Eng. & Tech. Journal, Vol. 30 , No.20 , 2012

Theoretical Prediction of Cavitation in Radial Inflow Turbines at Design and off-Design Conditions

Dr. Jafar M. Hassan
Machines & Equipments Engineering Department, University of Technology/ Baghdad
Email: jafarmehdi1951@yahoo.com
Dr. Mohammed I. Mohsin
Machines & Equipments Engineering Department, University of Technology/ Baghdad

Received on: 9/3/2011 & Accepted on: 7/6/2012

ABSTRACT

A theoretical method for the prediction of cavitation in hydraulic radial turbines is developed in this work. The method combines a steady, quasi-three dimensional analyses which are based on the streamline curvature technique with an approach for the prediction of cavitation based on the definition of "available" and "required" cavitation coefficients. A radial inflow turbine is selected as a test case to investigate the capability of the present method as a cavitation prediction tool at design and off-design conditions. The effects of various operating conditions on the cavitation inception are examined at constant values of total head and turbine suction head. It has been found that cavitation is formed near the outlet when the turbine operated at the design speed. While at off-design speeds and flow rates, cavitation covers longer distances along suction and pressure surfaces.

Keywords: Cavitaion, Hydraulic Turbine, Through Flow Analysis, Cavitaion Coefficient.

التنبؤ بالتكهف في التوربينات القطرية نظريا عند حالات التصميم و خارج التصميم

الخلاصة

يتناول هذا البحث تطوير طريقة نظرية للتنبؤ بالتكهف في التوربينات القطرية. يتضمن الحل الدمج بين طريقة لتحليل الجريان المستقر شبه ثلاثي الأبعاد التي تعتمد على تقنية تقوس خطوط الانسياب مع أسلوب للتنبؤ بحدوث التكهف بالاعتماد على تعريفي معاملي التكهف المتاح و المطلوب. تم تطبيق الطريقة الحالية على توربين قطري مختار لغرض التحقق من فاعليتها كأداة للتنبؤ بحدوث ظاهرة التكهف عند حالات التصميم و خارج التصميم. تم اختبار تأثير حالات التشغيل المختلفة على نشوء التكهف عند ثبوت القامة الكلية و قامة السحب. وجد أن التكهف يتكون قرب المخرج عندما يعمل التوربين بالسرعة التصميمية. أما عند السرعة و التدفق خارج التصميم فان التكهف يغطي مسافات أطول على سطحي الريشة.

https://doi.org/10.30684/etj.30.20.2
2412-0758/University of Technology-Iraq, Baghdad, Iraq

Nomenclature

g	=Acceleration due to gravity	m/s^2	β	=Angle between W and meridional plane	rad
H	=Net head	m	θ	=Angular coordinate	rad
hs	=Vertical distance measured from tail water to p_{min} point inside the turbine	m	$\Delta\theta$	=Angle between blade surfaces at given point	rad
$\dot{m}$	=Mass flow rate	kg/s	ρ	=Density	kg/m^3
N	=Turbine speed	rpm	σ	=Cavitation coefficient	
n_b	=Number of full blades		ψ	=Angle between q-o and radial direction	rad
p	=Pressure	N/m^2	ω	=Angular velocity	rad/s
P_A	=Atmospheric pressure	N/m^2		**Subscripts**	
P_V	=Water vapor pressure	N/m^2	av	=Available	
Q	=Discharge	m^3/s	Cr	=Critical	
r	=Radius	m	m	=meridional	
r_e	=Radius of curvature of meridional streamline	m	min	=Minimum	
s	=Distance along quasi-orthogonal	m	n	=Component in normal to quasi-orthogonal	
U	=Peripheral velocity of flow	m/s	θ	=Tangential component	
V	=Absolute velocity of flow	m/s	q-o	= Quasi-orthogonal	
W	=Relative velocity	m/s	re	=Required	
Z	=Axial distance	m	1	=First point of flow on blade	
α	=Angle between meridional streamline and z-axis	rad	2	=Second point of flow on blade	

INTRODUCTION

Cavitation phenomenon is a problem in hydraulic machines that negatively affects their performance and causes harmful damages. It is formed inside the machine when the static pressure of liquid for one reason or another is reduced below the vapor pressure of the liquid in current temperature. When cavities are carried to higher pressure region they collapse and vapor condenses instantaneously. The necessity of prediction of cavitation gives high interest for both manufacturers and hydraulic power plants engineers.

Many investigators using both experimental and theoretical prediction techniques have studied the cavitation in hydraulic machines. Koivula[1] explained the cavitation phenomenon and its effects on the systems, various methods for cavitation detection were presented such as monitoring of steady-state flow rate, monitoring of vibrations, monitoring of acoustic noise emission, and detection of cavities by flow visualization. A cavitation monitoring of Francis turbine model have been tested by Escaler, et al. [2]. The vibration and dynamic pressure were measured in a reduced scale Francis turbine model with different types of cavitation. Several conclusions were extracted concerning the most suitable sensor for detection of each type of cavitation and a detailed analysis of the processed data permits to infer some of their particular hydrodynamic characteristics that can be extrapolated to the real case for reliable identification. Recently, acoustic models have been used by Ruchonnet, et at.[3] to study the cavitation free and cavitating flows instability at part load operation of Francis turbine. To validate the parameters of those models, they designed an experiment so that the flow characteristics are similar to draft tube surge. The influence of vapor cavity on the pressure field and Eigen frequencies of the system was examined.

In general the experimental techniques for the prediction of cavitation need expensive instruments. On the other hand, in (CFD) methods such as 3D Euler codes and 3D Navier-Stokes codes [4,5,6], a significant amount of time has still to be spent with grid generation and grid modification. However, it was felt that a much preferable way is to develop a cavitation prediction method, which is relatively simple, reliable, and efficient in terms of computing effort. In the present work such a method is developed with the assumptions of steady, quasi-three dimensional, incompressible flow. The losses arising from viscous effects are accounted for. The present method couples a flow field analysis method which is based on the assumption of a mean flow surface between vanes with an approach for the prediction of cavitation based on the definition of "available" and "required" cavitation coefficients.

A radial inflow turbine is selected, as a test case, to investigate the capability of the present method as a cavitation prediction tool at design and off-design conditions. The effects of various operating conditions (such as turbine speed, flow rate) on the cavitation inception are examined at constant value of total head.

METHOD OF ANALYSIS

The present method includes two parts. The first one is concerned with the analysis of flow field through the turbine using streamline curvature technique to calculate the distribution of velocities and pressures inside the turbine runner. These results are then supplied to the second part of analysis which includes a special technique for the prediction of cavitation inside the turbine runner. Figure (1) shows the outline of the method of analysis.

FLOW FIELD ANALYSIS

The flow field analysis is based on the assumption of a mean flow surface between vanes. In general, the mean flow surface is assumed to be parallel to the mean vane surface, with arbitrary or empirical corrections made to take care of the difference between the flow angle and the vane angle at the inlet and the outlet. The mean stream surface is projected on a plane containing the axis of rotation (meridional plane), as shown in Figure (2).

In References [7, 8] an equation for velocity gradient along an arbitrary quasi-orthogonal in the eridional plane is derived which can be written as follows:

$$\frac{dW}{ds} = \left[A\frac{dr}{ds} + B\frac{dz}{ds} \right] W + C\frac{dr}{ds} + D\frac{dz}{ds} \tag{1}$$

Where

$$A = \frac{\cos a \cos^2 b}{r_c} - \frac{\sin^2 b}{r} - \sin a \sin b \cos b \frac{\partial q}{\partial r}$$

$$B = -\frac{\sin a \cos^2 b}{r_c} - \sin a \sin b \cos b \frac{\partial q}{\partial z}$$

$$C = \sin a \cos b \frac{dW_m}{dm} + 2w \sin b + r \cos b \left(-\frac{dW_q}{dm} + 2w \sin a \right) \frac{\partial q}{\partial r}$$

$$D = \cos a \cos b \frac{dW_m}{dm} + r \cos b \left(-\frac{dW_q}{dm} + 2w \sin a \right) \frac{\partial q}{\partial z}$$

The coordinate system and the notations are shown in Figures (3 and 4).

In addition to Equation (1), which is a force equilibrium equation, the continuity equation must be satisfied, this is done by requiring that the calculated mass flow across any line from hub to shroud be equal to the specified mass flow through the turbine. Equation (1) is solved simultaneously with the continuity equation in its integral form along each quasi-orthogonal. The mass flow distribution along the quasi-orthogonal from hub to shroud can be obtained from the continuity equation as follows.

$$m^{\cdot} = n_b \int_0^s r W_n r \Delta q ds \quad (2)$$

Inverse interpolation can be used to determine the spacing of the streamlines on the quasi-orthogonal that will given equal mass flow between any two adjacent streamlines. When this is done for every quasi-orthogonal from inlet to outlet a new meridional streamline pattern is obtained, this pattern together with the calculated velocity distribution can then be used for further iteration until convergence is achieved.

The pressure distribution through the turbine is calculated by applying the Bernoulli's equation for relative velocities along the streamlines.

$$P_2 = P_1 + \frac{1}{2} r \left[(W_1^2 - W_2^2) + (U_2^2 - U_1^2) \right] \quad (3)$$

Prediction of cavitation inside the turbine

No cavitation will originate inside the turbine at all points of flow passage, if the pressure P_{min} will be higher than the critical pressure P_{cr} .

i.e. $$P_{\min} \rangle P_{cr} \quad (4)$$

According to reference [9], the critical pressure P_{cr} and the corresponding cavitation number are defined so as to operate the set above a critical cavitation index σ_{cr} . Such a cavitation number σ is linked to certain operating data (K_w, K_v, λ, and ζ_D) of a certain turbine and hence denoted by σ_{re} meaning "σ required" for this and a certain head H, flow rate $m^{\cdot}$, and rotational speed ω.

$$s_{re} = l K_w^2 + (1 - z_D) K_v^2 \quad (5)$$

Where

λ = Pressure number.

K_w= Coefficient of relative velocity.

K_v= Coefficient of absolute velocity.

ζ_D = Loss coefficient.

defined as follows:

$$l = \frac{P_1 - P_{cr}}{0.5 r W_1^2} \tag{6}$$

$$K_w = \frac{W}{\sqrt{2gH}} \tag{7}$$

$$K_v = \frac{V}{\sqrt{2gH}} \tag{8}$$

The values of P_1, W, and V are obtained from the flow filed analysis.

The cavitaion index follows also from the data of the plant surrounding as: Figure (5)

$$s = (P_A - P_{cr})/(rgH) - h_s / H \tag{9}$$

Replacing P_{cr} in the above equation by ambient related vapor pressure P_v leads to:

$$s_{av} = (P_A - P_v)/(rgH) - h_s / H \tag{10}$$

where σ_{av} is "σ available"

A cavitation safe operation of a turbine requires that

$$s_{re} \langle s_{av} \tag{11}$$

Equation (11) is checked at each point in the flow filed along the streamlines from inlet to outlet of the turbine. If it is true then there is no cavitation inception, otherwise cavitation will originate inside the turbine when σ_{re} is greater than σ_{av}.

TEST TURBINE

The method of analysis outlined was applied to a radial inflow water turbine, Figure (6). The runner has 8 unshrouded backward-curved vanes. The meridional profile and the vane layout are shown in Figure (2). The main dimensions of the runner are given in Table (1). Table (2) shows the design operating conditions of the turbine.

Table (1) Dimensions of Turbine Runner

Number of vanes	8
Inner diameter	100 mm
Outer diameter	200 mm
Vane height	15 mm
Vane thickness	3 mm
Vane angle at inlet	70 deg.
Vane angle at outlet	65 deg.

Table (2) Design Operating Conditions

Turbine speed	900 rpm
Flow rate	10 l/s
Net head	10 m

RESULTS AND DISCUSSION

The capabilities of the present method for predicting the cavitation in the test turbine at design and off-design conditions are investigated through the following cases:

Figures (7 and 8) show the variation of relative velocity and static pressure through the turbine runner at mean, suction, and pressure surfaces respectively. These results are obtained at design conditions (see Table (2)). It is seen that velocities are generally increasing, except along the pressure surface near the inlet where they decrease slightly. The pressure level is always decreasing in the direction of flow. This is, of course, normal for a radial turbine. These results are in agreement with Reference [7].

Figure (9) shows the variation of required cavitation coefficient along suction and pressure surfaces of the runner at different turbine speeds. The available cavitation coefficient calculated by using Equation (10) is (σ_{av}=1.34). According to Equation (11), cavitation is formed at the surface when the value of required cavitation coefficient becomes greater than the available cavitation coefficient (σ_{re}> σ_{av}). It is seen that at design speed (900 rpm) cavitation is formed at the suction surface near the outlet (at about 47 mm in meridional distance from the vane inlet). At off-design speeds, cavitation covers longer distance on the suction surface near the outlet. It initiates about (42.5 mm), (45 mm) in meridional distance from the vane inlet when turbine speed was (575 rpm) and (1240 rpm) respectively. On the pressure surface, Figure (9), an abnormal increase in the required cavitaion coefficient is shown near the van inlet due to high off-design turbine speed (N=1240 rpm). Generally cavitation will initiate near the vane outlet at a meridional distance greater than (40 mm) from the vane inlet.

Figure (10) shows the variation of required cavitation coefficient along suction and pressure surfaces of the runner at different flow rates. The calculated available cavitation coefficient is (σ_{av}=1.34). At design flow rate ($m^{\cdot}$=10 kg/s), cavitation is formed at the suction surface near the outlet (at about 47 mm in meridional distance from the vane inlet). At off-design flow rates, cavitation covers longer distance on the suction surface near the outlet.

On the pressure surface, Figure (10), the reduction in flow rate below the design value ($m^{\cdot}$=6 kg/s) causes an abnormal increase in the required cavitation coefficient near the inlet (from 2 to 18 mm along meridional distance). The possible explanation for this behavior, there is a region of favorable pressure gradient near the vane inlet due to off-design conditions. This could lead to the behavior of cavitation coefficient shown in Figure (10).

CONCLUSIONS

1-A method for prediction of cavitation in radial inflow turbines has been developed. It may be used for the prediction of cavitation inception at design and off-design conditions.

2- For the selected turbine, at constant total head, it has been found that cavitation is formed at the suction and pressure surfaces near the outlet when the turbine is operated at the design speed. At off-design speeds, cavitation covers longer distances along suction and pressure surfaces. An abnormal increase in the required cavitation coefficient is indicated near the vane inlet due to high turbine speed.

3- At off-design flow rates, cavitation covers longer distance on the suction surface near the outlet. The reduction in flow rate below the design value causes an abnormal increase in the cavitation coefficient near the vane inlet due to the effect of favorable pressure gradient in this region.

4- It may be concluded that the present method may be used effectively in the preliminary design stages as a tool for the prediction of cavitation in hydraulic turbines.

REFERENCES

[1]. Koivula, T. "On cavitation in fluid power", Proc. Of 1st FPNI-PhD Symp. Hamburg, pp.371-382, 2000.

[2]. Escaler, X. ; Farhat, M. ; Ausoni, P. ; Egusquiza, E. and Avellan, F. "Cavitation monitoring of hydro turbines: tests in a Francis turbine model", 6th Int. Symp. On Cavitation, CAV 2006, Wageningen, The Netherlands, sep. 2006.

[3]. Ruchonnet, N. ; Nicolet, C. ; Alligne, S. ; Avellan, F. and Rudolf, P. "Acoustic resonance in cavitation free and cavitating flows", Proceedings of the 3rd IAHR Int. Meeting of the Work group on Cavitation and Dynamic Problems in Hydraulic Machinery and Systems, Brno, Vol.2, pp.501-514, 2009.

[4]. Keck, H. ; Drtina, P. and Sick, M. "Numerical hill chart prediction by means of CFD stage simulation for a complete Francis turbine", XVIII IAHR Symp. Hydraulic Machinery and Cavitation, Valencia, Spain, Sep. 1996.

[5]. Drtina, P. and Sallaberger, M. "Hydraulic turbine- basic principles and state-of-the art computational fluid dynamics applications', IMechE, Vol.213, Part C, pp.85-102, 1999.

[6]. Jalil, J.M.; Hassan, J.M. and Saleh, H.M. "Numerical and experimental study of three-dimensional water turbine of radial flow type" ARABIAN Journal for Science and Engineering, Saudi Arabia, 2010.

[7]. Al-Zuhairy, H. A. D. "Cavitation in radial turbines" M.Sc. Thesis, Al-Rasheed College of Engineering and Science, University of Technology, 2002.

[8]. Rahmatalla, M.A.F. and Idris, M. "Numerical calculation of flow through a centrifugal compressor using streamline curvature technique" Modelling, Simulation & Control, AMSE Press, Vol.39, No.4, pp.43-54, 1991.

[9]. Raabe, J. Hydro Power, VDI-Verlag GmbH, 1985.

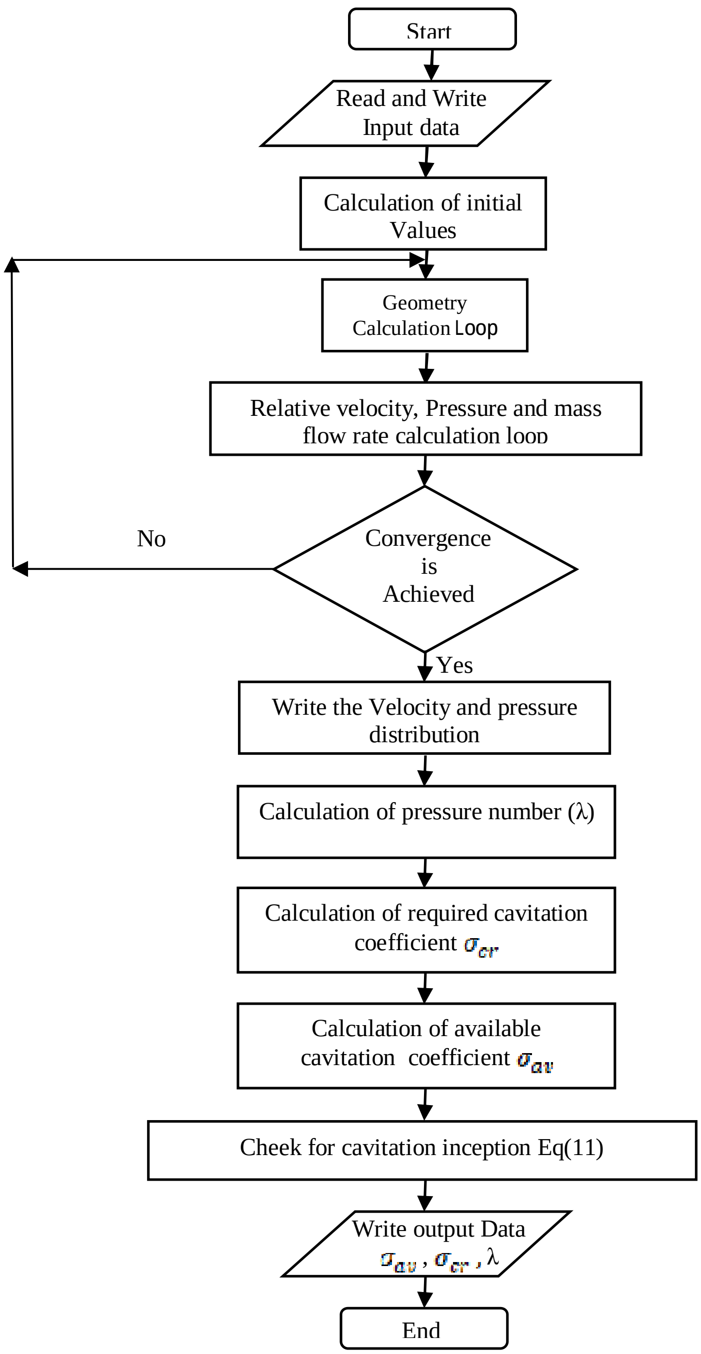


Figure (1) Outline of the present method.

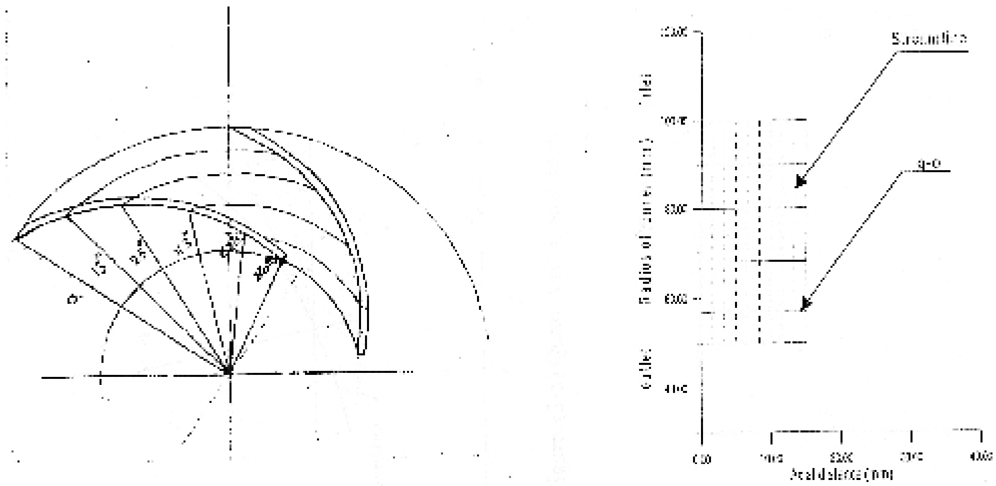


Figure (2) Vane layout and meridional plane of the test turbine.

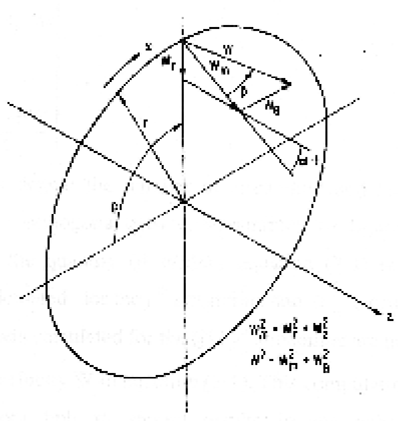


Figure (3) Coordinate system and relative velocity components.

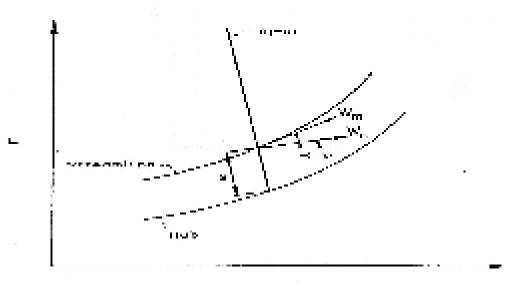


Figure (4) Components of relative Velocity W_n normal to the (q-o).

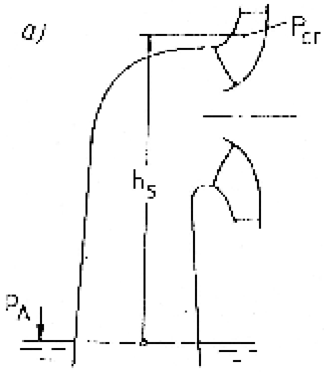


Figure (5) Cavitation coefficient σ_{av} (eq. 10).

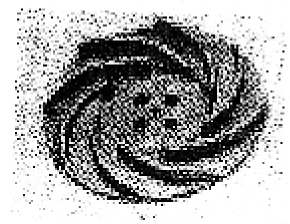


Figure (6) Test turbine runner.

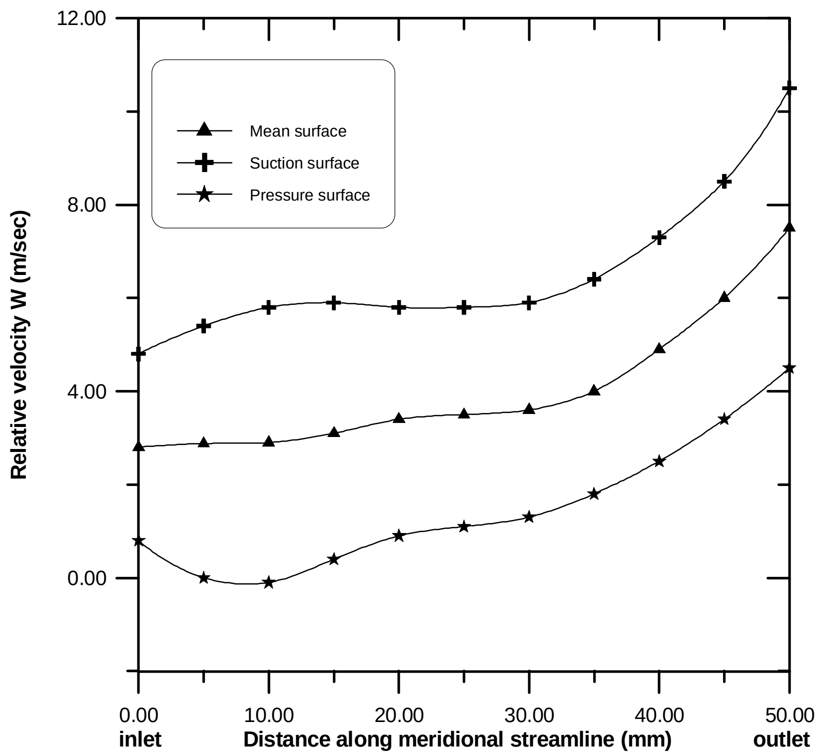


Figure (7) Variation of relative velocity through the turbine runner at design conditions (N=900 rpm, Q=10 l/s, H=10 m).

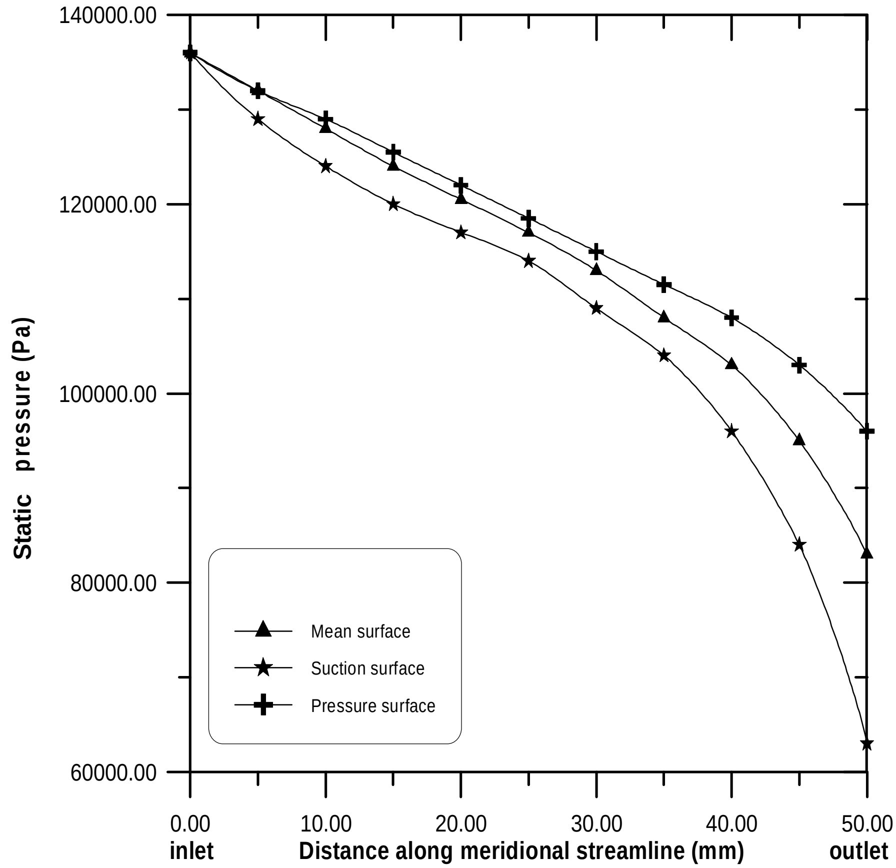
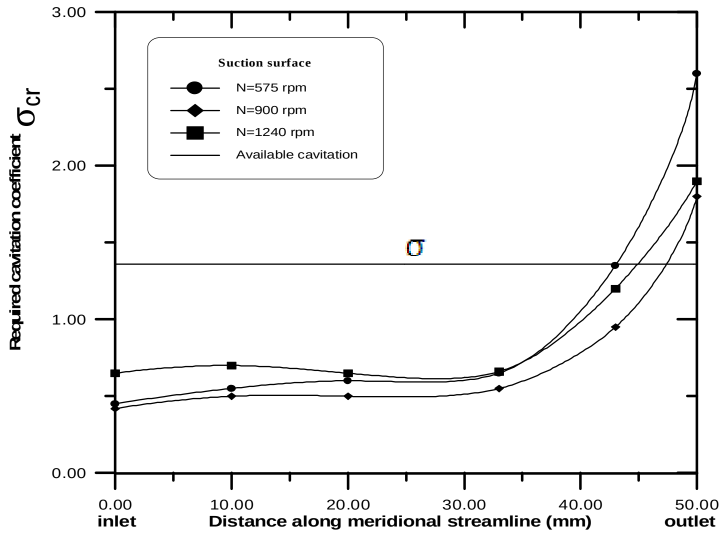


Figure (8) Variation of pressure through the turbine runner at design conditions (N=900 rpm, Q=10 l/s, H=10 m).



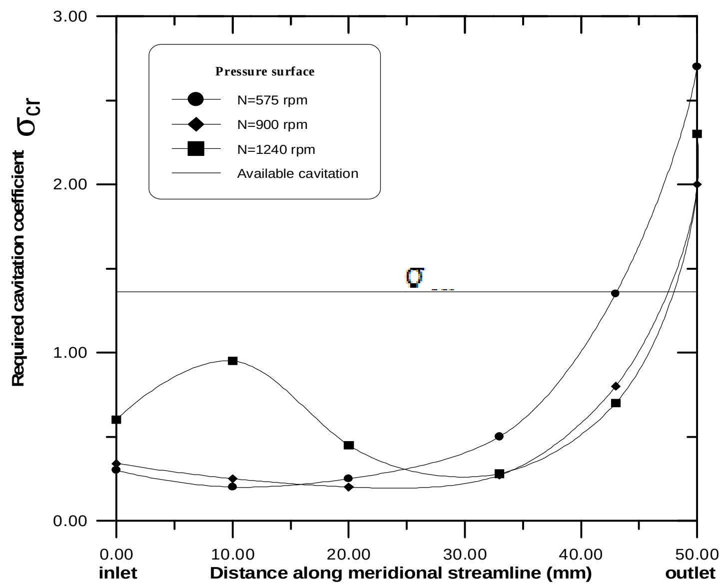


Figure (9) Variation of required cavitation coefficient at different turbine speeds (Q=10 l/s, H=10 m).

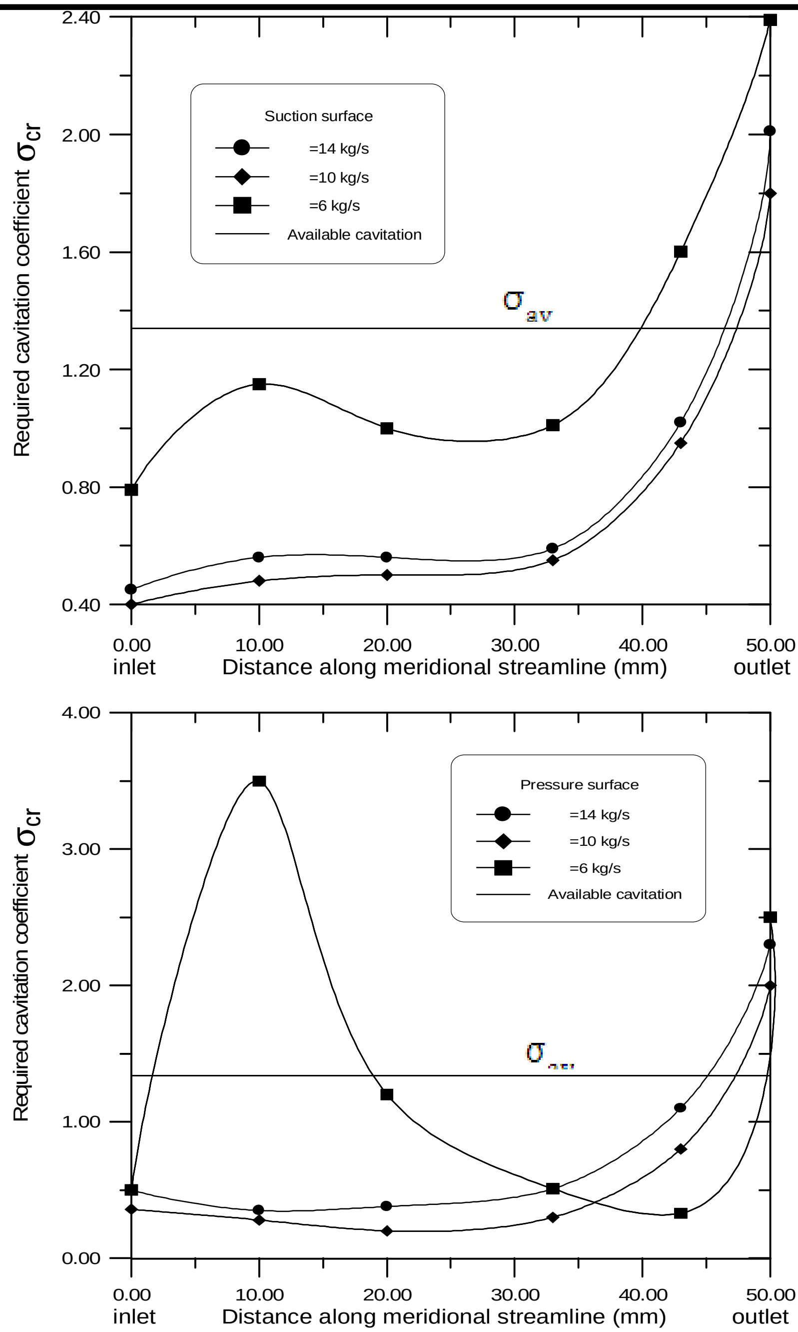


Figure (10) Variation of required cavitation coefficient at Different flow rates (N=900 rpm, H=10 m).