixed-out state. The historical assumption of an unloaded tip is used, hence $p_{2s} = p_{2p} = p_2$. Thus we get:

$$\int_0^\tau \left(-\int_0^{\Delta\phi}\int_0^{b_2} \rho(t)C(t)C_m(t)r_2 d\phi dx + \int_0^{\Delta\phi}\int_0^{b_2} \rho_{2m}C_{2m}C_{m2m}r_2 d\phi dx \right) dt$$

$$= \int_0^\tau \sum F(t)dt$$

This reduces to:

$$m_p C_{m2p} + m_s C_{m2s} = mC_{m2m} + (p_{2m} - p_2)2\pi r_2 b_2$$

(*Radial Momentum*)

Using the same approach, the tangential component of the momentum equation becomes:

$$\rho_{2p}C_{\theta 2p}(C_{m2p}A_{2p}) + \rho_{2s}C_{\theta 2s}(C_{m2s}A_{2s}) - \rho_{2m}C_{\theta 2m}(C_{m2m}A_2) = 0$$

This reduces to:

$$m_p C_{\theta 2p} + m_s C_{\theta 2s} = mC_{\theta 2m} \ (Tangential\ Momentum)$$

Similarly the energy equation ([c] above) becomes:

$$m_p C_P T_{02p} + m_s C_P T_{02s} + (P_{disc\_friction} + P_{recirc}) = mC_P T_{02m}$$

(*Energy Equation*)

And the continuity equation ([b] above) becomes:

$$m = 2\pi r_2 b_2 C_{m2m}\rho_{2m} \ (Conservation\ of\ Mass)$$

$$T_{2m} = T_{02m} - C_{2m}^2 / 2C_P \ (Stagnation\ Temprature)$$

$$\rho_{2m} = p_{2m} / RT_{2m} \ (Equation of\ State)$$

$$C_{2m}^2 = C_{\theta 2m}^2 + C_{m2m}^2 \ (Vector Identity)$$

$$M_{2m} = C_{2m} / \sqrt{kRT_{2m}} \ (Mach Number)$$

$$P_{02m} = p_{2m}(T_{02m} / T_{2m})^{k/(k-1)} \ (Equivalance)$$

Plus assorted dependent variables:

$$\lambda_{2m} = C_{\theta 2m} / C_{m2m} \,(Swirl\,Parameter)$$
$$\sigma_{2m} = \mu_{2m}(\lambda_{2m} - \tan\beta_{2b}) / \lambda_{2m} \,(SlipFactor)$$
$$\mu_{2m} = C_{\theta 2m} / U_2 \,(WorkInputFactor)$$

2.2. Diffuser equations

The vane less diffuser is frequently employed in process compressors, refrigeration compressors, and turbocharger and micro turbines compressors. For a vane less diffuser, the conservation equations are solved by integration through the vane less diffuser. In a one–dimensional compressor analysis this is done as a succession of states through the vane less diffuser. The full set of equations account for wall friction effects in both the linear momentum equations: [8]

$$C_m \frac{dC_m}{dr} - \frac{C_\theta^2}{r} + C_f \frac{C^2 \cos\beta}{b \sin\phi} + \frac{1}{\rho}\frac{dp}{dr} = 0$$
$$C_m \frac{dC_\theta}{dr} + \frac{C_m C_\theta}{r} + C_f \frac{C^2 \sin\beta}{h \sin\phi} = 0$$
$$\frac{1}{\rho}\frac{dp}{dr} + \frac{1}{C_m}\frac{dC_m}{dr} + \frac{1}{h}\frac{dh}{dr} + \frac{1}{r} = 0$$
$$\rho = p / RT$$
$$T_0 = T + C^2 / 2C_P$$

Where the ϕ is the diffuser inclination. This set of equations can be solved directly by Runge- Kutta integration.
The equation:

$$C_f = k\left(\frac{1.8 \times 10^5}{\mathrm{Re}}\right)^{0.2}$$

Where k is a constant has been used (Japikse 1982) [8]. A value of k=0.01 is frequently used based on a review of different vane less diffusers in industrial machines.

Based on these equations, the 1D design of compressor has been done and results of preliminary design will be used in 3D design and CFD calculation of this compressor will be complete.

3. Three-Dimensional Design and CFD calculations

In radial turbomachines, the first blade passage measurements were published by Eckardt (1976) for a centrifugal compressor. The computation methods which have been devised and published, large in number and diverse in type to combination with Eckardt useful results to have detail information about flow field in centrifugal impeller passages. Many of researchers such Mc. Nally and Sockol (1985), Japikse (1976-1996), were used CFD methods to turbomachines [7]. Moore et al. (1984) used three-dimensional viscous CFD method to examine the flow in a medium pressure ratio impeller (which used largely in turbochargers and micro turbines). The CFD results, although not directly compared with measurements, showed several aspects of loss production in the impeller. Hathaway et al. (1993) and Skoch et al. (1997) are helping to fill some gaps in CFD analysis [5]. Hirsch et al. (1996) calibrated their CFD method using Krain's data (1981) and performed numerical simulations quoted by theoretical notions concerning secondary flow [5]. In the most nearly years, the use of CFD for the computation of turbo machinery flows has significantly increased, and by combination with measurements it provides a completely tool for simulation, design, optimization and most important of all, analysis of the flow field inside the turbomachines [1]. The main purpose of the presented work is to compare 1D and 3D results to complete design process of a centrifugal compressor for a micro turbine.

3.1. Numerical Approaches for Turbo machinery

The solution of flows in moving reference frames can be best handled using "moving" cell zones. The computation domain can consist only of one moving cell zone or a combination of moving and stationary cell zones. The single rotating frame option "RRF" can be used to model flows in turbo machinery where the flow is unsteady in an inertia frame because the rotor/impeller blades sweep the domain periodically. However, in the absence of stators or volute, it is possible to perform calculations in a domain that moves with the rotating part. In this case, the flow is steady relative to the rotating frame, which simplifies the analysis. If stators are present in addition to a rotor, that it is not possible to render the computational problem steady by choosing a calculation domain that rotates with the rotor or impeller. The CFD programs offers different approaches for treating this problem, all of them are using multiple frames of reference: [1]

-the frozen rotor model

-the mixing plane model

-the sliding mesh model

Fig.2 shows a resume from the different numerical approach used in turbo machinery. In this work, a micro turbine compressor which designed one-dimensionally with CCD code, have been simulated three-dimensionally using the commercial CFD code FLUENT 6.

The often used model for turbo machinery, known as "frozen-rotor"-model only yields satisfying results for efficiency and pressure ratio, at and near the best efficiency [2].

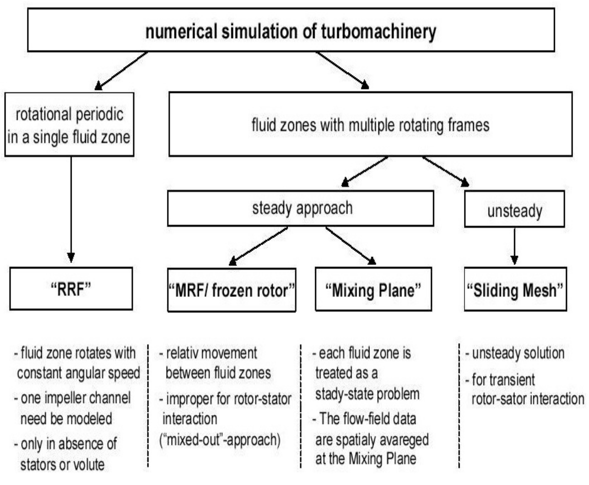

Fig.2: CFD approaches used in turbo machinery [1].

3.2. Grid Generation

One of the most important and time-consuming tasks in the process of a CFD simulation is the generation of the computational grid. First of all is the decision whether the domain shall be discretiz with hexahedral, tetrahedral or hybrid cells. Using tetrahedral cells can save a lot of time because the meshing of the volume works almost automatically. The grids for compressor channel were completely generated with the preprocessor GAMBIT which offers some CAD tools as well as meshing tools. Three-Dimensional (3D) models of the machine described above were generated. In the centrifugal compressor, due to rotational periodicity of the blades, only one channel consisting of pressure and suction side of the main blade had to be generated [4]. The whole impeller is made up of 13 blades. The whole impeller geometry had to be divided into several hexahedral volumes, each of which could be meshed with hexahedral cells (Fig. 3).

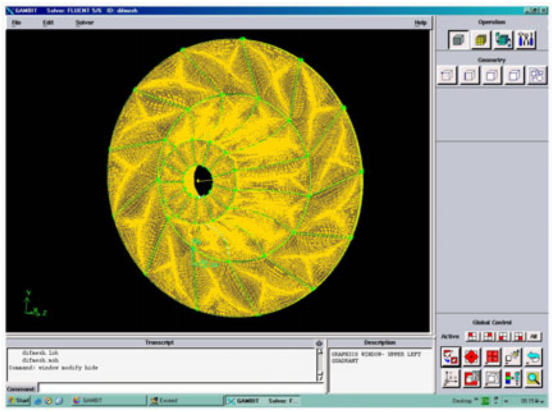

Fig 3: Modeling & Grid generation for Centrifugal Compressor with GAMBIT

3.3. Numerical Implementation

3.3.1. Model parameter

The impeller region was modeled as a reference rotating with respective rotary speed. The most appropriate discretization scheme was the segregated implicit solver. It immediately turns the initial backflow, and solution parameter come to convergence. Turbulence is modeled using the standard $k-\varepsilon$ model without realizable modifications. Also the energy equation has to be solved additionally. The medium is treated as ideal gas with viscous heating. The momentum equation is solved using the QUICK scheme, the pressure correction with the SIMPLE scheme. The under relaxation factor pressure must be very conservative, about 0.3 or 0.2; otherwise the pressure correction might diverse.

3.3.2. Boundary conditions

For the compressible flow, mass flow rate normal to boundary was used as the inlet boundary condition. At the inlet total temperature, turbulent intensity and length scale must be defined too (Tu=1.5% L=10mm) [3]. A velocity at the outlet, the static pressure and backflow properties must be defined. All in inertia reference frame moving walls are part of the rotating reference frame. These walls are treated as moving walls with rotational speed of 0 relative to the adjacent cell zone, which is rotating. Non- rotating walls in the inertia frame of reference, which are part of the rotating reference frame, are treated as moving walls with a rotational speed of absolute 0. The walls are adiabatic.

3.4. Equation for a rotating reference frame

When the equations of motion are solved in a rotating frame of reference, the acceleration of the fluid is augmented by additional terms that appear in the momentum equations. FLUENT allows you to solve rotating frame problems using either the absolute velocity, V, or the relative velocity V_r, as the dependent variable. The tow velocities are related the following equation: [3]

$$V_r = V - \Omega \times r$$

Here, Ω is the angular velocity vector and r is the position vector in the rotating frame.

The left-hand side of the momentum equation appears as follows for an inertia reference frame:

$$\frac{\partial}{\partial t}(\rho V) + \nabla.(\rho V V)$$

For a rotating reference frame, the left-hand side written in terms of absolute velocity becomes:

$$\frac{\partial}{\partial t}(\rho V) + \nabla.(\rho V_r V) + \Omega \times V$$

In terms of relative velocities the left-hand side is given by

$$\frac{\partial}{\partial t}(\rho V_r) + \nabla.(\rho V_r V_r) + 2\Omega \times V_r + \Omega \times \Omega \times r + \rho \frac{\partial \Omega}{\partial t} \times r$$

Where $2\Omega \times V_r + \Omega \times \Omega \times r$ is the Coriolis force. Note that FLUENT neglects the $\rho(\frac{\partial \Omega}{\partial t}) \times r$ term, so it can not accurately model a time-varying angular velocity [3].

Using the relative velocity formulation for flows in rotating domains, the equation for conservation of mass, or continuity equation, can be written as follows for both the absolute and the relative velocity formulations:

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x_i}(\rho V_r) = S_m$$

Energy equation: FLUENT solves the energy equation in the following form:

$$\frac{\partial}{\partial t}(\rho E) + \frac{\partial}{\partial x_i}(u_i(\rho E + p)) = \frac{\partial}{\partial x_i}(k_{eff}\frac{\partial T}{\partial x_i} - \sum_{j} h_{j'} J_{j'} + u_j(\tau_{ij})_{eff}) + S_h$$

Where k_{eff} is the effective conductivity, and $J_{j'}$ is the diffusion flux of species j'. The first three terms on the right-hand side of Energy Equation represent energy transfer due to conduction, species diffusion, and viscous dissipation, respectively S_h includes heat of chemical reaction, and any other volumetric heat sources you have defined. In Energy Equation, $E = h - \frac{p}{\rho} + \frac{u_i^2}{2}$.

Where sensible enthalpy h is defined for ideal gases as $h = \sum_{j'} m_{j'} h_{j'}$ and for incompressible flows as $h = \sum_{j'} m_{j'} h_{j'} + \frac{p}{\rho}$. In this equations $m_{j'}$ is mass fraction of species j' and $h_{j'} = \int_{T_{ref}}^{T} C_{P,j'} dT$ (Where T_{ref} is 298.15 K).

4. Results and Comparison to 1D data

As said above, the main purpose of this work was to design a centrifugal compressor for a micro turbine. Both of the preliminary design and CFD methods were described, and said that the 1D design has been done with CCD code and essential data for 3D design and CFD calculation were obtained from this code. Now with comparison these results, the design process of centrifugal compressor will be complete. Not that by using of CFD results we able to improve performance parameters, and describe the flow field in compressor channel that measurements unable to give this ability. The compressor data in the point of best efficiency (b.e.p) are shown in table 1.

Table 1

Efficiency	η	82%
Rotating Speed	N	7000 (rad/sec)
Mass Flow Rate	$\dot{m}_a$	0.865 (kg/sec)
Pressure Ratio	Pr	4

The flow field inside the computational domain is shown in Figure 5. Contours of static pressure (Fig 4a) and tangential velocity (Fig 4b) are presented in passage of channel. Static pressure contours, for impeller also are shown in Figure 5. It must be emphasized that the flow field inside the impeller shows a good qualitative agreement with the 1D data. In areas of high static pressure, the radial velocity is rather small; the effect has an influence on the flow field inside the impeller. High radial velocities are located in areas of low static pressure. These results are in different mass flow rates through each single impeller channel.

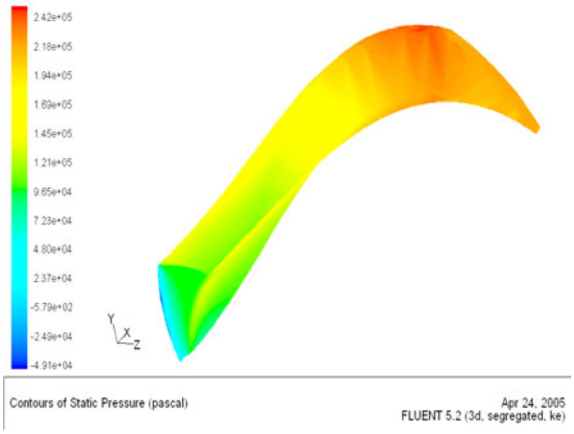

Fig. 4a: Static Pressure Contour

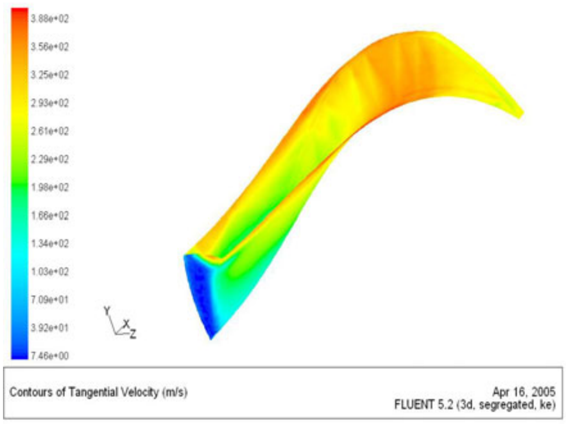

Fig 4b: Tangential Velocity Contour

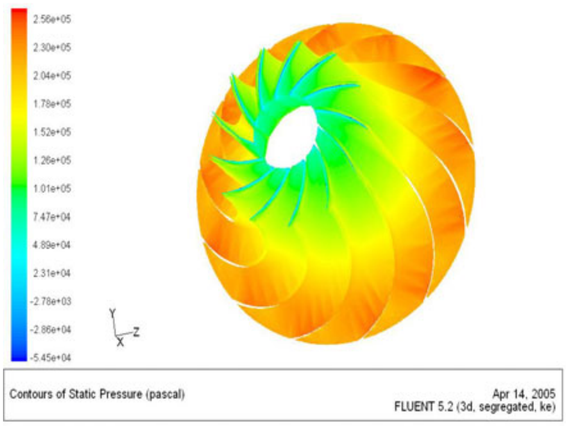

Fig. 5: Static Pressure Contours for Impeller

The comparison of 1D (base on experimental data) and numerical analysis of the centrifugal compressor is shown in table 2. As seen, static pressure and velocity magnitude for impeller, in the b.e.p. show good agreement with 1D data, and maximum difference between CFD calculations and 1D design is about 15%.

Table 2

	1D		CFD (average)	
Impeller	Inlet	Outlet	Inlet	Outlet
Static Pressure (kpa)	84.72	222.07	81.62	222.0
Radial Velocity (m/sec)	0	173	10.19	184.85
Tangential Velocity (m/sec)	28	361.29	44.53	242.83
Relative Velocity (m/sec)	274.89	216.45	281.84	242.66

A good match between 1D and CFD calculations, offer a little change in design process. It means that design of compressor was successfully done, and designers can obtain these data, and use them to improve performance of compressor or for a new design. By using CFD results, also it is possible to describe impeller passage flow phenomena.

5. Conclusions

The centrifugal compressor for a micro turbine has been designed completely. One-Dimensional Design was complete with CCD code, which written and developed by authors, then by using FLUENT code, CFD analyses base on 1D data have been done. The common CFD-package offers different approaches to simulate systems with both stationary and rotating components. With the general single or multi-frames, using local reference frames which are stationary or rotating as appropriate, can steady-state analyses in turbo machinery be performed. One of these different ways to model the interaction of rotating and stationary parts is the frozen-rotor method, which is useful when the circumferential flow variation of each blade passage experiences is large during a full revolution. With this option, computations are again performed in a steady-state mode, based on the assumption of quasi-steady flow around the rotating component at every rotating angle. The detailed CFD model of compressor and it comparison with the experimental data (obtained from CCD code) are explained and shown in this work. A good match between CFD and 1D data was seen for a centrifugal impeller of approximately 3.5:1 pressure ratio. Overall, the CFD gave a good prediction of the performance and resolved enough of the local flow details. By using this information of flow phenomena, it is possible to improve compressor performance, or make a new design of centrifugal compressor for different machines.

REFERENCES

1. Tamma A, Gurge M., Stoffel B. “Experimental and 3-D Numerical Analysis of the Flow Field in Turbomachines Part One” Darmstadt University of Technology, 1999.
2. Tamma A, Gurge M., Stoffel B. “Experimental and 3-D Numerical Analysis of the Flow Field in Turbomachines Part Tow” Darmstadt University of Technology, 1999.
3. Fluent5 User’s Guide, FLUENT Inc. 1998.
4. GAMBIT Modeling Guide, FLUENT Inc. 1998.
5. Larosiliere L.M, Skoch G.J, “Aerodynamic Synthesis of a Centrifugal Impeller Using CFD and Measurements”, NASA Technical Memorandum 107515, AIAA-97-2878, 1997.
6. Pitkanen H, Esa H, Sallinen P, Larjola J, “Time-Accurate CFD Analysis of a Centrifugal Compressor”, Department of Energy Technology, Lappeenranta University of Technology, Finland, 1999.
7. Whitfield A, Baines N.C, “Design of Radial Turbomachines”, First Edition, Longman, New York, 1990.
8. Japikse D, “Centrifugal Compressor Design and Performance”, Concepts ETI. 1996.
9. Ibaraki S, Matsuo T, Kumo H, Sumida K, Suita T, “Aerodynamics of a Transonic Centrifugal Compressor Impeller”, Nagasaki R&D Center, Transactions of the ASME, Vol. 125, APRIL 2003.

10. Pitkanen H, Esa H, Sallinen P, Larjola J, Backman J,“Centrifugal Compressor Design and Testing in the Finnish High Speed Technology”, Department of Energy Technology, Lappeenranta University of Technology, Finland, 2000.
11. Niazi S, Stein A, Sankar L.N, “Development and Application of a CFD Solver to the Simulation of Centrifugal Compressors”, School of Aerospace Engineering, Georgia Institude of Technology, Atlanta, GA, 1998.
12. Energy Nexus Group, Arlington Virginia, “Technology Characterization: Microturbines”, Prepared for Environmental Protection Agency Climate Protection Partnership Division, Washington, DC, March 2002.

BIOGRAPHY OF PRESENTER

Name: Reza Aghaei Tog

Education: Msc. Degree in Aerospace Engineering from Tehran Polytechnic

Affiliation(s) and Function(s): Amir Kabir University of Technology, Niroo Research Institute of IRAN (NRI), Aerospace Industries Organization of IRAN

Experience: Design of Turbomachines, CFD Analysis, Rotor Dynamic

Address: No.33, Taleghani Allay, Lorestan st., 10th Farvardin st., Piroozi av., Tehran, IRAN

P.C.: 1764913915

Tel: 0098213069492

E-mail: REZA_TOG@YAHOO.COM