# ANALYSING SHOCK LOSSES THROUGHOUT THE IMPELLER OF THE CENTERFUGAL FAN

Ghassan Fadhil Muhammad
College of Engineering
Kufa University

Zaid M. Hassan
Technical College
Najaf

Muayad Muhammad Sadiq
College of Engineering
Kufa University

**Abstract**

The centrifugal fan is the most widely used because it can efficiently move large quantity of the gas over a wide range of pressures. The flow that enters throughout the impeller is analyzed by using a streamline curvature technique, this method used for a compressor, by neglect the term of gas density change. The turbulent flow is modeled by using algebraic eddy viscous model which based on the mixing length, the turbulent model is applied on the hub to shroud stream surface. The shock in the present work is form by increasing the volume flow rate in put to the impeller .The impeller that chosen in this work is the impeller of the pre-heater fan in the heating system of clinker in AL-Muthanna factory of cement, which has back curved blade. The results prove that the increasing in the volume flow rate (forming shock losses) has an effect on the velocity profile and static pressure, so that where the volume flow rate increase the velocity profile increase and static pressure decrease.

## تحليل خسائر الخنق خلال دافعة مروحة طاردة مركزية

غسان فاضل محمد حسين
كلية الهندسة -جامعة الكوفة

زيد معن حسن
الكلية التقنية- النجف

مؤيد محمد صادق
كلية الهندسة - جامعة الكوفة

**الخلاصة**

إن الاستخدام الواسع للمروحة الطاردة المركزية هو بسبب كفأتها العالية لنقل كميات كبيرة من الغاز ولمديات ضغط مختلفه. لقد تم تحليل الجريان الداخل إلى البشارة (الدافعة) باستخدام طريقة تقوس خطوط الجريان، هذه الطريقة تستخدم عادة للضاغطات، بواسطة إهمال كل ما هو يتعلق بتغير كثافة الغاز. تم تحليل الجريان بواسطة استخدام طرقة تقوس الجريان أما موديل الجريان الاضطرابي فقد تم استخدام طرقة رياضية هي (طريقة الدوامات اللزجة الرياضي( algebraic eddy viscous model) و قد تم تطبيق موديل الجريان الاضطرابي على سطح الجريان قمة - قعر (hub to shroud). إن الدافعة المختارة في هذا البحث هي دافعه مروحة التسخين المسبق في منظومة تسخين الكلنكر في معمل سمنت المثنى حيث إن ريش هذه المروحة هي من نوع الريش المرتدة للخلف .إن النتائج التي تم الحصول عليها من هذا البحث بينت بان الزيادة في معدل التدفق (الذي يكون الخنق في هذه الحالة )يكون له تأثير على نسبة السرعة و على الضغط الستاتيكي حيث كلما زاد معدل التدفق زادت نسبة السرعة (velocity profile ) وقل الضغط الستاتيكي .

## Nomenclature

| Symbol | Description | Unit. |
|---|---|---|
| b | blade width | m |
| T | gas temperature | K |
| m | distance along meridional streamline | m |
| r | radius from axis of rotation | m |
| $r_c$ | radius of streamline curvature | m |
| s | distance along quasi-orthogonal in meridional | m |
| z | axial distance | m |

**Greek Symbols**

| | | |
|---|---|---|
| $\alpha$ | angle between meridional streamline and z-axis | rad |
| $\beta$ | angle between relative velocity and merdional streamline | rad |
| $\theta$ | angular coordinate | rad |
| $\Delta\theta$ | angle between blade surfaces | rad |
| $\Delta n$ | distance between adjacent streamline in the meridional plan | m |
| $\mu$ | viscosity | kg/s.m |
| $\rho$ | density | $kg/m^3$ |
| $\tau$ | shear stress | $N/m^2$ |

**Subscripts**

| | |
|---|---|
| i | inlet |
| j | number of streamline |
| m | component in direction of meridional streamline |
| r | radial component |
| $r\theta$ , $\theta z$ , zr etc . | component of shear stress |
| z | axial component |
| $\theta$ | tangential component |

## Introduction

Fluid particles will follow a certain path, a streamline through the impeller. The first rule of a good impeller design is that the same energy, and therefore the same angular momentum, should be imparted to fluid on each of the streamline[1,2,3].

In order to explain the shock losses on the performance of the centrifugal fan impeller firstly must be analysis the flow in the impeller of centrifugal fan impeller, i.e. as it was assumed laminar flow, then add turbulent flow to determine the effect of shock losses.

The analysis of the flow in the impeller by using streamlines curvature method by using hub to shroud section. The analysis of turbulent flow by using algebraic model depending mixing length.
The shock losses occur when the flow increase above the normal value, i.e., it a result of any variation from the normal of volume flow rate. These losses have two kinds, according to the place occurring it, as followings:

1. The first type of these losses occurs in the impeller entrance.
2. The second type of these losses occurs due to guide vanes.

**Flow Analysis**

**1- Velocity Gradient Equation**

The flow distribution is calculated by solving the velocity gradient equation for the directional derivative of the relative velocity along the quasi-orthogonal on (H-S) stream surfaces.

For plane (H-S), the velocity field is obtained by solving by the velocity gradient along the quasi-orthogonal in the meridional plane. This equation is shown as the following[4]:

$$\frac{dw}{ds}=\left[A\frac{dr}{ds}+B\frac{dz}{ds}\right]w+C\frac{dr}{ds}+D\frac{dz}{ds}+\left[\frac{dI}{ds}-T\frac{dS}{ds}\right]\frac{1}{w}-\left[E\frac{dr}{ds}+F\frac{dz}{ds}\right]\frac{1}{w} \quad (1)$$

Where:-

$$\left.\begin{aligned}
A&=\frac{\cos\alpha\cdot\cos^2\beta}{r_c}-\frac{\sin^2\beta}{r}\cdot\sin\alpha\cdot\cos\beta\left(\frac{\partial\theta}{\partial r}\right)_f\\
B&=-\frac{\sin\alpha\cdot\cos^2\beta}{r_c}+\sin\alpha\cdot\sin\beta\cdot\cos\beta\left(\frac{\partial\theta}{\partial z}\right)_f\\
C&=\sin\alpha\cdot\cos\beta\,\frac{dw_m}{dm}-2w\sin\beta+r\cos\beta\left[\frac{dw\theta}{dm}+2w\sin\alpha\right]\left[\frac{\partial\theta}{\partial r}\right]_f\\
D&=\cos\alpha\cdot\cos\beta\,\frac{dw_m}{dm}+r\cos\beta\left[\frac{dw_\theta}{dm}+2w\sin\alpha\right]\left[\frac{\partial\theta}{\partial z}\right]_f\\
E&=\frac{1}{\rho}\frac{\partial}{\partial r}\left(2\mu_{eff}\frac{\partial w_r}{\partial r}\right)+\frac{1}{\rho}\frac{\partial}{\partial z}(\tau_{zr})+\frac{2\mu_{eff}}{\rho r}\left(\frac{\partial w_r}{\partial r}-\frac{w_r}{r}\right)+\\
&\quad\left[\frac{r}{\rho}\frac{\partial}{\partial z}(\tau_{z\theta})+\frac{r}{\rho}\frac{\partial}{\partial r}(\tau_{r\theta})+\frac{2}{\rho}(\tau_{r\theta})\right]\left(\frac{\partial\theta}{\partial r}\right)_f\\
F&=\frac{1}{\rho}\frac{\partial}{\partial z}\left(2\mu_{eff}\frac{\partial w}{\partial z}\right)+\frac{1}{\rho r}\frac{\partial}{\partial r}(r\tau_{zr})+\left[\frac{r}{\rho}\frac{\partial}{\partial z}(\tau_{z\theta})+\frac{r}{\rho}\frac{\partial}{\partial r}(\tau_{r\theta})+\frac{2}{\rho}(\tau_{r\theta})\right]\left(\frac{\partial\theta}{\partial z}\right)_f
\end{aligned}\right\} \quad (2)$$

The coordinate system and the notation are shown in **Figure (3).** The distance (s) is along the quasi-orthogonal (q-o) and w is considered as a function of (s) a lone.

The relative velocity components are[4]:

$$\left.\begin{aligned}
Wm&=W\cos(\beta)\\
W\theta&=W\sin(\beta)\\
Wr&=W\sin(\alpha)\\
Wz&=W\cos(\alpha)
\end{aligned}\right\} \quad (3)$$

**2- Continuity Equation**

In addition to Eq. (1), the continuity equation must be satisfied from blade to blade. This is done by solving the Eq. (1) simultaneously with the continuity equation in its integral form along each quasi-orthogonal until the calculated mass flow across any quasi-orthogonal is equal to the specified mass flow through the turbomachine [1, 2 & 3].

The form of the continuity equation used is as follows [1, 2, and 4]:

$$\dot{m} = z\int_{0}^{s} \rho \cdot w_n \cdot r \cdot \Delta\theta \cdot ds \qquad (4)$$

$$Wn = Wm \cos(\psi-\alpha) \qquad (5)$$

$$\psi = 90^{\circ}$$

$$\alpha = arc \quad \tan\left(\frac{dr}{dz}\right)$$

$\Delta\theta$ is the angular distance between blades.

$$\Delta\theta = \frac{2\pi}{z} - \frac{tn}{r} \qquad (6)$$

## Fan Efficiency

The efficiency of any turbo machine is a ratio of output power to mechanical input power and is usually expressed as a percentage.

The output power is define as the product of volume flow rate and the pressure[4].

$$\text{Fan Static efficiency } \zeta_s = \frac{air \quad power(static)}{input \quad power} * 100\%$$

$$= \frac{Q.P_s}{input \quad power} 100\% \qquad (7)$$

## Viscous Calculation

The velocity gradient eq.(1), for H-S plane, included a viscous terms to take into account the effect of viscosity and turbulence. The shear stress in these terms is a total shear stress (i.e. is the sum of the shears due to-molecules-viscous action-and shear stress due to turbulence action).

The viscosity model used in the present work is similar to that used in reference [5], with some necessary changes in order to fit with fan, this model depends upon the assumption of the effective viscosity, and on algebraic model which used to calculate the turbulent viscosity, so that the determination of the total shear stress is based on the assumption of effect viscosity [6].

$$\mu\, eff = \mu L + \mu T \qquad (8)$$

$\mu$ is the laminar viscosity which is constant through the flow field, and it is determined [6] by:

$$\mu_L = 1.458 \times 10^{-6} \left[ \frac{T^{1.5}}{110.4 + T} \right] \qquad (9)$$

While, in order to find the turbulent viscosity, turbulent model is required Algebraic eddy viscosity models are based on the "mixing length concept. The mixing length model" can be written as

$$\mu_T = \rho L^2 \left[ \left( \frac{\partial W_y}{\partial Y} \right)^2 + \left( \frac{\partial W_\theta}{\partial Y} \right)^2 \right]^{1/2} \qquad (10)$$

Where:

L = K.y [1-exp(y+/A+)] (a)

$$y^+ = \frac{\sqrt{\rho \cdot \tau_w} \cdot y}{\mu_l} \qquad (b)$$

**<u>Turbulent Flow:</u>**

**1-Basic Concept of Turbulent Flow:**

Most turbomachinary flow is turbulent, with laminar and transitional region occurring near the edge of the blade. Turbulence influences the aerodynamic and thermodynamic performance of turbomachinary, and therefore its consideration is critical to turbomachinary analysis.

Turbulence is characterized by irregular or random fluctuations. It often originates as instability of high-Reynolds number (laminar flow that results in transition to turbulent.Turbulence is very complex phenomenon, from both measurement and analysis of view.

Because of its random variation in space of time, much analysis relies on statistical approaches. The equations governing the statistical properties of a turbulent flow involve more unknowns than the equations available; additional terms, which appear in these equations, must be modeled, and therefore the problem of including turbulence in the formulation is often referred to as the closure problem. Statistical approaches vary cured purely empirical) models to highly sophisticated (often impractical) models.

Turbulence, in addition to being random or irregular, is three-dimensional. Velocity fluctuations exist in all direction, even if the mean flow is one-or-two dimensional.

Turbulence is diffusive and dissipative. The former phenomenon gives rise to rapid mixing and increased rates of moment, heat and mass transfer. Dissipation is characterized by deformation work associated with velocity fluctuations, and it increases the internal energy of the fluid at the expense of kinetic energy in the flow mean flow and turbulence[4].

**2 Algebraic Viscosity Model:**

Algebraic eddy viscosity models are based on the "mixing length concept. The mixing length model" can be written as

$$\mu_T = 2\rho L^2 \sqrt{S_{ij} S'_{ij}} \qquad (11)$$

Where:

L is termed the mixing length.

Sij is representative of the large-scale motion in a turbulent flow.

In algebraic eddy viscosity models, an algebraic expression for the mixing length, L, is provide for closure, several groups have developed algebraic eddy viscosity models (Cebeci and Smith, 1974; Crawford and Kays, 1975; Mellor and herring, 1973 Patankar and Spalding, 1970).

One of the most widely used models for the engineering calculation of boundary layers is that due to Cebeci & Smith, Later modified by Baldwin and Lomax (1978). The model proposed by Baldwin and Lomax has wider applications and avoids the necessity of finding the edge of the boundary layer. Both of these models incorporate empirical relations. The Ballwin and Lomax model is given by:

$$\mu_T = \rho L^2 \left[ \left( \frac{\partial W_y}{\partial Y} \right)^2 + \left( \frac{\partial W_\theta}{\partial Y} \right)^2 \right]^{1/2} \tag{12}$$

The above equation solved for each quasi-orthogonal in stream flow.
Where:

$$L = K.y\ [1-\exp(y+/A+)] \tag{a}$$

$$y^+ = \frac{\sqrt{\rho \cdot \tau_w} \cdot y}{\mu_l} \tag{b}$$

Where y is the distance normal to the wall and y+ is non-dimensional coordinate, A* is constant and equal to (A+ = 26).
K: is von Karman constant and it has a value of 0.41
Therefore

$$L = Ky = \left[ 1 - \exp\left( \frac{-y\sqrt{\rho \cdot \tau_w}}{26\mu_l} \right) \right] \tag{c}$$

The relation (c) is the (Van Driest) formula [4] and it is used in the near-wall regions of the boundary layer. The quantity in paren these is worked as a damping function used to bridge the gap between the fully turbulent region where (L = 0.41y) and the laminar sublayer where (L→0).

In may turbomachinary also in the present work the-wall mesh points in the flow field is for from the laminar sub layer. Therefore, the damping function in (c) can be dropped, and the mixing length (L) is taken as the smaller of

$$\left. \begin{array}{l} L = 0.41y \\ L = 0.08\delta \end{array} \right] \tag{13}$$

Where δ is the boundary layer thickness. In fan, when the flow incompressible, so that the temperature assumed constant, therefore to calculate the boundary layer thickness is determine as follow:

A local dimension stagnation pressure $\Delta P_L^*$ is defined as:

$$\Delta P_L^* = \frac{P^* - P_r}{P_{\max}^* - P_r} \tag{14}$$

Where:

P* is the rotary stagnation pressure.

Pr is the reduced static pressure.
$P^{*}_{max}$ is the maximum value of P*.
On the current quasi-orthogonal. P* and Pr are calculated using the relation as following :

$$P_r = P - \frac{1}{2}\rho \cdot U^2 \quad (15)$$

$$P^* = P + \frac{1}{2}\rho (W^2 - U^2) \quad (16)$$

Now, at each mesh point on a quasi-orthogonal, a determined the value of $\Delta P_L^*$ which ranging from a maximum of (1) when $P^* = P^*_{max}$ , to a minimum of (0) when the relative velocity (w) equal to zero at the walls (i.e. at the hub and the shroud).

The following relative is used to checked, at each mesh point quasi-orthogonal stating from the wall (i.e. the hub or the shroud) and proceeding to the passage center.

$$\left| \frac{\partial (\Delta P_2^*)}{\partial y} \right| > \frac{1}{passage \quad width} \quad (17)$$

At given mesh point, if the equation (3-34) is true, then the mesh point is within the boundary layer; else, the mesh point is outside or at the of boundary layer.

An interpolation procedure can be used to determined the edge of boundary layer (when gradient $\Delta P_L^*$ equal to one over the passage width).

**Shock Losses**

As pointed in above , the shock losses is divided into two types, the first type dial with the increasing of the flow and the second dial with add guide vanes. These two type cause losses in the performance of impeller of fan, in the present research the first kind was study only [5].

**Impeller Entrance Loss:**

The direction of the relative flow will no longer coincide with the blade angle if the normal volume flow rate is varied.

As a result of any variation from the normal volume flow rate, a losses called a "shock loss" arises. This can be reading understand from the entry velocity triangle shown in **Figure** (7) increase in the normal volume flow rate from Q to $Q_X$ required an increase in the meridional velocity from Vf1 to Vf1x.

Hence

$$V_{f1x} = V_{f1}\frac{Q_x}{Q} \quad (18)$$

As the air should enter radially in case in both case, the blade angle must be increased to β1 in order to change the direction of the before entry to the tangential flow into the impeller. The deflection β1 causes so-called shock component (VC ) if Vf1 is to remain constant. One can prove by the theorem of momentum that at impact the velocity head of the geometrical gradient be transformed to a pressure loss, i.e.

$$\Delta P_t = \frac{\rho}{2} V_s^2 \tag{19}$$

In the case of finite number of blades, the loss due to shock may be some that less. This loss consider in term of a coefficient μshock which is in the order of 0.7-0.9.

$$\Delta P_t = \mu_{shock} \frac{\rho}{2} V_s^2 \tag{20}$$

A cording to figure (3-6), the value of Vs calculates as the following equation:

$$V_s = U_1 \left[ \frac{V_{f1x}}{V_{f1}} - 1 \right] = U_1 \left[ \frac{Q_x}{Q} - 1 \right] \tag{21}$$

By substituting $U_1 = U_2 \left( \frac{R_1}{R_2} \right)$, then

$$V_s = U_2 \left( \frac{R_1}{R_2} \right) \left[ \frac{Q_x}{Q} - 1 \right]$$

(22)

The losses are given by:

$$\Delta P_t = \mu_{shock} \frac{\rho}{2} U_2^2 \left( \frac{R_1}{R_2} \right)^2 \left[ \frac{Q_x}{Q} - 1 \right]^2 \tag{23}$$

for the introduction of coefficient take $\psi_t = \dfrac{P_t}{\frac{1}{2} \rho \cdot V_2^2} \Rightarrow \frac{1}{2} \rho \cdot V_2^2 = \dfrac{P_t}{\psi_t}$

substitution $$\frac{\Delta P_t}{P_t} = \mu_{shock} \frac{1}{\psi_t} \left( \frac{R_1}{R_2} \right)^2 \left[ \frac{Q_x}{Q} - 1 \right]^2 \tag{24}$$

**2 Performance Coefficient of the Fan:**

There are three kind of coefficient are used for comparison of fans [2,3,5], as follow:

1- Pressure Coefficient (**Ψ**).

There are two kinds of pressure coefficient:

A- Total Pressure Coefficient (**Ψt**).

It is defined as the ratio of the fan total pressure to the dynamic pressure of impeller peripheral velocity.

$$\psi_t = \frac{\text{Fan total pressure}}{0.5 \rho \cdot \text{V}^2} \tag{25}$$

B- Static Pressure Coefficient (**Ψ**s).

It is defined as the ratio of the fan static pressure to the dynamic pressure of the impeller peripheral velocity.

$$\psi_t = \frac{\text{Fan total pressure}}{0.5\rho \cdot \text{V}^2} \quad (26)$$

2- Volume Coefficient ( $\phi$ ):

It is defined as the ratio of flow velocity to the impeller peripheral velocity.

$$\phi = \frac{V_f}{V} \quad (27)$$

$$U = \frac{2\pi N}{60} r \quad (28)$$

3- Power Coefficient (λ).

It is defined as the ratio of result of multiplied a volume coefficient and total efficiency of the fan.

$$\lambda = \frac{\phi \cdot \psi_t}{\eta} \quad (29)$$

**Results and Discussion**

In this section, the effect of shock losses is studying on both the velocity profile and static pressure.

**1 First Case:**

This case represented the original case of ref.[4] and it consist of 14 blades and 34° angle of blades.

**Figure** (8) shows the relation between the velocity profile and fractional distance along each quasi-orthogonal from first quasi-orthogonal to eleventh quasi-orthogonal. The figure show the effect of shock losses on the velocity profile, from figure as the flow rate increase velocity profile increase because the relative velocity increase but this increase in velocity profile from losses in the efficiency of the impeller. The value of flow rate choose are 81.4, 85, 90 and 95 $m^3/s$, the first volume flow rate represented the natural flow rate calculated from the dimensions of impeller but the other represented the volume flow rate were chosen to study the shock losses.

**Figure (9)** shows the relation between the static pressure and fractional distance along the odd quasi-orthogonal lines, the figure shown the effect of shock losses on the velocity profile. The volume flow rate which choose to study the effect of losses are 81.4, 85, 90 and 95 m3/s where the first volume flow rate represented the volume flow rate calculated by the dimensions of impeller but the other are chosen to calculate the effect of shock losses. The figure show that shock losses reduce the static pressure because it is increases the relative velocity and as according to the equation (7).

**2 Second Case:**

again as pointed above, this case presented the best case ref.[4], which is consist of 10 blades and 34° inlet angle of blade.

**Figure (10)** represents the effect of shock losses on the relation between the velocity profile and fractional distance along each quasi-orthogonal from first quasi-orthogonal to eleventh quasi-orthogonal. The volume flow rate, which choose to study the effect of shock losses are 91.743, 95, 100 and 105 m3/s where 91.743 represented the volume flow rate calculated from the dimension of impeller but the other used to study effect of shock losses. The shock losses mean that increasing in volume flow rate and as the volume flow rate increase the relative velocity increase so that the shock losses tend to increases the velocity profile.

**Figure (11)** shows the effect of shock losses on the relation between static pressure and fractional distance along odd quasi-orthogonal. The volume flow rate which chosen are the same for the above, the shock losses reduces the static pressure because it is increases the relative velocity according to equation (7).

**<u>Conclusion</u>**

The two-dimensional turbulent flow calculation has been applied to the cases of ref.[4], first and second cases as pointed above . The velocity of the present calculation method was verified by comparing with result of ref. [4], and experimental result from this comparison, the following behaviors have been observed.

1. The shock losses increasing velocity profile because are depended on the increasing of the volume flow rate.
2. The shock losses decreasing static pressure because it are increasing the relative velocity because are depended on the increasing in the volume flow rate.

**References**

1-Katsanis, T., “Use of Arbitrary Quasi-Orthogonal for Calculating Flow Distribution in the Meridional Plan of A Turbomachine”, NASA ,TN D-2564, 1964

2-Abu-Tabikh, M., ”The Utilization of Turbulence Model in Fluid Flow Calculation Through Turbomachines”, M.Sc Thesis, Department of Mechanical Engineering, University of Technology, September, 1990

3-Novak, R. A., ”Streamline Curvature Computing Procedures for Fluid Flow Problem”, J. of Engineering for Power, ASME, PP. 478-490, 1967

4- Duabel, H.A.," Use Of Streamline Curvature Technique To Re-Design Centrifugal Fan Impeller",M.Sc. Thesis, Department of Mechanical Engineering, University of Kufa, March, 2002

5-Howard, J. H. G., Patankar, S. V. and Bordynuik, R. M., “ Flow Prediction in Rotating Ducts Using Coriolis-Modified Turbulence Models”, J. of Fluid Engineering , ASME Vol. 102, PP. 456-461, 1980

6-J. L. Walsh, J.H. Ahlberg and E. N. Nilson, ”Best Approximation Properties at the Spline Fit “, J. of Mathematics and Mechanics, Vol.11, No,2, PP.225-234, 1962

7-Jorgenson, P. C. E. and Chima, R. V., “ Explicit Runge-Kutta Method for Unsteady Rotor/Stator Interaction “, AIAA Journal, Vol.27, No.6, 1989, PP.743-749

8-Howard, J. H. G. and Osborne, C., ” A Centrifugal Compressor Flow Analysis Employing a Jet-Wake Passage Flow Model”, J. of Fluid Engineering ,ASME PP.141-147, 1977,

9-Davis, W. R. and Millar, D. A. J., “ A Comparison of the Matrix and Streamline Curvature Methods of Axial Flow Turbomachinary Analysis From a User’s Point of Voew”, J. of Engineering for Power , ASME PP.549-560, 1975

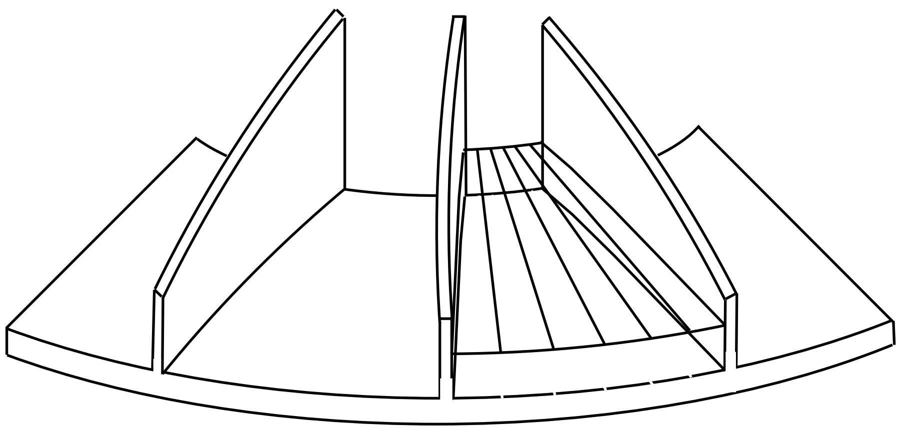


Fig.1 Two-dimentional streamline surface for flow analysis

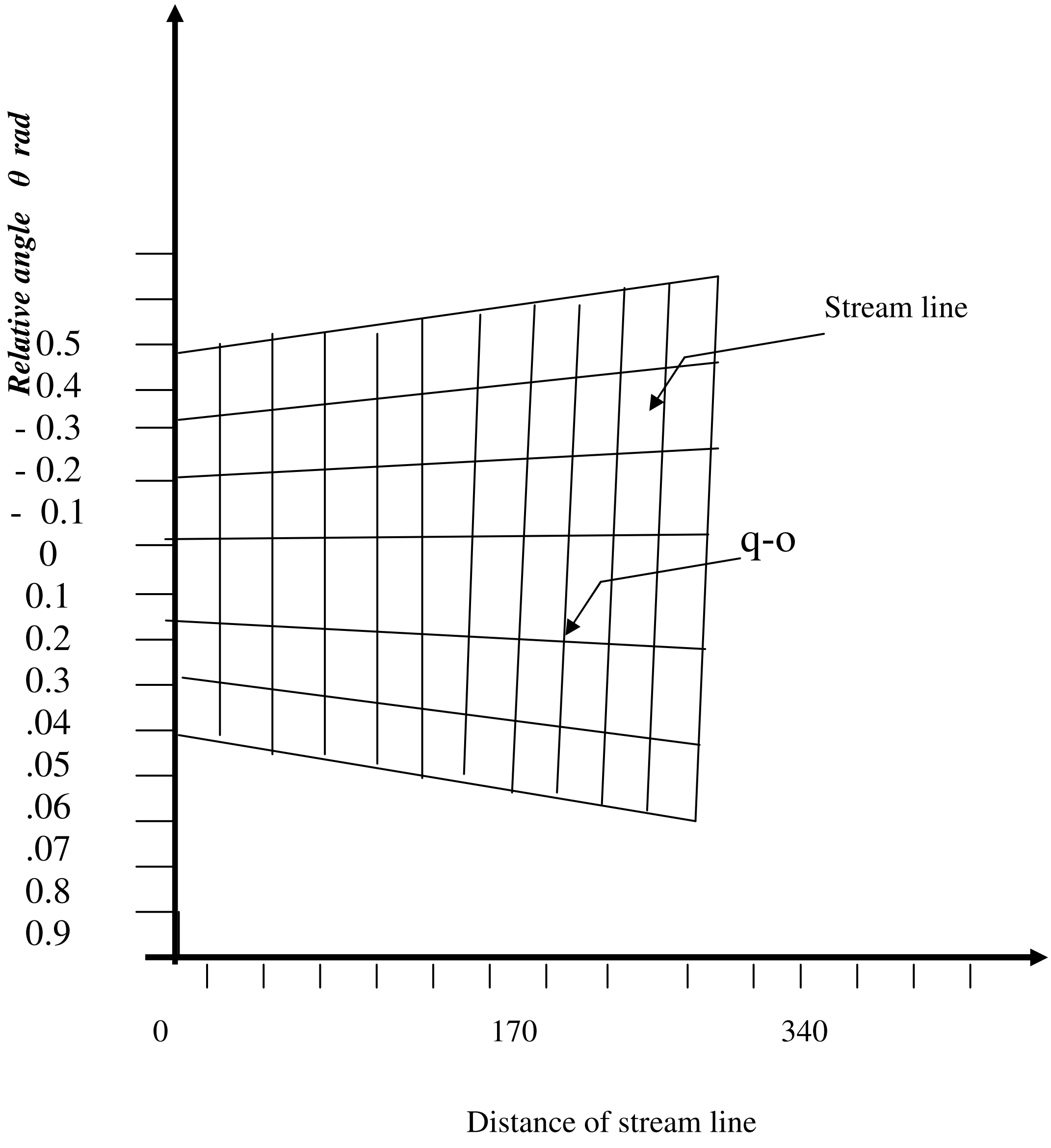



Figure 2 Meridional plane

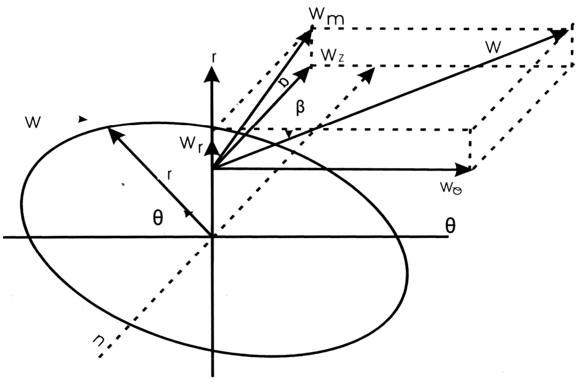



Figure 3 Coordinate system and velocity component

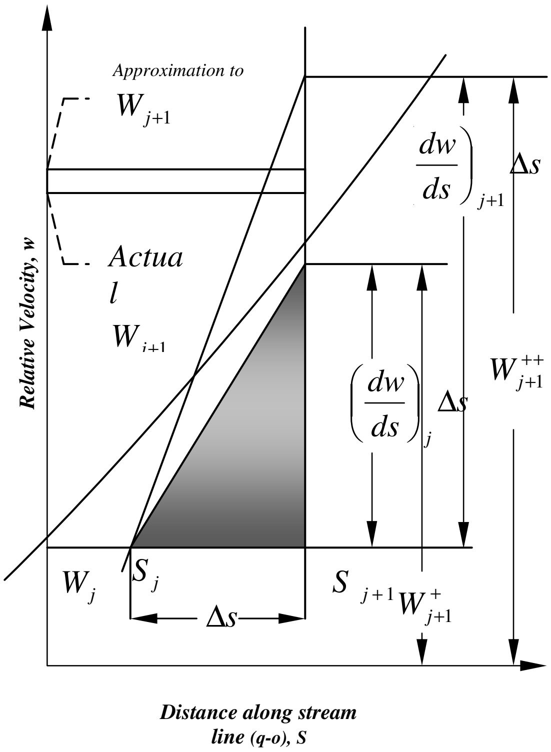



Figure 4 Approximation to solution of equation (14)

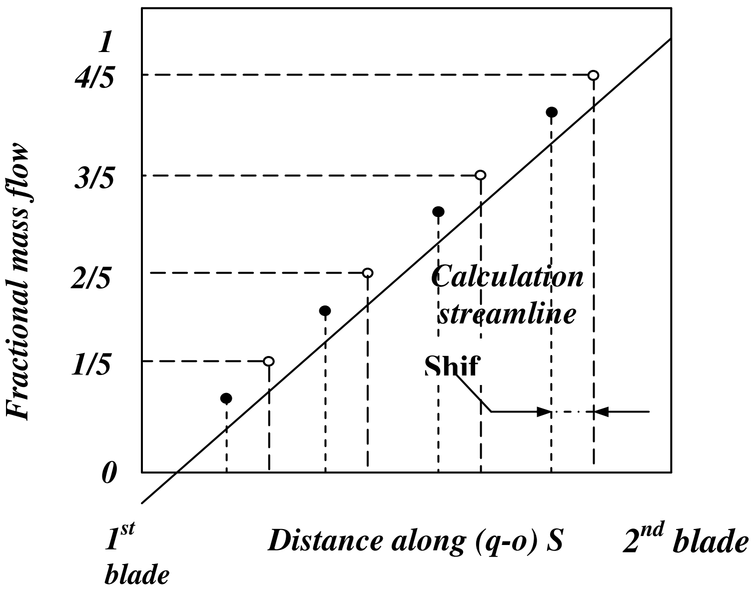



Figure 5 Mass flow distribution along q-o .

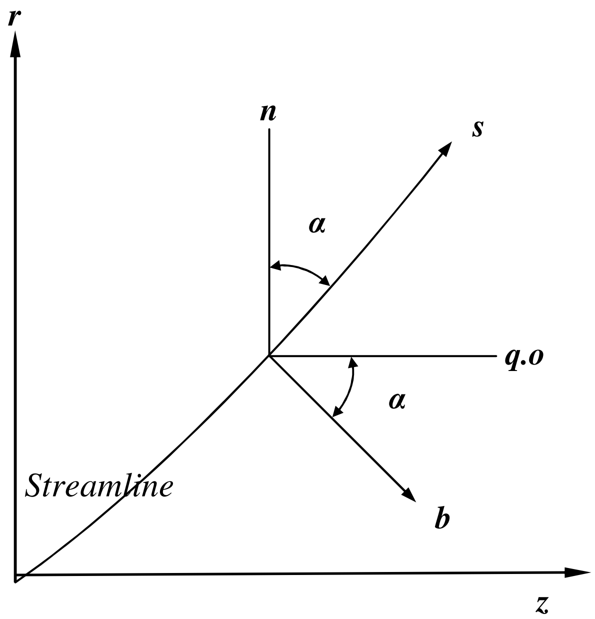
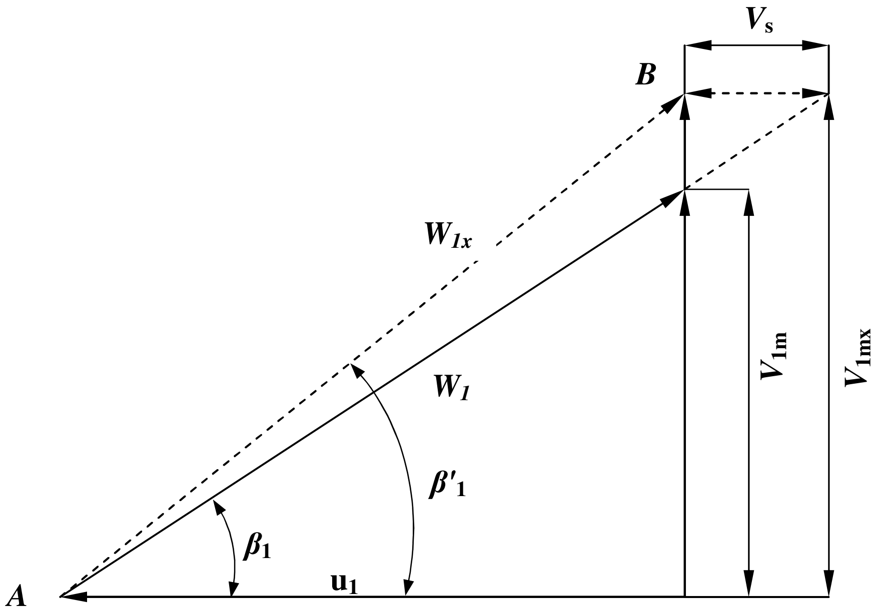



Figure (6) Streamwise and bi-normal direction




Figure( 7) Impeller entrance shock effect.

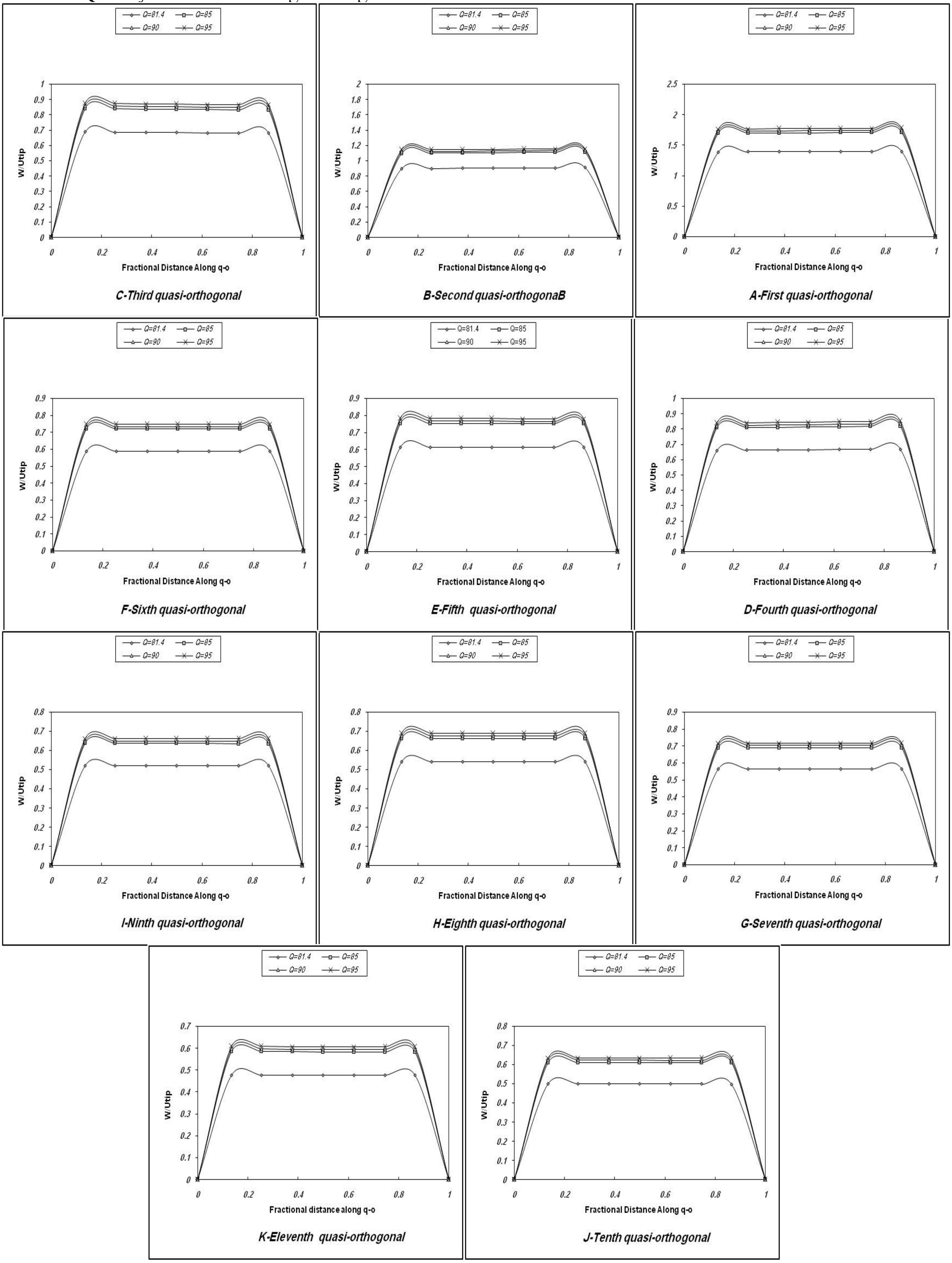
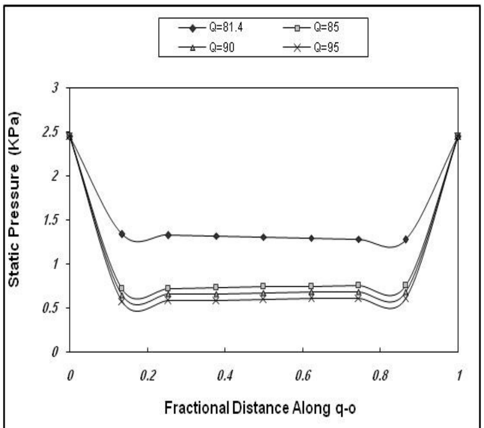
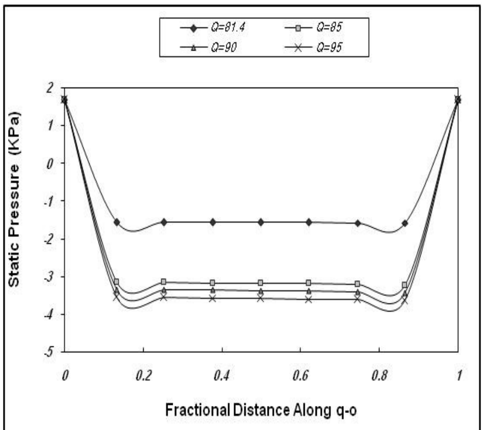


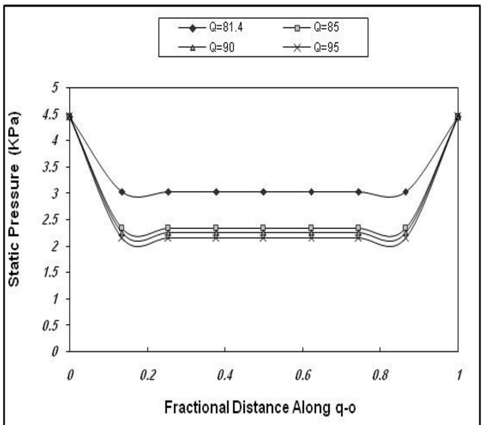

Figure (8) Effect of shock losses on velocity profile for first case.



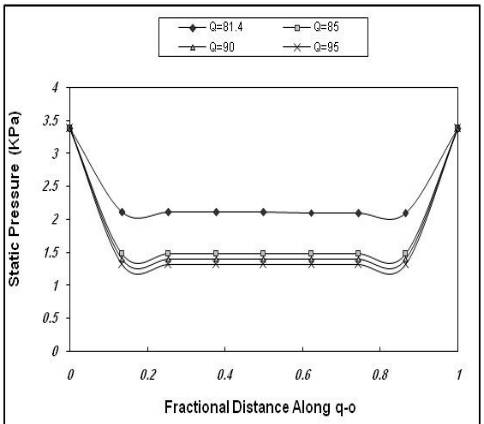

***B-Third quasi-orthogonal.***



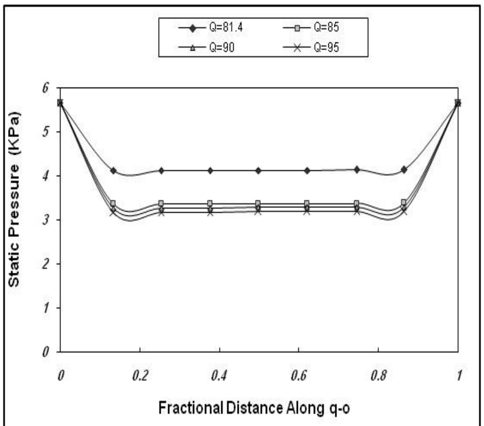

***A-First quasi-orthogonal.***



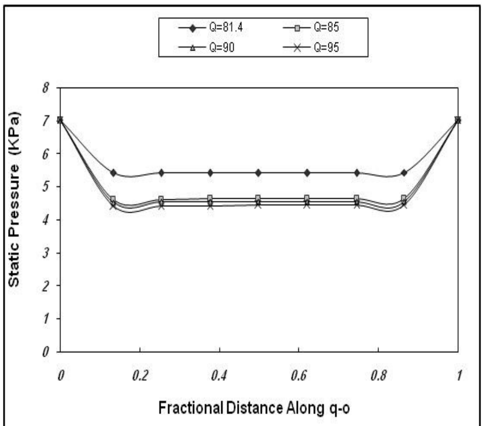

***D-Seventh quasi-orthogonal.***




***C-Fifth quasi-orthogonal.***

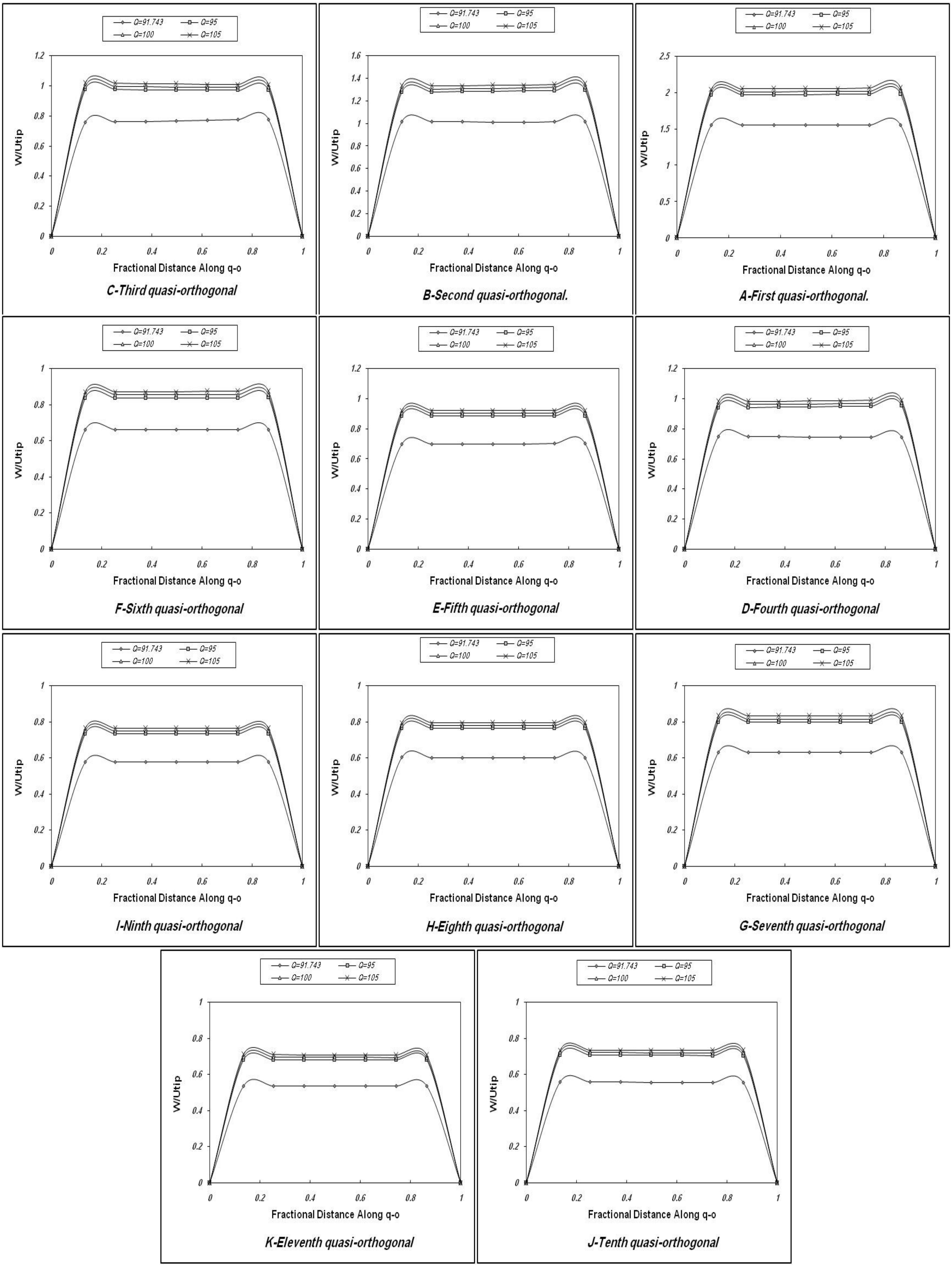



***E-Ninth quasi-orthogonal.***




***F-Eleventh quasi-orthogonal.***

Figure (9) Effect of shock losses on static pressure for first case.




Figure(10) Effect of shock losses on velocity profile for second case.

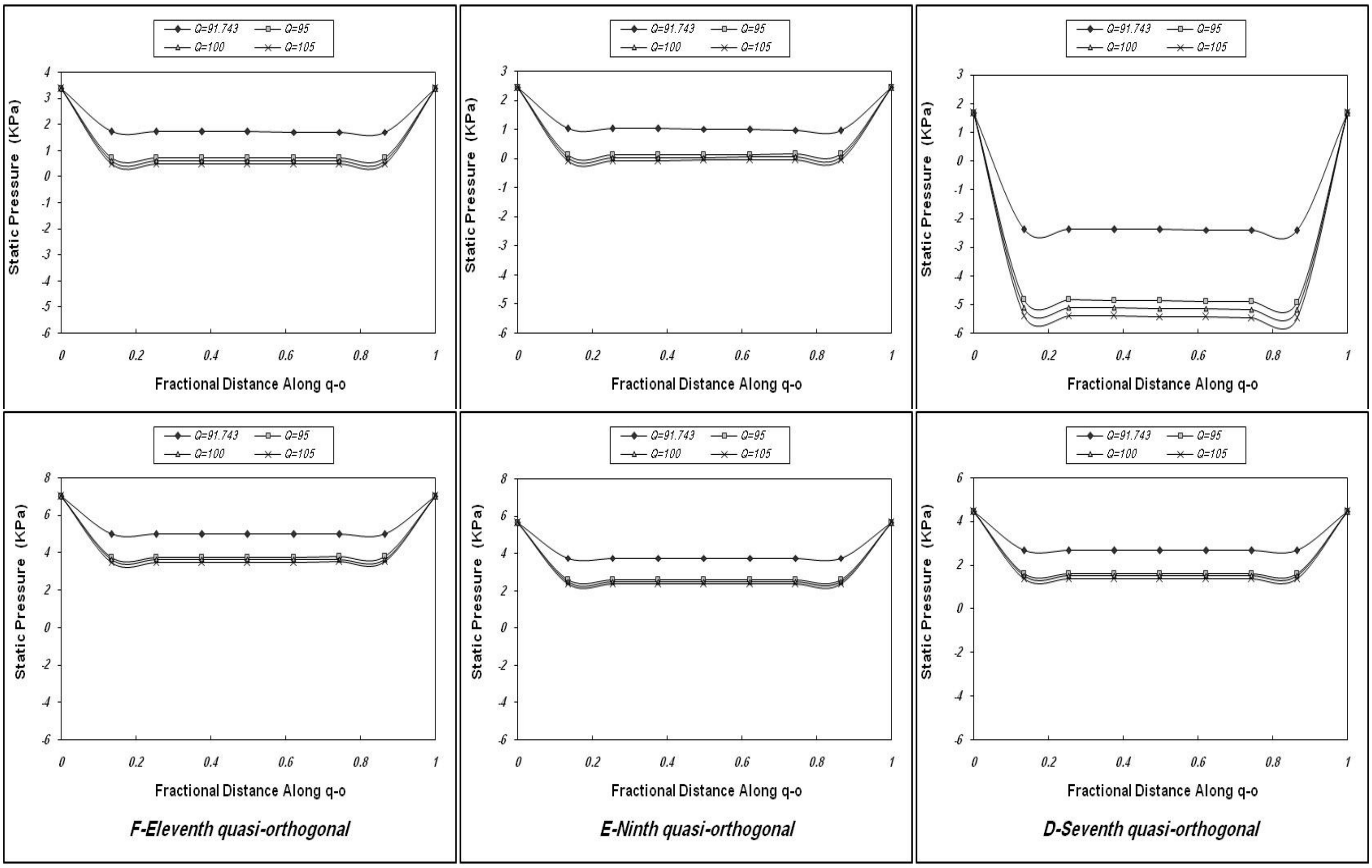



Figure (11) Effect of shock losses on static pressure for second case.