One-Dimensional Numerical Analysis of Cross-Sectional Area Variation Effects on Flow through the Gas Turbine Diffuser

Nima Zamani Meymian [1], Reza Khodadadi [1], Hiva Rashidzadeh[1], Jafar Hooshmand[1]

[1] TURBOTEC Co., Sharif Energy Research Institute, Tehran, Iran

ABSTRACT

In the paper, one-dimensional compressible flow of gas inside gas turbine's diffuser has been simulated. The modeling has been performed to the aim of obtaining boundary conditions of outlet gas from diffuser and inlet gas to the combustion chamber. Depending on working flow regimes of fluid including subsonic, transonic, and supersonic flows, changes of diffuser cross-section have different effects on gas flow characteristics. For these effects to be correctly imposed, Mach number of the gas flow in each time-step affected by changes of cross-section would be determined, depending on the local Mach number in the same time-step. Obtaining distribution of Mach number along diffuser length, changes in other main characteristics of flow such as pressure, temperature, speed, and density for all of the points along diffuser length would be obtained. In order to verify the validity of the numerical algorithm used, the gas flow would be solved in a divergent nozzle, and compared to other numerical methods. At the end, using gas turbine diffuser's geometrical information, compressible gas flow inside it would be studied using the actual boundary conditions for a 25 MW gas turbine.

Keywords: *Gas Turbine, Diffuser, One-Dimensional Simulation, Cross Section Change*

Introduction

In designing gas turbines, compressor's compressed air outlet has to have the required condition to be combined with fuel so that stable combustion would be achieved. From among most important of these conditions is appropriate air speed so that high air velocity would not be resulted in blowing out the flame. To obtain the appropriate air conditions, in the path of inlet gas to the gas turbine combustion chamber, a divergent diffuser is used [1-3]. In Figure (1), one 25MW gas turbine is shown.

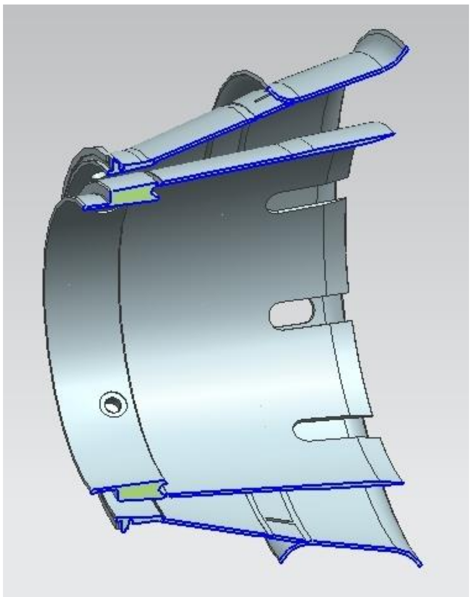

Figure 1- 3D view of the diffuser

Inlet gas of the diffuser come from the compressor [4]. Outlet gas passes through the two annular cylinders shown in Figure (1). Knowing gas conditions in the diffuser outlet requires a precise simulation of the diffuser. The effects of gas turbines diffuser has been extensively studied via experimental and numerical methods [5-7]. In the paper, the effects of diffuser's cross-section change on characteristics of gas flow have been studied in one-dimensional form. It should be noted that, due to 3D geometry of cone-shaped burner in diffuser's path, passage of fuel entrance to burner which is placed perpendicular to diffuser, as well as holders, a comprehensive analyses of obstruction effects of these factors and their effects on pressure drop inside diffuser have been neglected, however, the accuracy of developed model would be increased if these effects are added to the model.

Numerical Solution Algorithm

Diffuser's modeling and effects of cross-section changes are imposed in one-dimensional form, using quasi-one-dimensional flow equations. Quasi-one-dimensional equations are valid upon this assumption that the rate of cross-section change in the chamber would not be so large in the longitudinal direction [8]. In those areas with relatively considerable changes in cross-section size, rate of cross-section change for each cell can be reduced, through minimizing computational network.

In achieving a mathematical model of effects imposed by cross-section variations, it has been supposed that in all points except for those areas of probable of shock occurrence, fluid flow is isentropic. This assumption in terms of those flows occurring in a diffuser is fully acceptable in which effects of friction and heat transfer from the wall is limited to a narrow strip adjacent to solid boundaries. So, these effects could be ignored or their effects could be considered through usage made of engineering and empirical coefficients [8].

In this study and to find the numerical algorithm of the effect of cross-section change on gas flow, Mach number would be used as a primary parameter. To facilitate calculations, an algebraic relationship would be used to find the change of Mach number affected by the change of cross-section, along with the chamber. Considering ideal gas relationships during a specified procedure, values of changes of other flow characteristics such as temperature, pressure, speed, and density along the channel would be obtained, dependent on new Mach number and cross-section.

To obtain governing equations on fluid flow inside a diffuser, a control volume like the one presented in Figure (2) can be taken into consideration. Using conservation laws, ideal gas law, and definition of Mach number in control volume input-output, changes in Mach number in control volume can be obtained based on changes of dependent variables to Mach number [8]. Relationships are presented for an ideal gas in steady-state.

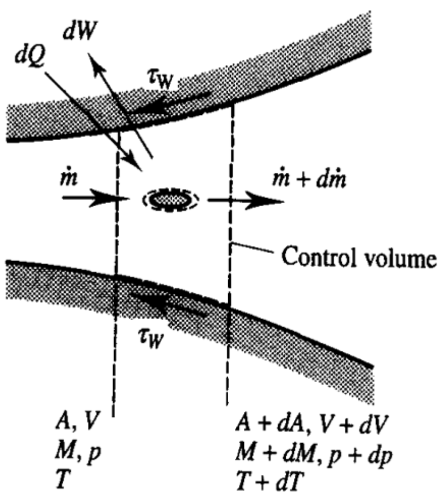

Figure 2- A view of control volume in general state [1]

$$M^2\left(1+\frac{dM^2}{M^2}\right)=\frac{V^2}{\gamma RT}\left(1+2\frac{dV}{V}-\frac{dT}{T}\right) \tag{1}$$

Simplifying above equation and considering a degree of accuracy, equations of survival laws could be placed in equation (1). Finally, the following relationship would be obtained as for changes in Mach number.

Ignoring small coefficients resulted from multiplication of two differential variables and considering a degree of equation accuracy (1) would be simplified as relationship (2).

$$\frac{dM^2}{M^2}=\frac{2dV}{V}-\frac{dT}{T} \tag{2}$$

Through algebraic calculations for three conservation of mass and energy, as well as the principle of conservation of momentum for control volume in Figure (2), changes of all variables in control volume can be obtained with related details provided in reference (1). Through calculations and simplification, 8 differential equations would be obtained for 8 variables. At this stage, the main focus is on obtaining $\frac{dM^2}{M^2}$ as a main computational parameter. The related equation would be stated as:

$$\frac{dM^2}{dx}=\left\{\frac{M^2\left[1+(\gamma-1)M^2/2\right]}{\left(1-M^2\right)}\right\}G \tag{3}$$

Where,

$$G=-\frac{2}{A}\frac{dA}{dx}+\frac{\left(\gamma M^2+1\right)}{T_0}\frac{dT_0}{dx}+\frac{2\left(1+\gamma M^2\right)}{\dot{m}}\frac{d\dot{m}}{dx}+\gamma M^2\left(f\frac{p}{A}+\frac{2}{\gamma pM^2A}\frac{dF_D}{dx}-\frac{2r_i}{\dot{m}}\frac{d\dot{m}}{dx}\right)$$

In the above equation, sentences are respectively related to cross-section change, heat transfer from wall, mass flow rate of cooling air, wall friction, diffuser’s pressure drop, and air inlet angle, compared to control volume. Having Mach number’s profile of changes in solution range, other flow characteristics would be determined according to the following relationships:

$$\frac{T_2}{T_1}=\left[\frac{1+(\gamma-1)M_1^2/2}{1+(\gamma-1)M_2^2/2}\right]\frac{T_{02}}{T_{01}} \tag{4}$$

$$\frac{p_2}{p_1}=\frac{\dot{m}_2}{\dot{m}_1}\frac{M_1}{M_2}\frac{A_1}{A_2}\sqrt{\frac{T_2}{T_1}} \tag{5}$$

$$\frac{V_2}{V_1}=\frac{M_2}{M_1}\sqrt{\frac{T_2}{T_1}} \tag{6}$$

$$\frac{\rho_2}{\rho_1}=\frac{p_2}{p_1}\frac{T_1}{T_2} \tag{7}$$

Considering $\dot{m}$ being zero, heat transfer and friction coefficients of the wall could be calculated from above-simplified relationships. Solution process concerns solving compressible flow for a diffuser with no consideration of any of the stimulating factors of flowing energy. Then, using the solution algorithm with its relationship presented above, the effect of changes in cross-section, the mass flow rate of cooling flow, and heat transfer would be implemented in code. It should be noted that relationships extracted can only be used for steady-state flow when the transonic point does not exist along the channel.

Solution begins with entering boundary conditions for the first cell. Considering the solving process as spatial marching, the information in each cell would be only transferred from upstream. As a result, in the last cell, there would be no need for the outlet boundary conditions, and required data enter the cell from the cell next to the last cell.

Validation

To validate the numerical algorithm presented, the gas flow would be modeled in a supersonic divergent nozzle; then, it would be compared with two reliable numerical methods. Cross-section of divergent nozzle changes according to a relationship (8).

$$S(x)=1.398+0.347\tanh(0.8x-4) \tag{8}$$

Air with a specific heat of 1.4 enters a 10-foot long nozzle, in supersonic form. Solving the problem aims at obtaining changes in flow characteristics such as pressure, density, and speed along the channel length. The boundary condition at nozzle entrance section would be defined as followings:

$$\begin{cases} M = & 1.5 \\ P = 2000 & lb_f \ / \ ft^2 \\ T = & 520R \end{cases}$$

Boundary conditions used in the developed code are total pressure, density, and energy. According to the solving method of characteristics in governing equations of fluid's movement, three analytical boundary conditions are required in the inlet in the supersonic flow [9]. In solving the problem and in addition to the three above conditions, input speed also would be obtained from total pressure and energy values. Total energy value in problem input is equal to 3636557. Due to the supersonic regime in the nozzle outlet also, no analytical boundary condition has to be entered [10]. Explaining this, all flow components are defined in numerical form and in the form of zero gradients at outlet boundary of the nozzle. Also, input values of pressure, density, and speed would be used as initial conditions to solve the problem [11]. Problem-solving results with a present code for 200 cells have been compared to those of other numerical methods and analytical method of solving.

Figures 3, 4, and 5 respectively show comparisons made on changes of pressure, speed and density along with the nozzle obtained from present code, and the results from other numerical methods presented in reference [9].

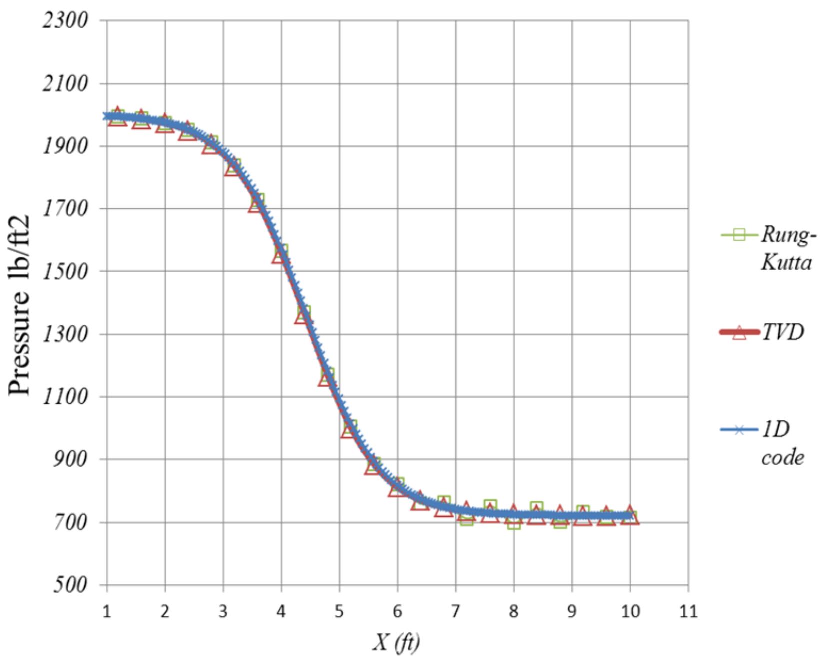

Figure 3- Pressure change inside nozzle (sample problem)

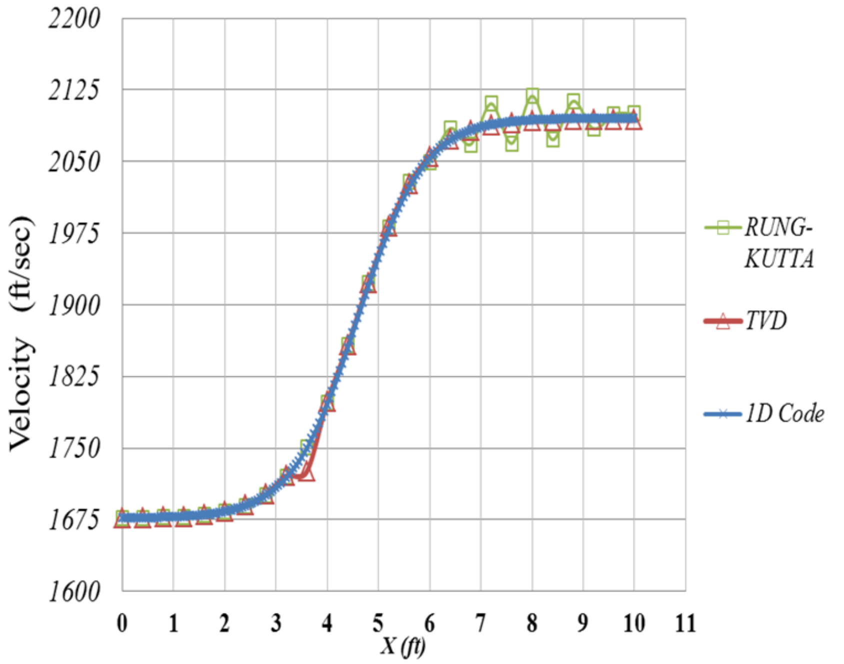

Figure 4- Speed change inside nozzle (sample problem)

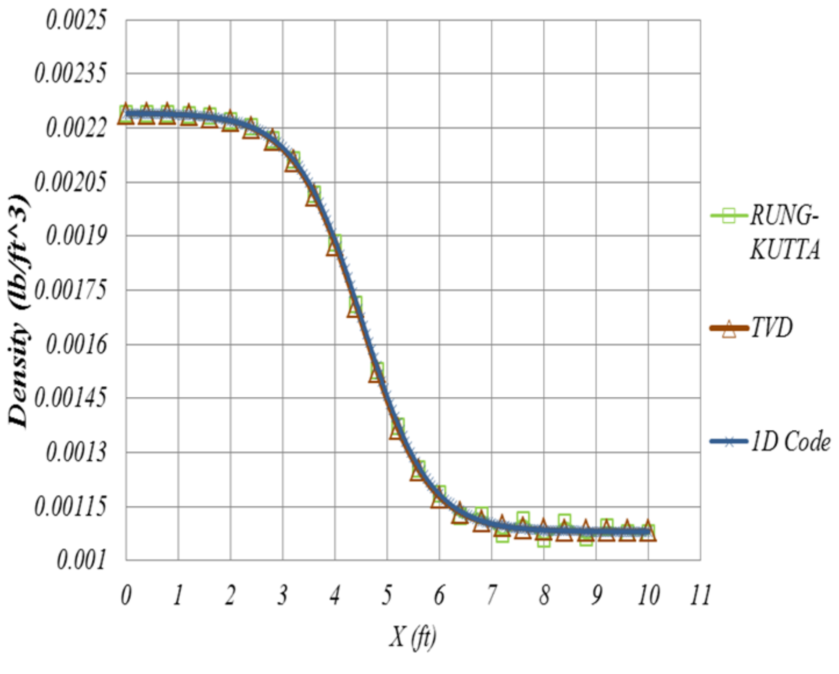

Figure 5- Density change inside nozzle (sample problem)

As observed, the numerical algorithm presented that has been added to the code for calculations related to change of cross-section predicts with high accuracy profile of changes in flow variables. Also, results show that Rung-Kutta algorithm in end sections of the nozzle represents some fluctuations in the distribution of pressure, speed, and density. So, this is not an appropriate method of solving the problem.

Results

As far as the problem is solved through spatial marching, only boundary conditions determining flow in diffuser's entrance section is required. These conditions include total pressure (P=939833.3pa); total temperature (T=600K); and, mass flow rate (70kg/s) [12]. These values have been obtained from experimental measurements of gas turbines in gas compressor stations. Using thermodynamic relationships and expressed boundary conditions, Mach number can be obtained through following non-linear relationship, as well as try and error [3]:

$$\dot{m} = \frac{P_T}{RT_T}\left(1+\frac{\gamma-1}{2}M^2\right)^{(\gamma-1)} \times M\sqrt{\gamma RT_T\left(1+\frac{\gamma-1}{2}M^2\right)^{-1}} \times A \tag{9}$$

Inlet Mach number is 0.42. Accordingly, when the numerical algorithm for boundary conditions and geometry of gas turbine are solved, results from the profile of changes in characteristics would be achieved as follows.

Figure 6 shows the change of static pressure in a diffuser. Inlet pressure (9.4bar) increases to almost 11bar as outlet pressure. The increase results from the transformation of gas kinetic energy to potential energy. In Figure (7), it is observed that the decrease is made in inlet Mach number (0.42) to almost 0.07. This amount of reduction in Mach number and decrease made in gas speed, as a result, are necessary for the flame not to be pulled out. Changes in density, speed, and temperature also are shown, in continuation. Changing the process of density like pressure change takes place in a diffuser. The reason is the linear relation of flow main variables for an ideal gas.

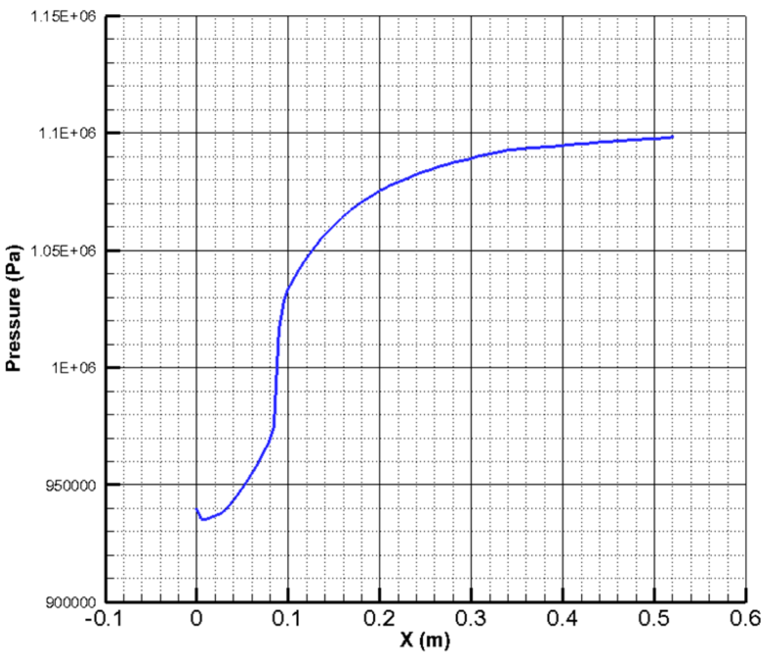

Figure 6- Pressure change inside diffuser

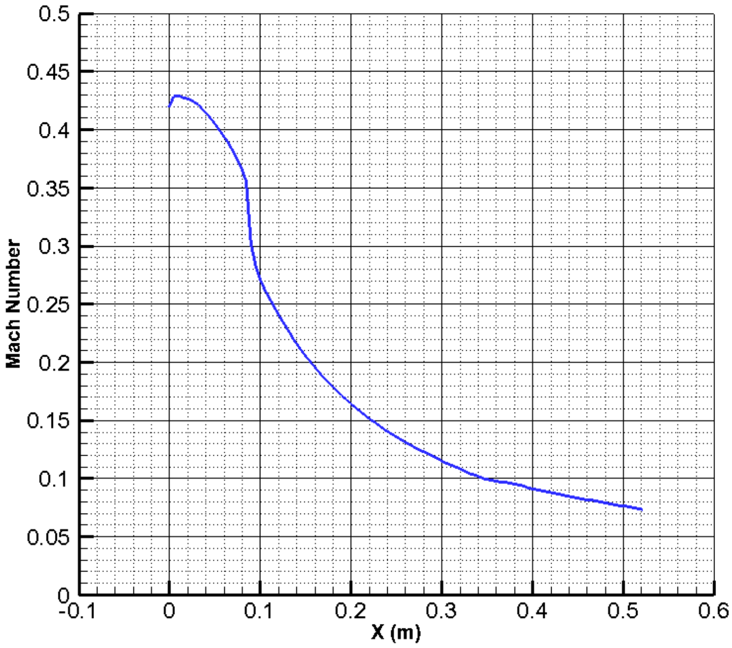

Figure 7- Change of Mach number inside the diffuser

About 12% increase is observed in gas density along with diffuser (Figure 8). Speed value shows a severe decrease, reaching to lower than 40m/s in diffuser outlet. This is a desirable speed for air to be entered into and mixed with fuel (Figure 9). The gas temperature would be subjected to 20K increase, reaching to 620K before entering the burner (Figure 10). Speed counter in a diffuser is shown in figure 11. As it is observed, even small sudden changes in cross-section of diffuser have a high effect on characteristics in high-speed flows. For example, change of cross-section in the outer liner where X=0.09m would not be much, compared to increase made in cross-section of the diffuser as a whole; however, speed would be considerably reduced.

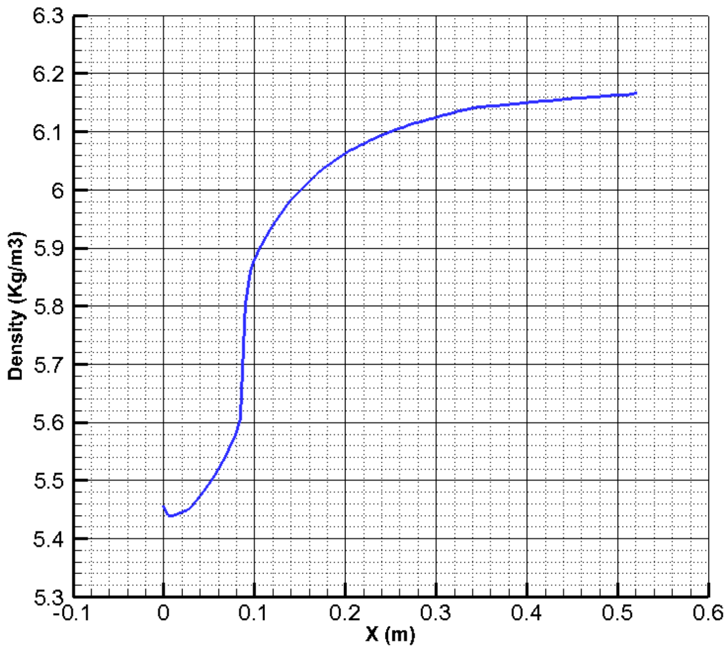

Figure 8- Density change inside diffuser

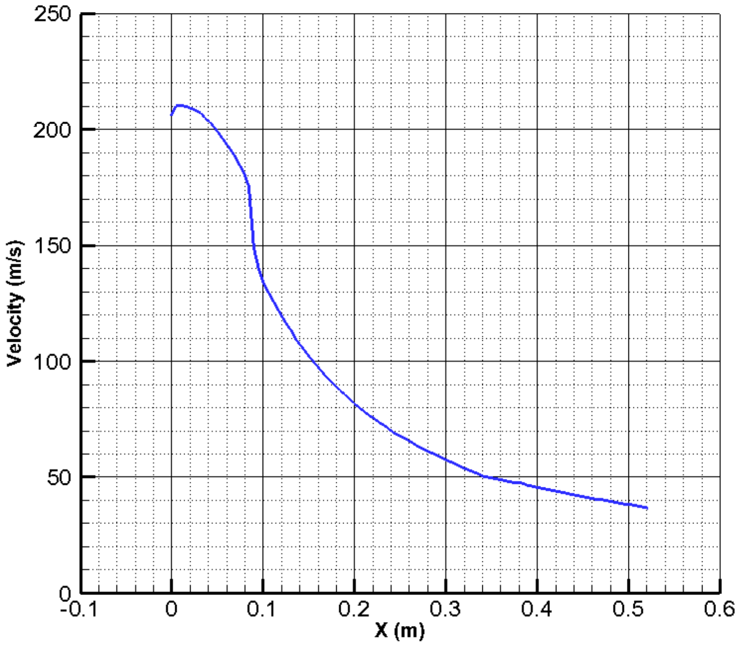

Figure 9- Speed change inside diffuser

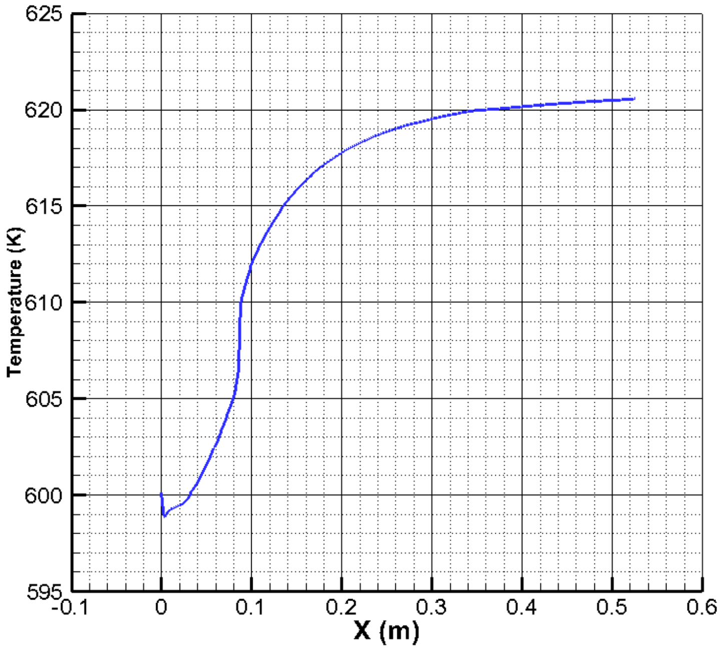

Figure 10- Change of temperature inside the diffuser

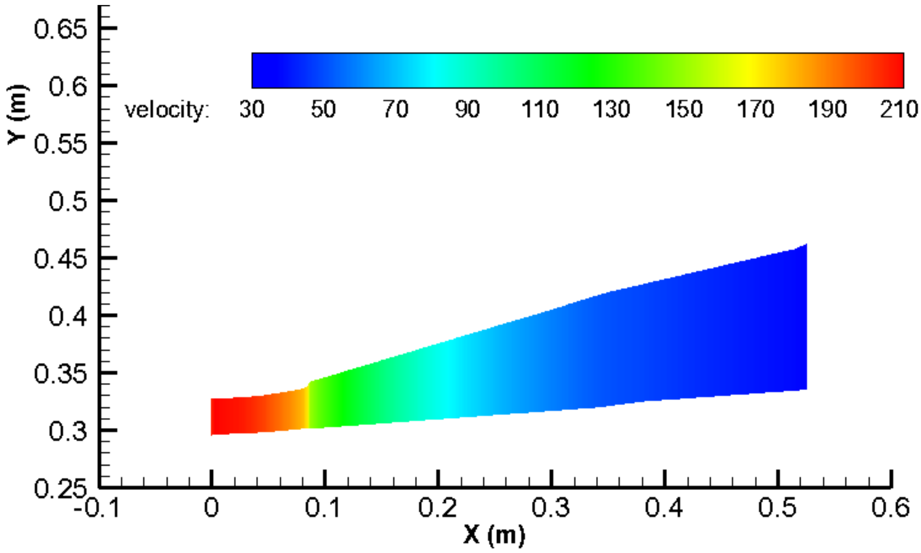

Figure 11- Speed counter inside the diffuser

Conclusion

A one-dimensional numerical algorithm has been used for solving compressible flow in a gas turbine diffuser. The precision of the algorithm in steady-state is very high and it is capable of predicting outlet flow from the diffuser in high accuracies. To do so, one-dimensional continuity equations of steady-state has been solved. To achieve the profile of changes in main fluid characteristics, the Mach number parameter has been used as the reference variable. Using the extracted equations, distribution of Mach number along the diffuser was obtained; then, other parameters were determined with the help of Mach number. Using the numerical algorithm presented in this paper, the problem of compressible flow in nozzle has been solved; and, it became clear that in the supersonic regime of flow, the algorithm is highly accurate. Finally, through usage made of outlet boundary conditions of the compressor in a gas turbine under maximum power, gas turbine diffuser has been studied. Increase in pressure, density, temperature and decrease in Mach number and gas velocity were amongst the results. According to the results, the slope of changes in characteristics in the first half of the length of the diffuser is higher, despite the uniform increase in cross section. Also, even small sudden changes in cross section area have a high influence on the outlet flow conditions. Not considering obstruction effect of such surfaces as burner and fuel inlet passage, the results have been obtained. Using a reliable scientific method for consideration of these factors is necessary so that modeling accuracy would be increased. The report results are being used as the inlet boundary conditions of the combustion chamber.

References

[1] A. Hirschmann, S. Volkmer, M. Schatz, C. Finzel, M. Casey and M. Montgomery, "The Influence of the Total Pressure Profile on the Performance of Axial Gas Turbine Diffusers," ASME Turbo Expo 2010: Power for Land, Sea, and Air, Paper No. GT2010-22481, pp. 1197-1207; 11 pages, doi:10.1115/GT2010-22481.

[2] R. C. Adkins, A. C. Benim, P. C. Escher, J. Hellat and W. M. Koenig, "A Combustor Diffuser of Annular Configuration Suitable for Industrial Gas Turbines," ASME 1992 International Gas Turbine and Aeroengine Congress and Exposition, Paper No. 92-GT-041, pp. V004T11A004; 7 pages, doi:10.1115/92-GT-041.

[3] Ajay K. Agrawal, Jayanta S. Kapat and Tah-teh Yang, "Flow Interactions in the Combustor-Diffuser System of Industrial Gas Turbines," ASME 1996 International Gas Turbine and Aeroengine Congress and Exhibition, Paper No. 96-GT-454, pp. V003T06A045; 8 pages, doi:10.1115/96-GT-454.

[4] Navai, A., & Zamani Meymian, N. (2019, July 17). A Novel Method for Developing Compressor's Characteristic Curves Due to the Guide Vane Stagger Angle Variation. https://doi.org/10.31224/osf.io/d2ype.

[5] A. Duncan Walker, Paul A. Denman and James J. McGuirk, "Experimental and Computational Study of Hybrid Diffusers for Gas Turbine Combustors," ASME Turbo Expo 2003, collocated with the 2003 International Joint Power Generation Conference, Paper No. GT2003-38406, pp. 311-320; 10 pages, doi:10.1115/GT2003-38406.

[6] K. C. Karki, V. L. Oechsle and H. C. Mongia, "A Computational Procedure for Diffuser-Combustor Flow Interaction Analysis," ASME 1990 International Gas Turbine and Aeroengine Congress and Exposition, Paper No. 90-GT-035, pp. V003T06A013; 9 pages, doi:10.1115/90-GT-035.

[7] Farrokh Zarifi-Rad, Hamid Vajihollahi and James O'Brien, "Validating Numerical CFD Simulations With Experimental Data for Turbulence Phenomena in Axial Flow Gas Turbine Diffusers," ASME Turbo Expo 2009: Power for Land, Sea, and Air, Paper No. GT2009-59086, pp. 807-819; 13 pages, doi:10.1115/GT2009-59086.

[8] Patrick H. Oosthuizen & William E. Carscallen, "Compressible Fluid Flow," Book, USA: McGraw Hill, 1977.

[9] Klaus A. Hoffman & Steve T. Chiang, "Computational Fluid Dynamics," Book, Kansas: Engineering Education System, 2000.

[10] John D. Anderson, JR, "Computational Fluid Dynamics," Book, USA: McGraw Hill, 1995.

[11] John D. Anderson, JR, "Modern Compressible Flow with Historical Perspective," Book, New York, USA, McGraw Hill, 2003

[12] Glenn Douglas Garrard, “The Aerodynamic Turbine Engine Code for the Analysis of Transient and Dynamic Gas Turbine Engine System Operations,” Knoxville, 1995.