Iranian Journal of Science & Technology, Transaction B, Engineering, Vol. 29, No. B2
Printed in The Islamic Republic of Iran, 2005

PERFORMANCE PREDICTION OF TWIN-ENTRY TURBOCHARGER TURBINES*

S. GHASSEMI**, E. SHIRANI[1] AND A. HAJILOUY-BENISI[2]

[1]Dept. of Mechanical Engineering, Isfahan University of Technology, Isfahan, I. R. of Iran
[2] Dept. of Mechanical Engineering, Sharif University of Technology, Tehran, I. R. of Iran
Email: sghassemi@gmail.com

Abstract– In this paper, the performance of the twin-entry radial flow turbine under steady state and partial admission conditions is modeled. The method, which is developed here, is based on one-dimensional performance prediction. In one-dimensional modeling, the flow properties are assumed constant on a plane normal to the flow direction. This assumption is in contrast with the flow at the rotor entry of a twin-entry turbine under partial admission condition. In this study the one-dimensional performance prediction method for a single-entry turbine is modified to analyze the twin-entry turbine. In particular, the loss coefficients due to friction, clearance and blade loading, which are already developed for single-entry turbines, are modified. Also, additional losses in the rotor are considered because of twin-entry rotor inlet conditions and the rotor-mixing losses. Indeed, in a single-entry turbine with symmetric volute, the flow tends to move toward the shroud. A correlation for the radial velocity profile at the rotor entry for this case is obtained and is considered to be optimum. Then the rotor mixing loss is estimated. Finally a model based on the above mentioned matters is developed. The results obtained from the model are compared with the experimental results and good agreements are obtained. In this paper, special behaviors of the flow in the twin-entry turbine are also investigated and some physical interpretations are presented.

Keywords – Turbocharger, radial flow, turbine, twin-entry, one-dimensional modeling, performance prediction

1. INTRODUCTION

Turbochargers are one of the most important components of the diesel engines. They increase the power output to the engine weight ratio and the efficiency. Using twin-entry radial flow turbines in turbocharging systems of small diesels gives the possibility of using the energy of pulsating exhaust gases, and is a very common practice. Despite its importance, there are not many research activities in the area of twin-entry turbocharger turbines. Almost all of the research in this area are experimental [1-4]. The researchers commonly stated that the shroud and hub side entries have different effects on the turbine performance. The best efficiency is obtained when the mass flow rate in the shroud side entry is more than the hub side entry [2-4]. Different values of the optimum mass flow ratio are given by different authors. For example, Capobianco and Gambarotta [3] gave 0.014 and Baines and Yeo [4] gave 0.6. The large difference between these values may be due to the different shape of the turbine casing they used. Capobianco and Gmbarotta [3] used a turbine with inclined volute and unequal entry areas, but Baines and Yeo [4] used a turbine with a symmetric volute. It is worth mentioning that the minimum efficiency is obtained when one of the entries, in particular the shroud side, is fully closed [2-4]. The efficiency of the twin-entry turbine at full admission condition is about 7 percent less than the single entry turbine with same rotor [3].

The solution methods for the flow in the radial flow turbines can be classified as numerical and modeling. The two or three-dimensional solutions are obtained numerically. In this procedure, the researcher must be an expert in numerical methods and do not need to model the flow characteristics. But in one dimensional flow modeling, the modelers must have a good insight of the flow. In this procedure, different losses are identified and each one is modeled separately. The one-dimensional approach is used for the preliminary design procedure and is simpler and easier to use.

*Received by the editors November 13, 2002 and in final revised form November 20, 2004
**Corresponding author

The only numerical flow simulation obtained for this flow, as far as the authors know, is the work of Lymberopoulos *et al*. [5]. They simulated the flow in the volute of a twin-entry turbine by solving the quasi three-dimensional Euler equations.

The one-dimensional modeling of the single-entry radial turbines has been studied by many authors [6-12]. However, modeling of the twin-entry turbines has not been the center of the researchers' attention. The majority of the 1-D analysis is for single-entry turbines. Despite the fact that twin-entry turbines are used extensively in industries, to the extent of the authors' knowledge, there has not been any paper in the literature which deals with the 1-D analysis of two-entry turbines. Also, there are very few discussions about the flow behavior and loss effects in these turbines. Baines and Yeo [4] explained the efficiency characteristics of the turbine based on the flow patterns in the rotor. They stated that under a partial admission condition, when the relative flow angle is not at its optimum value, an unsuitable inflow angle at the rotor hub side results in higher losses. Although this analysis is interesting, it is not sufficient. For example, this analysis cannot explain why the best efficiency is obtained when the mass flow rate of the shroud side entry is more than that of the hub side entry. Another problem is that the available one-dimensional flow equations are obtained only for single-entry turbines and they cannot be used for twin-entry turbines directly.

The aims of this paper are 1) to give explanations for some of the behaviors of the turbine which have not been discussed in the literature. These explanations help the extension of 1-D modeling to twin-entry turbines by using more information about the flow and the losses inside the turbine, 2) to develop 1-D modeling for a twin-entry turbine using similar models for a single-entry turbine. Since the coefficients in the model are generally obtained from the experimental data, to accurately calculate the coefficients of the model, it can be improved by using more experimental data, as they will be available in the future.

2. FLOW PATTERN IN THE TURBINE

a) Flow in the casing

Figure 1 shows the flow velocity, the flow angle and the incidence angle at the rotor entry (volute exit), for different partial admission conditions, based on the measurements obtained by Baines and Yeo [4]. The total mass flow rate is held constant at all cases. According to these experimental results, except when one of the entries is fully closed, as the mass flow rate increases at one entry, the flow velocity also increases at the same passage, and the flow angle does not change much at this section. Based on Whitfield and Baines [11], simple considerations of angular momentum, the angle of flow leaving a vaneless volute is a function of geometry only. But this situation does not hold for a twin-entry turbine when one of its entries is completely closed. In this case, the flow velocity at the open side is at its maximum value, as expected. The velocity at the other side is not close to zero, but is about the full admission case. At the same time, the flow angle at the closed side increases considerably. Baines and Yeo [4] stated that this velocity increase is due to a sudden expansion. But this conclusion seems to be invalid. Figure 2 shows the radial velocity components at the rotor entry for three different cases. In this figure, the area under the curves indicates the volumetric flow rate. It is clear from the figure that the volumetric flow rate (the area under the curve) when one of the entries is fully closed is not increased, but decreased. So, the statement made in [4] for this phenomenon cannot be justified and the flow is compressed in this case.

This behavior can be explained by the following statements. When the mass flow rates at the hub and shroud side entries are different, the flow streams in the interspace affect each other and the flow with higher velocity tends to move the flow at the other side along itself. As the velocity difference between the two entries is increased, the above-mentioned behavior becomes more pronounced due to the friction between two streams and the low momentum of the flow with smaller velocity. At the same time, because of the mass flow rate limitation, the flow at the side with the lower mass flow rate cannot be moved faster along the radial direction. But at the tangential direction, this limitation does not exist. Thus the tangential component of the velocity is increased in this area. As a result, the absolute flow angle and the velocity are increased in the closed side. This explanation is used for modeling in this work.

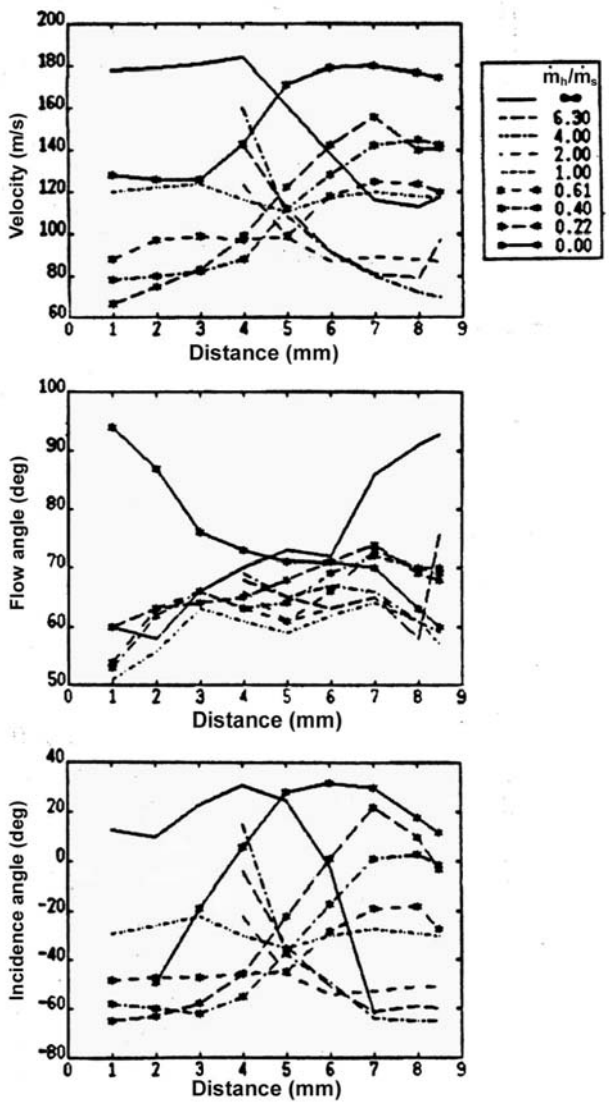

Fig. 1. Flow characteristics at the volute exit from hub side to shroud side [4]

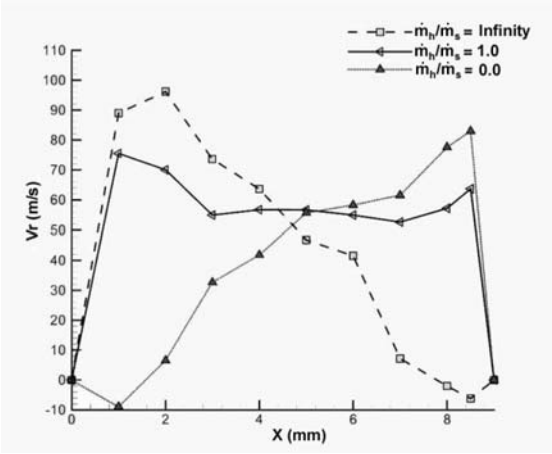

Fig. 2. Radial velocity component at the volute exit from hub side to shroud side

b) Flow in the rotor

The flow pattern in the rotor affects the characteristics of twin-entry radial turbines. Figure 3 shows the radial component of the flow obtained by laser anemometry at the entrance to the rotor of a single-entry turbine [13]. Figure 4 presents the relative Mach contours in the rotor of a single-entry turbine [14]. As can be seen from these figures, the flow at the rotor entry tends to pass through the region near the shroud. As the flow always passes through the lines that have optimum behavior and minimum losses, in this paper, we have chosen the optimum radial velocity distribution at the entrance to the rotor to be the same as the radial velocity distribution at the rotor entrance of a single-entry turbine with a symmetric volute. So, if the mass flow ratio in the twin-entry turbine entries is such that the radial velocity at the rotor entry is the same as the optimum case, there will not be extra losses in the twin-entry turbine rotor. Otherwise, it is expected to have additional losses in the rotor. Indeed, according to the experimental results [4], the flow pattern at the rotor exit is a weak function of the flow distribution at the rotor entry. In the other hand the flow angle and absolute velocity at the rotor exit under a different partial admission condition is similar to the full admission

case. So when the flow pattern at the rotor entry does not match with the optimum case, the flow during a mixing process along the rotor becomes similar to the optimum condition at the rotor exit. This process produces additional loss in the rotor, which we call *rotor mixing* loss. Another case which is worth mentioning, is the effect of the tangential velocity component on the losses. When the tangential velocity at the rotor entry is different from its optimum value, the incidence losses occur.

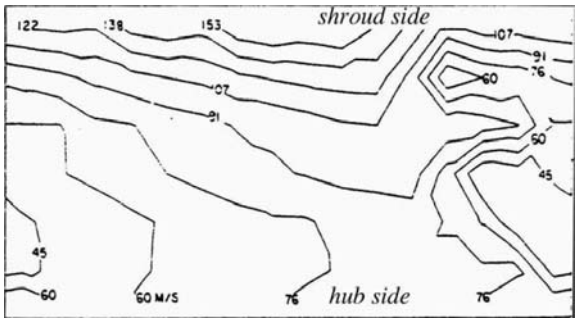

Fig. 3. Radial velocity contours at the entrance to the rotor of a single entry turbine [13]

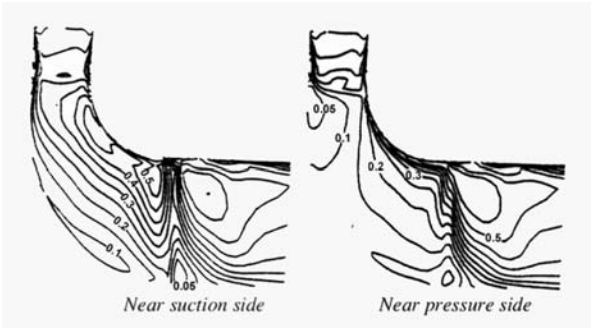

Fig. 4. Relative Mach contours in the rotor of a single-entry turbine [14]

In the twin-entry turbines, according to above discussions, it is expected that the higher efficiency can be obtained when the mass flow rate at the shroud side entry is more than the other side and the minimum efficiency is obtained when the radial velocity distribution at the rotor entry has the maximum deficiency with respect to the optimum case. That occurs when the shroud side entry is fully closed.

3. MODELING

a) One-dimensional equations

In one-dimensional modeling, the flow passage is divided into several regions, as shown in Fig. 5, and the flow is modeled in each part separately. The following equations are used for the flow modeling at the single-entry turbine rotor [11].

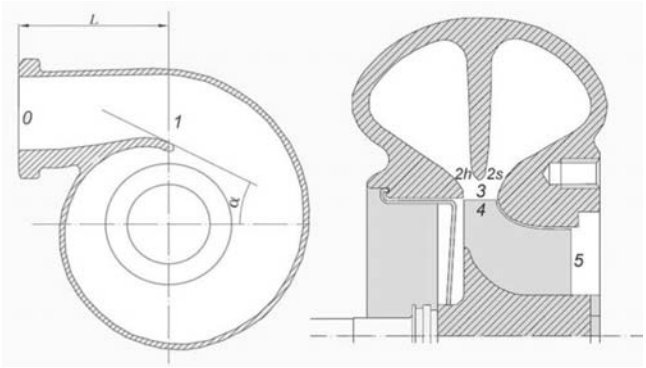

Fig. 5. Flow configuration in a twin-entry turbine

$$\frac{\dot{m}\sqrt{RT'_{0i}/\gamma}}{A_e P'_{0i}} = \sigma \cdot \cos(\beta_e) M'_e \times \left(1+\frac{(\gamma-1)}{2}M'^{2}_{e}\right)^{\frac{-(\gamma+1)}{2(\gamma-1)}} \left(1-\frac{U_i^{\,2}-U_e^{\,2}}{2c_p T'_{0i}}\right)^{\frac{(\gamma+1)}{2(\gamma-1)}} \quad (1)$$

where

$$\sigma = e^{(-\Delta s/R)} = \left(1-\frac{(\gamma-1)}{\gamma RT'_{0e}}U_t^{\,2}\Delta q\right)^{\gamma/(\gamma-1)} \quad (2)$$

$$\Delta q = \frac{h'_{0e}-h'_{0es}}{U_t^{\,2}} = \frac{\Delta h'_{0es}}{U_t^{\,2}} \quad (3)$$

$$h'_0 = h + \tfrac{1}{2}W^2, \quad T'_0 = h'_0/c_p, \quad P'_0 = P\left(\frac{T'_0}{T}\right)^{\frac{\gamma}{(\gamma-1)}}, \quad M' = W/a \tag{4}$$

Equation (1) is the non-dimensional mass flow rate equation for the moving passages (rotor). In Eq. (3), Δq is the non-dimensional energy loss in the rotor. Obviously Eq. (1) can only be used for a single-entry turbine. In this paper, this equation is modified for a twin-entry turbine. The derivation of the new formula is given in [15], which leads to

$$\frac{\dot{m}}{A_5}\sqrt{\frac{R}{\gamma}}\cdot\left(\frac{\sqrt{T'_{04,s}}}{P'_{04,s}}\right)^{Mr_s}\cdot\left(\frac{\sqrt{T'_{04,h}}}{P'_{04,h}}\right)^{Mr_h} = \sigma\cdot\cos(\beta_5)M'_5\times\left(1+\frac{\gamma-1}{2}M'^2_5\right)^{\frac{-(\gamma+1)}{2(\gamma-1)}}\times \left\{\left(1-\frac{U_4^2-U_5^2}{2c_pT'_{04,h}}\right)^{Mr_h}\left(1-\frac{U_4^2-U_5^2}{2c_pT'_{04,s}}\right)^{Mr_s}\right\}^{\frac{(\gamma+1)}{2(\gamma-1)}} \tag{5}$$

$$\sigma = \sigma_s^{\,Mr_s}\sigma_h^{\,Mr_h} \tag{6}$$

$$Mr_s = \frac{\dot{m}_s}{\dot{m}_s+\dot{m}_h}, \; Mr_h = \frac{\dot{m}_h}{\dot{m}_s+\dot{m}_h} \tag{7}$$

The loss coefficient, σ, depends on the hub and shroud side flow losses. Due to the complexity of the flow in the rotor, it is not feasible to calculate σ_s and σ_h separately. Therefore, the following model, which is obtained by combining Eqs. (2) and (7), is suggested here.

$$\sigma = \left\{\left(1-\frac{\gamma-1}{\gamma RT'_{05}}U_t^{\,2}\Delta q_s\right)^{Mr_s}\left(1-\frac{\gamma-1}{\gamma RT'_{05}}U_t^{\,2}\Delta q_h\right)^{Mr_h}\right\}^{\frac{\gamma}{(\gamma-1)}}$$

This relation can be simplified by using Taylor series expansion, and then assuming that $(\gamma-1/\gamma RT'_{05})\times U_t^{\,2}\Delta q$ is small. This assumption is reasonable for a turbine with acceptable performance. The result is

$$\sigma = \left\{1-\frac{\gamma-1}{\gamma RT'_{05}}U_t^{\,2}\left(Mr_s\Delta q_s + Mr_h\Delta q_h\right)\right\}^{\frac{\gamma}{(\gamma-1)}} \tag{8}$$

The above relation means that one can consider the effect of shroud and hub side flow losses in one relation, that is, we estimate Δq_s and Δq_h altogether. So, we can modify the correlations for the loss coefficient in single-entry turbines, to obtain losses in twin-entry turbines.

b) Friction losses

These losses are computed in the same way as the friction losses in a curved passage.

$$\Delta h'_{0s,Fr} = f_c\frac{L_H}{D_H}\frac{\overline{W}^2}{2} \tag{9}$$

where f_c is the skin friction coefficient in curved pipes and is obtained by Schlichting [16], and $\overline{W}$ is the average relative velocity in the passage. They are obtained as follows:

$$f_c = f\left[1+0.075\,\mathrm{Re}^{0.25}\sqrt{\frac{D_H}{2r_c}}\right] \tag{10}$$

and

$$\overline{W}^2 = \frac{1}{2}\left[\left(Mr_sW_{4s}+Mr_hW_{4h}\right)^2 + W_5^{\,2}\right]$$

where L_H, D_H, and r_c are hydraulic length and diameter, and mean radius of the passage, respectively. f is the skin friction coefficient for smooth-walled straight pipes.

c) Blade loading losses

According to Rodgers [12], the following correlation is used to calculate the blade loading losses.

$$\Delta h'_{0s,BL} = \frac{V_{t3}{}^2 D_t}{ZL_R} \tag{11}$$

$$V_{t3} = \left(Mr_s V_{t3,s} + Mr_h V_{t3,h}\right) \tag{12}$$

d) Clearance losses

The clearance losses are calculated from the correlation given by Rodgers [12].

$$\Delta h'_{0s,Cl} = 0.4\left(e_{Cl}/b_3\right).V_{t3}{}^2 \tag{13}$$

where V_{t3} is obtained from Eq. (12).

e) Rotor mixing losses

These extra losses exist only in twin-entry turbines and are due to the mixing of the flow in the rotor when the mass flow ratio of the shroud and hub side entries is different from its optimum value. The optimum condition is defined as the condition in which the radial velocity distribution at the rotor entry is the same as the radial velocity distribution at the rotor entry of a single-entry turbine with symmetric value. This radial velocity distribution is studied and modeled using the experimental result shown in Fig. 3. The model, which is introduced here for estimating the radial velocity component, V_{r3}, at the rotor entry is

$$r^m V_{r3} = Const \tag{14}$$

If the m is chosen between 0.41 and 0.56, Eq. (13) matches very well with the experimental data in a large portion of blade to blade space, as shown in Figs. 6 and 7. In order to make our model more precise and to make sure that it works in more general cases, ideal flow in a two-dimensional curved passage is numerically simulated and our model is tested and compared with the numerical results. Figure 8 shows the curved passage and Figs. 9 and 10 show the results. As can be shown from the figures, the model with suitable m, which is between 0.47 and 0.5, for wide rang of curvature radius, matches very well with the numerical data. From the above results, the value of m can be obtained by curve fitting. The result is

$$m = \frac{r_0}{2r_0 + 0.0348 b_3} \tag{15}$$

For the rotor of a radial flow turbine, the value of r_0 can be calculated from the following formula.

$$r_0 = B^2/A - b_3/2 \tag{16}$$

where

$$A = L_R - b_3/2 \ , \ B = r_t - \sqrt{\frac{r_{5,s}{}^2 + r_{5,h}{}^2}{2}}$$

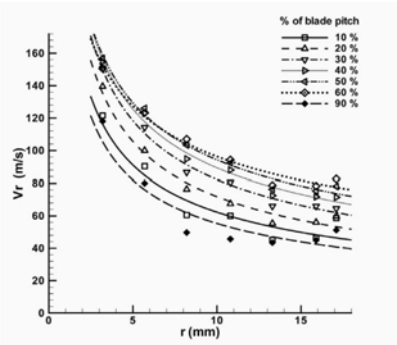

Fig. 6. Radial velocity distribution along spanwise direction at the rotor entry. Lines are model and symbols are experimental data from ref. [13]

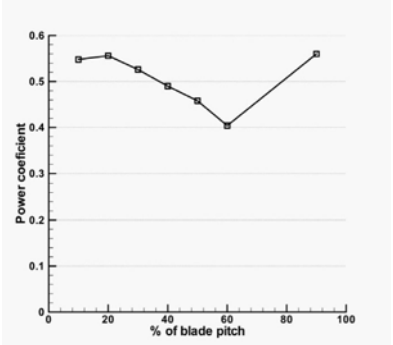

Fig. 7. Values of m for Fig. 6

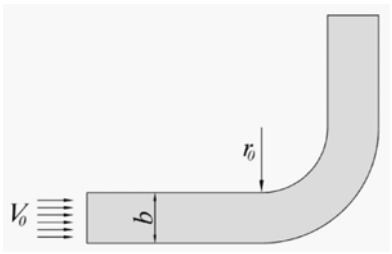

Fig. 8. Simple curved passage

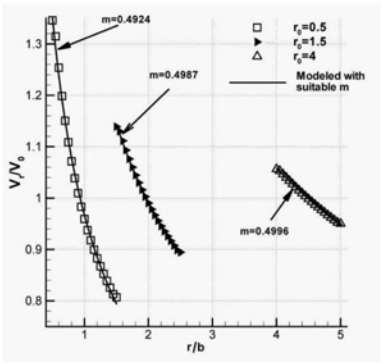

Fig. 9. Radial velocity distribution at the entrance to the passage in Fig. 8

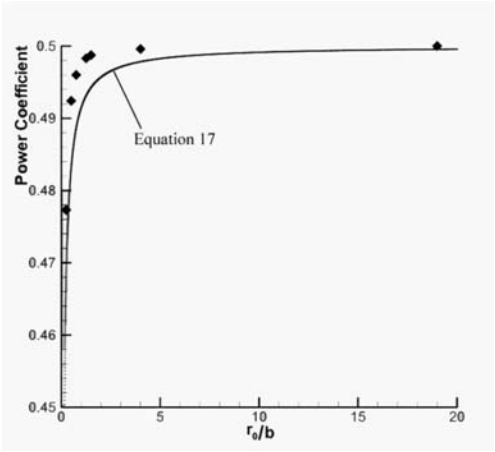

Fig. 10. Value of *m* for Fig. 9

For a twin-entry turbine, the optimum one-dimensional velocity distribution at the rotor entry is assumed as shown in Fig.11. This distribution of the velocity is simplified and a one-dimensional form of Eq. 17.

In order to calculate the rotor-mixing losses, the mixing procedure shown in Fig. 12 has been considered. In this figure, the actual radial velocity distribution at the rotor entry in a particular mass flow ratio and its optimum value, which is obtained after a hypothetical mixing, are shown at sections 1 and 2, respectively. The entropy increase for this process, Δs, can be obtained using one-dimensional thermodynamic relations. So, σ_{RM} is obtained by

$$\sigma_{RM} = e^{(-\Delta s_{RM}/R)} \tag{17}$$

As shown in Fig. 12, Δs is estimated from the mixing of the flow in a straight passage. Because the aim of this work is one-dimensional modeling, a straight passage is used rather than a curved passage which corresponds to 2-D or 3-D flows. The optimum condition for the flow leaving a straight passage is a uniform flow. So the modeling for the mixing process is performed by using Fig. 13b. That is, the flow pattern in Fig. 12 is changed to Fig. 13b. This method gives a crude estimation of Δs. Finally, the rotor loss coefficient is obtained by the following relations

$$\sigma_{rotor} = \sigma_{4-5} = \sigma \cdot \sigma_{RM} \tag{18}$$

where

$$\sigma = \left(1 - \frac{\gamma - 1}{\gamma RT'_{05}} U_t^{\,2} \Delta q\right)^{\frac{\gamma}{\gamma - 1}}, \quad \Delta q = \Delta h'_{0s} / U_t^{\,2}, \quad \Delta h'_{0s} = \sum \Delta h'_{0s} = \Delta h'_{0s,Fr} + \Delta h'_{0s,Cl} + \Delta h'_{0s,BL} \tag{19}$$

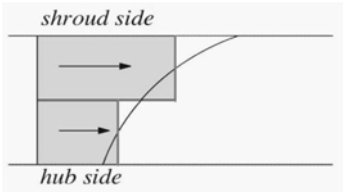

Fig. 11. One-dimensional velocity at the rotor entry

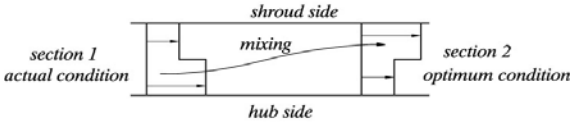

Fig. 12. Mixing process in a hypothetical passage

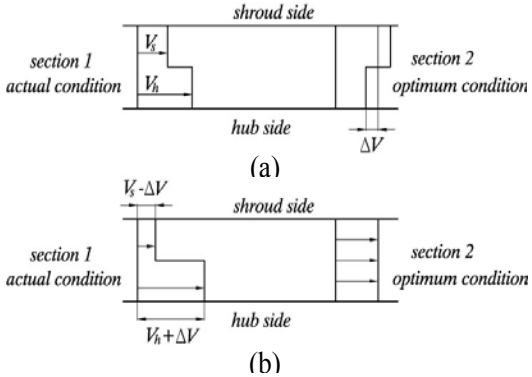

Fig. 13. Modification of the flow pattern leaves the passage shown in Fig. 12 for 1-D modeling

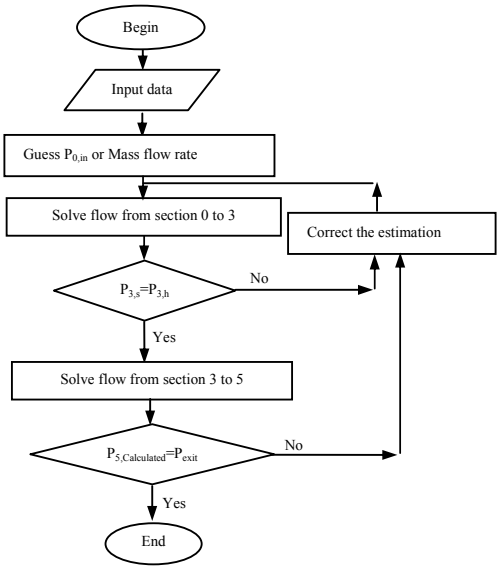

Fig. 14. Solution procedure

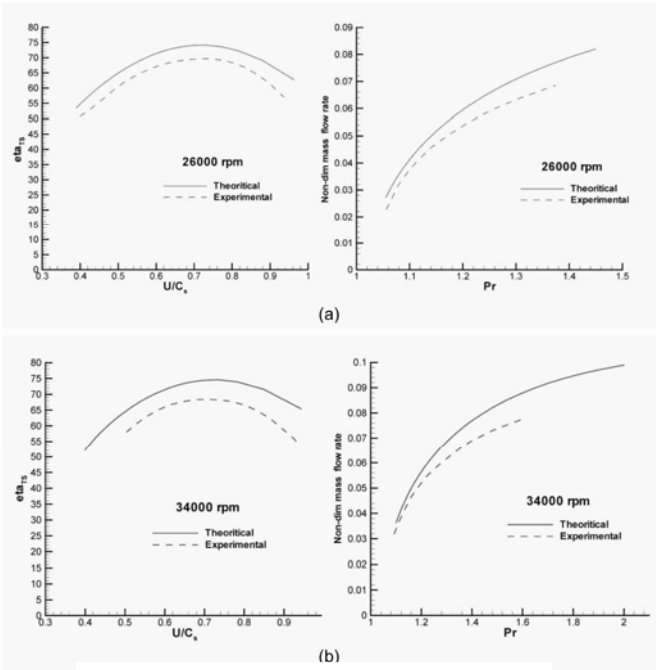

Fig. 15. Full admission results

4. SOLUTION PROCEDURE

As shown in Fig. 5, the flow passages can be divided into several parts. Flow at sections 0 to 2 is considered similar to the single-entry turbine and can be considered as stated in references [7, 8, 10, 11]. For flow between sections 2 and 3, flows in the shroud and hub side are considered separately, but their interactions are considered for calculating the exit flow angles and loss coefficients. Losses due to the friction between two streams are calculated from relations of a mixing layer between two incompressible streams. For the region between sections 3 and 4, the flows in the shroud and hub sides are modeled separately using the NASA incidence loss model [6]. Finally, the flow between sections 4 and 5 is modeled using Eq. (8).

The solution procedure is outlined in the flow chart of Fig. 14. Two types of boundary conditions are used to solve the problem. For the first case, the mass flow rate of each entry is given and for the second case, the total pressure at each entry is identified. More details of the flow solution are given in [15].

5. RESULTS

The results are compared with experimental results presented by Dale and Watson [2] and shown in Figs. 15 and 16. In Figs. 15a and 15b, the mass flow characteristics and the turbine efficiency, at full admission for 26000 rpm and 34000 rpm are shown respectively. As shown in the figures, the calculated efficiency and the mass flow rate are somewhat overestimated, but they are consistent. That is, as the efficiency is overestimated, the losses in the turbine are underestimated, thus the mass flow rate is more than what it should be.

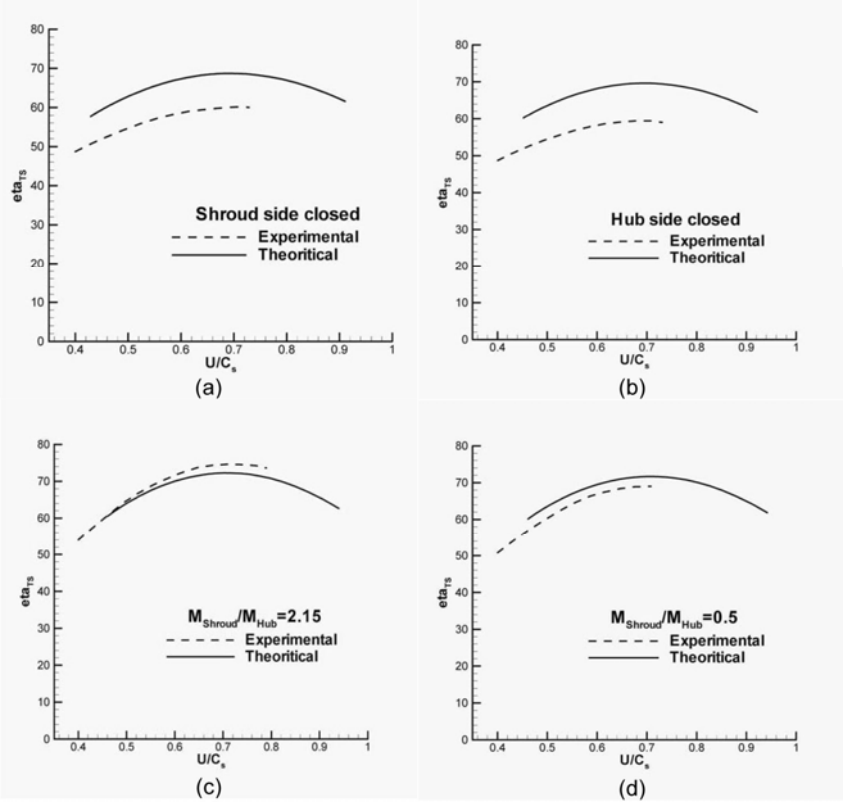

Fig. 16. Partial admission results

Figures 16a to 16d show the efficiency of the turbine at partial admission conditions for 26000 rpm, compared with the experimental data. As shown in the figures, rather good agreements are obtained. Although the trends of the curves are very well matched, there are some differences between the calculated and experimental date. These differences can be due to the following: The uncertainty in the experimental data, especially for high *U/Cs*, as shown in Fig. 17, gives some inaccurate data. This makes sense especially

because at high *U/Cs*, more disagreement between the experimental and calculated data is indicated. Secondly as shown in the figures, when the mass flow ratio of the turbine entries is far from its optimum value, the difference between the calculated and experimental results is higher. This mater suggests that the rotor mixing losses are estimated less accurately.

The difference between the efficiencies for two partial admission conditions (where the inflow is either higher in the shroud-side or in the hub-side entry) is due to the effect of the rotor-mixing losses, which are different in the two cases. As shown in Fig. 18, this difference is rather underestimated.

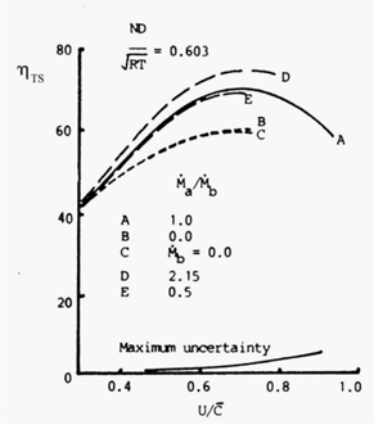

Fig. 17. Experimental results of twin-entry turbine [2]

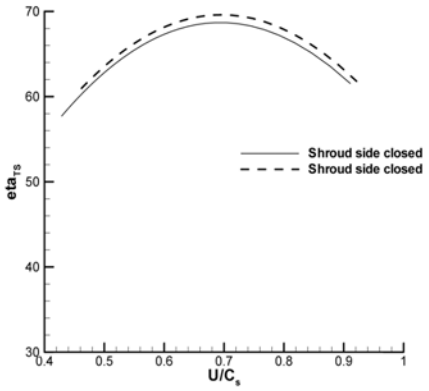

Fig. 18. The difference between the efficiency of the turbine under two partial admission conditions

6. CONCLUSION

In this paper, a one-dimensional performance prediction method is developed for modeling the flow behavior in the twin-entry turbine under partial admission and steady flow conditions. The one-dimensional equations, which are used to analyze the flow in a single-entry turbine, are modified for twin-entry turbine. Also, the rotor mixing losses are added as extra losses. The results are compared with the experimental data [2] and good agreements are obtained. The difference between the results is maybe due to the experimental uncertainty and also underestimating the rotor mixing losses. The agreements are much better when the shroud side and hub side mass flow ratio is close to its optimum value, which corresponds to when the rotor mixing losses are negligible. Because obtaining the loss coefficients in one-dimensional modeling depends on the available experimental data, the model presented here can be improved by using more experimental data.

NOMENCLATURE

A	area	$\dot{m}$	mass flow rate
a	speed of sound	P	pressure
b	blade height, Passage with	R	gas constant
C_s	isentropic expansion velocity	Re	Reynolds number
D	diameter	r	radius
e	clearance between rotor blades and shroud	r_c	radius of curvature
f	skin friction coefficient for smooth-walled straight pipe	s	entropy
		T	temperature
f_c	skin friction coefficient for curved pipe	U	blade velocity
L	length	V	absolute velocity
Mr_h	shroud side entry mass flow rate to total mass flow rate ratio	W	relative flow velocity
		Z	rotor blade numbers
Mr_s	hub side entry mass flow rate to total mass flow rate ratio	β	relative angle of flow
		γ	specific heat ratio

η	efficiency
σ	loss coefficient

Subscripts

0	turbine entry, Stagnation properties
0-5	locations within turbine (see Fig.5)
BL	blade loading
Cl	clearance
E	passage exit
Fr	friction
h	hub side
H	hydraulic
i	passage inlet
RM	rotor mixing
R	rotor
r	radial direction
s	isentropic process, Shroud side
TS	total to static
t	blade tip, Tangential direction

REFERENCES

1. Pischinger, F. & Wunsche, A. (1977). The characteristic behaviour of radial turbines and its influence on the turbocharging process. *MAC Conference*, Tokyo.
2. Dale, A. & Watson, N. (1986). Vaneless radial turbocharger turbine performance. *IMechE Conference Paper C110/86 in Turbocharging and Turbochargers*.
3. Capobianco, M. & Gambarotta, A. (1993). Performance of a twin-entry automotive turbocharger turbine. ASME Paper 93- ICE-2.
4. Baines, N. C. & Yeo, J. H. (1991). Flow in a radial turbine under equal and partial admission conditions. IMechE Paper C423/002.
5. Lymberopoulos, N., Baines, N. C. & Watson, N. (1988). Flow in single and twin entry turbine volutes. ASME Paper 88-GT-59.
6. Futral, S. M. & Wasserbauer, A. C. (1965). Off-design performance prediction with experimental verification for a radial-inflow turbine. NASA TN D-262.
7. Wallace, F. J., Baines, N. C. & Whitfield, A. (1976). A unified approach to the one-dimensional analysis and design of radial and mixed flow turbines. ASME paper 76-GT-100.
8. Abidat, M., Hachemi, M. K. & Baines, N. C. (1988). Prediction of the steady and non-steady flow performance of a highly loaded mixed flow turbine. *Proceedings of Institution of Mechanical Engineers, Part A*: *Journal of Power and Energy, 212*, 173-184.
9. Chen, H. & Baines, N. C. (1994). The aerodynamic loading of radial and mixed-flow turbines. *International Journal of Mechanical Science, 36*, 63-79.
10. Whitfield, A. & Baines, N. C. (1976). A general computer solution for radial and mixed-flow turbomachine performance prediction. *International Journal of Mechanical Science, 18*, 179-184.
11. Whitfield, A. & Baines, N. C. (1990). *Design of radial turbomachines*, Longman Scientific & Technical.
12. Rodgers, C. (1987). Mainline performance prediction of radial inflow turbines. VKI Lecture series 1987-07.
13. Benisek, E. F. & Struble, A. G. (1990). Laser velocity measurements in a turbocharger turbine. *IMechE* paper C405/009.
14. Zangeneh-Kazemi, M., Dawes, W. N. & Hawthorne, W. R. (1988). Three dimensional flow in radial-inflow turbines. ASME Paper 88-GT-103.
15. Ghassemi, Sh. (2001). Experimental and theoretical investigation of twin-entry turbine behavior. Mechanical Engineering Department, Isfahan University of Technology, Isfahan, Iran, M.Sc. Thesis in Farsi.
16. Schlichting, H. (1968). Translated by Kestin, J., *Boundary-layer theory*, Sixth edition, McGraw-Hill Book Company.

پيش بيني عملكرد توربينهاي دو ورودي توربوچارجر

شهرام قاسمی، ابراهیم شیرانی[1] و علی حاجیلویي بنیسی[2]

[1]دانشکده مهندسی مکانیک، دانشگاه صنعتی اصفهان، اصفهان، جمهوری اسلامی ایران

[2]دانشکده مهندسی مکانیک، دانشگاه صنعتی شریف، تهران، جمهوری اسلامی ایران

چکیده: در اين مقاله، عملكرد توربينهاي دو ورودي جريان شعاعي تحت شرايط جريان دايم و ورودي نامساوي مدلسازي شده است. روشي كه طي اين تحقيق توسعه پيدا كرده براساس روش يك بعدي پيش بيني عملكرد مي باشد. در مدلسازي يك بعدي، فرض مي شود كه خواص جريان در صفحة عمود بر جهت جريان يكسان باشد. چنين فرضي در تضاد با مشخصات جريان در ورودي روتور يك توربين دو ورودي با جريان ورودي نامساوي است. در اين مطالعه، به منظور تحليل رفتار توربين دو ورودي، روش مدلسازي يك بعدي در توربينهاي تك ورودي بهينه شده است. بويژه ضريب تلفات ناشي از اصطكاك، لقي و بارگذاري پره كه قبلاً براي توربينهاي تك ورودي گسترش پيدا كرده بود اصلاح شدند. همچنين تلفات اضافي به دليل شرايط جريان در ورودي توربينها و اختلاط جريان داخل روتور درنظر گرفته شد. در واقع در يك توربين تك ورودي با محفظة متقارن جريان تمايل دارد كه از سمت شرود بگذرد. يك رابطه براي توزيع جريان شعاعي در ورودي چنين توربيني بدست آمد و فرض شد كه حالت بهينه را نشان مي دهد، سپس تلفات اختلاط روتور تخمين زده شد. در نهايت يك مدل بر اساس فرضيات فوق توسعه پيدا كرد. نتايج بدست آمده از مدل با نتايج تجربي مقايسه شدند و هماهنگي خوبي بدست آمد. در اين مقاله همچنين رفتارهاي خاص جريان در توربينهاي دو ورودي مورد بررسي قرار گرفته و تعابير فيزيكي براي اين رفتارها ارائه شده است.