

THE PERFORMANCE CHARACTERISTICS EVALUATION OF AN IMPELLER OF CENTRIFUGAL PUMP WITH A VOLUTE USING COMPUTATIONAL FLUID DYNAMICS TECHNIQUE⁺

تقويم مميزات أداء دافع لمضخة طاردة مركزية مع الحلزون باستخدام طريقة ديناميكا الموائع الحسابي

Mustafa Tariq Mustafa *

Ahmed M. Ibrahim **

Abstract :

The objectives of this research is to predict the values of the (head, efficiency, pressure and velocity) of the three dimensional flow inside impeller and volute by using computational fluid dynamics technique CFD commercial code (CFX-TASCflow V. 2.12). The governing equations are continuity and three dimensional Navier-Stokes equations. These equations are discretized by finite volume method and solved by SIMPLE logarithm.

The observed data for pump are collected from Al-Karama Company. The predicted results of the performance characteristics curves are validated with the observed results. The validations are properly predicted at design and below the design flow rate, but above the design flow rate; the prediction starts to deviate and over at off-design conditions. The recirculation zone appears at the diffuser near the tongue at off-design conditions. The mean square error (MSE) is used to evaluate the deviation between predicted and observed results. The maximum deviation between predicted and observed results is equal to 5%. The results demonstrate the capability of the software package to calculate the performance characteristics for centrifugal pump with a volute.

المستخلص:

يهدف البحث الى تخمين قيم (الارتفاع، الكفاءة، الضغط و السرعة) للجريان ثلاثي الابعاد داخل دافع ذو جريان شعاعي مع غطاء حلزوني باستخدام برنامج ديناميك الموائع الحسابي CFD التجاري (CFX-TASCflow 2.12). تمت الدراسة نظريا باستخدام معادلة الاستمرارية و معادلة نافير- ستوك بالابعاد الثلاثية (r, θ, z) . حلت هذه المعادلات بصيغة الحجم المحدد (finite volume) و simple a logarithm. قورنت النتائج النظرية مع البيانات العملية لمضخة تم تجميعها في شركة الكرامة و اظهرت توافق مع النتائج العملية حيث ان منحنيات الاداء اظهرت توافق عند معدل الجريان التصميمي $(Q=100 \text{ m}^3/\text{hr})$ و اظهرت انحراف عند معدل جريان اعلى من القيمة التصميمية. عند ظروف بعيدة عن القيم التصميمية ظهرت منطقة الانفصال عند الناشرة. اعتمدت طريقة معدل الخطأ التربيعي لحساب الانحراف بين النتائج النظرية و العملية و كان أقصى انحراف

⁺Received on 23/4/2008 ,Accepted on 30/4/2009 .

*Assistant professor / Technical College/Baghdad

**Assistant teacher / Technical College/Baghdad.

(5%) . كما اظهرت النتائج كفاءة البرنامج المستخدم لحساب مميزات اداء مضخة طاردة مركزية مع غطاء حلزوني.

INTRODUCTION

Computational Fluid Dynamics (CFD) codes are used to simulate the flow field and to predict performance characteristics of centrifugal pumps and volutes. Many studies have been done to investigate the flow in all types of centrifugal pumps. [1] developed a three-dimensional time-marching incompressible Navier-Stokes solver with the pseudo-compressibility technique to study the flow field through mixed-flow water pump impeller. [2] study the influence of the volute geometrical parameters (like shape of volute cross section, circumferential variation of the cross sectional area, radial position of the volute cross section) on the global volute performance. The results show that internal type of volute which has a cross section center radius smaller than the diffuser exit radius has a very high loss coefficient compared to the other types. Their global performance measurements indicate that the volute with the circular cross sectional shape correspond to minimum losses compared to the rectangular ones. [3] study flow inside the compressor volute with elliptical cross section by solving the three dimensional compressible Euler equations modified with the loss model. The results show that the developed model accurately predicts the velocity and pressure distribution inside the volute. A similar model is also used by [4] for flow calculations inside the external volute of rectangular cross sections.

The objective of the present research are to calculate the head, efficiency, pressure and velocity characteristics of the three dimensional flow inside the centrifugal pump with volute. These calculations are predicted by using CFD commercial code CFX-TASCflow V. 2.12. Also, these predicted results are compared with the observed characteristics extracted from [5] for verifications and show agreements.

NUMERICAL SIMULATION:

The performance characteristics of impeller and volute have been numerically modeled by using the commercial CFD code, CFX-TASCflow V. 2.12. The observation results for impeller and volute are available at the Al-Karama Company. The CFD code solves numerically the Reynolds-averaged Navier-Stokes equations employing an element-based finite volume method. The numerical algorithm used is SIMPLE, in which the pressure parameter is discretized utilizing the second-order upwind scheme. The numerical models are verified using the experimental data [5].

1-Governing equations

For three-dimensional incompressible flow, the continuity and Navier-Stokes equations can be written as follow as reported by [6]

The continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{1}{r} \frac{\partial (\rho r u_r)}{\partial r} + \frac{1}{r} \frac{\partial (\rho u_\theta)}{\partial \theta} + \frac{\partial (\rho u_z)}{\partial z} = 0 \quad (1)$$

The Navier-Stokes equation:

In r- direction

$$\rho \left(\frac{\partial u_r}{\partial t} + u_r \frac{\partial u_r}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_r}{\partial \theta} - \frac{u_\theta^2}{r} + u_z \frac{\partial u_r}{\partial z} \right) = -\frac{\partial p}{\partial r} + \rho g_r + \mu \left[\frac{\partial}{\partial r} \left(\frac{1}{r} \frac{\partial}{\partial r} (r u_r) \right) \right]$$

$$+ \mu \left(\frac{1}{r^2} \frac{\partial^2 u_r}{\partial \theta^2} - \frac{2}{r^2} \frac{\partial u_\theta}{\partial \theta} + \frac{\partial^2 u_r}{\partial z^2} \right) \quad (2)$$

In θ -direction

$$\rho \left(\frac{\partial v_\theta}{\partial t} + u_r \frac{\partial u_\theta}{\partial r} + \frac{u_\theta}{r} \frac{\partial v_\theta}{\partial \theta} - \frac{u_r u_\theta}{r} + u_z \frac{\partial u_\theta}{\partial z} \right) = -\frac{1}{r} \frac{\partial p}{\partial \theta} + \rho g_\theta + \mu \left[\frac{\partial}{\partial r} \left(\frac{1}{r} \frac{\partial}{\partial r} (r u_\theta) \right) \right]$$

$$+ \mu \left(\frac{1}{r^2} \frac{\partial^2 u_\theta}{\partial \theta^2} - \frac{2}{r^2} \frac{\partial u_r}{\partial \theta} + \frac{\partial^2 u_\theta}{\partial z^2} \right) \quad (3)$$

In z-direction

$$\rho \left(\frac{\partial u_z}{\partial t} + u_r \frac{\partial u_z}{\partial r} + \frac{u_\theta}{r} \frac{\partial u_z}{\partial \theta} + u_z \frac{\partial u_z}{\partial z} \right) = -\frac{\partial p}{\partial z} + \rho g_z + \mu \left[\left(\frac{1}{r} \frac{\partial}{\partial r} \left(r \frac{\partial u_z}{\partial r} \right) \right) \right]$$

$$+ \mu \left(\frac{1}{r^2} \frac{\partial^2 u_z}{\partial \theta^2} + \frac{\partial^2 u_z}{\partial z^2} \right) \quad (4)$$

The above equations with boundary equations can be simulated by this CFD model.

2- Geometrical model and grid generation

The centrifugal pump (radial flow) has six blades. Table 1 shows a specification of the pump in detail [5].

Table (1): The specifications of the radial flow pump [5]

Flow rate (Q)	100 m ³ /hr
Head (H)	85.6 m
Rotational speed (N)	2000 rpm
Impeller inlet diameter (D _{in})	100 mm
Impeller outlet diameter (D _{out})	360 mm

The geometry of impeller and volute model extracted from descriptions in Al- Karama Company. The observation results for performance characteristics head and efficiency are as follow:

Table (2): The observation results for performance characteristics [5]

Flow rate (Q) m ³ /hr	Head (H) m	Efficiency (η)
40	93	34.9
60	92.3	52.5
80	88.5	66.4
100	85.6	80.3
120	70	78.8

The computational impeller and volute grids as shown in Figure (1) were generated using the CFX-Turbogrid mesh generator program and CFX-VoluteGen program respectively. The simulation was performed with the impeller fixed in one position (frozen rotor). The grid was approximately 250,000 hexahedral elements [6].

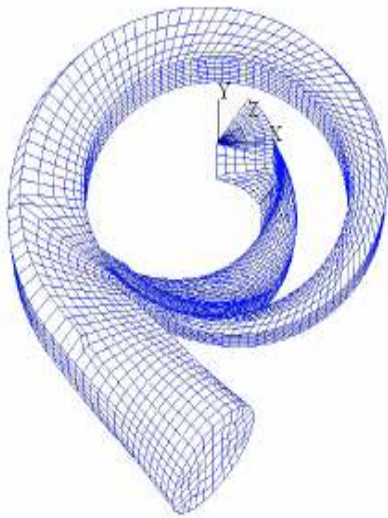


Figure (1): Computational meshes for geometrical model

3- Boundary conditions and assumptions

The pump blade and the hub wall, extending to the exit, are rotating walls, with zero relative velocity. The shroud, from the inlet to the exit, is a stationary wall, with zero absolute velocity. The inlet flow is modeled as uniform flow, at the exit boundary; the mass flow rate is specified. The simulation of the model can be revealed by the flowchart as shown in Figure (2).

Start

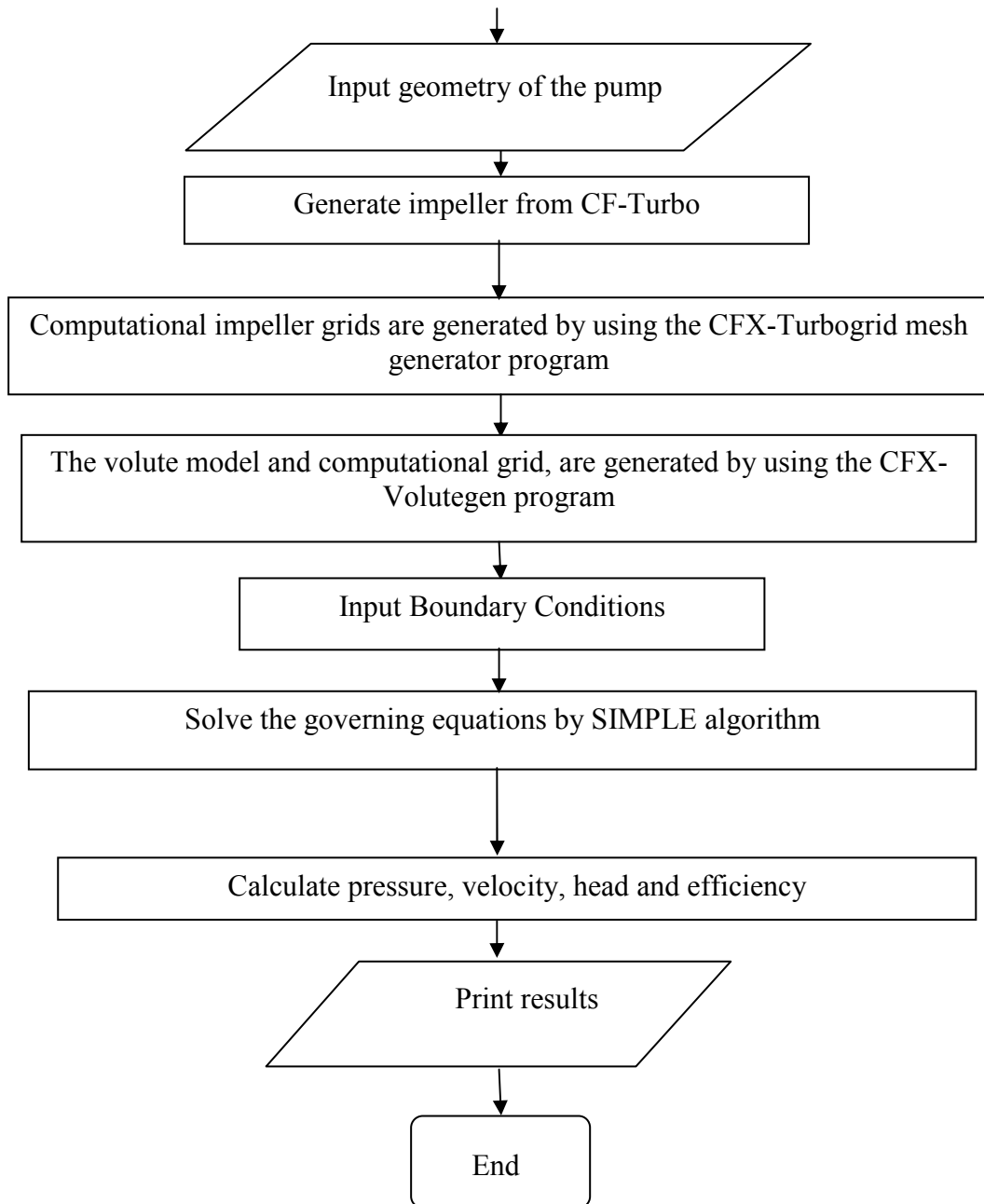


Figure (2): Flowchart for numerical simulation

RESULTS AND DISCUSSIONS

The main focus from this investigation is to demonstrate the capability of CFD commercial code CFX-TASCflow to predict the performance characteristics of impeller centrifugal pump with a volute at different operating points.

1- Flow inside impeller and volute

Figures 3, 4 and 5 show the predicted velocity contours for impeller and volute at partial load $80 \text{ m}^3/\text{hr}$, design flow and over load $120 \text{ m}^3/\text{hr}$ respectively. The recirculation zone appears in the diffuser near the tongue at off-design conditions.

Figures 6, 7 and 8 show the predicted pressure contour for impeller and volute at partial load $80 \text{ m}^3/\text{hr}$, design flow and over load $120 \text{ m}^3/\text{hr}$ respectively. These figures show the variation of pressure zone from impeller eye to the volute which is roughly constant in tangential direction but it decreases in radial direction.

2- Performance curve

Figures (9 and 10) give the typical performance characteristics curves as computed with CFX-TASCflow software. The head and efficiency characteristics can be determined and plotted in these figures. These figures include the actual head and efficiency measurements.

These figures show the predicted and measured head and efficiency curve versus flow rate at constant rotational speed. From these figures, can be seen that the head rise and efficiency curve is properly predicted near design flow for the centrifugal pump (impeller and volute). Below design flow the prediction is also in agreement with the measurements, but above design flow the CFX-TASCflow prediction starts to deviate and over predicts the head and efficiency increase. The mean square error is used to evaluate the deviation between predicted and observed results. The maximum deviation between predicted and observed results is equal to 5% for these figures.

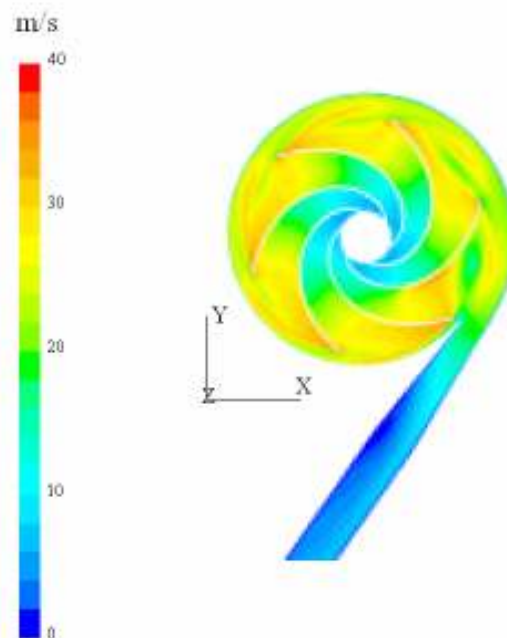


Figure (3): Velocity contour for impeller and volute at $80 \text{ m}^3/\text{hr}$

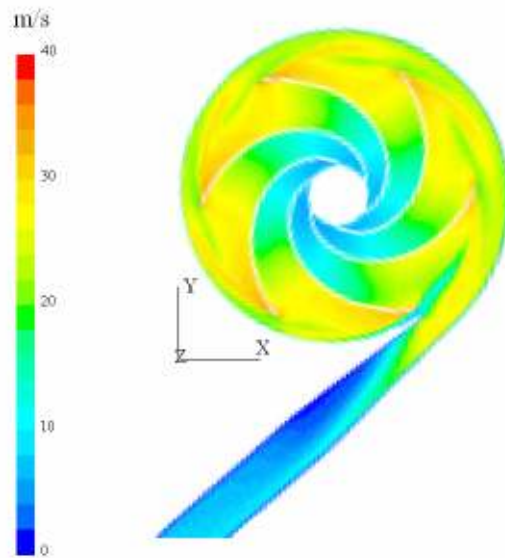


Figure (4): Velocity contour for impeller and volute at 100 m³/hr

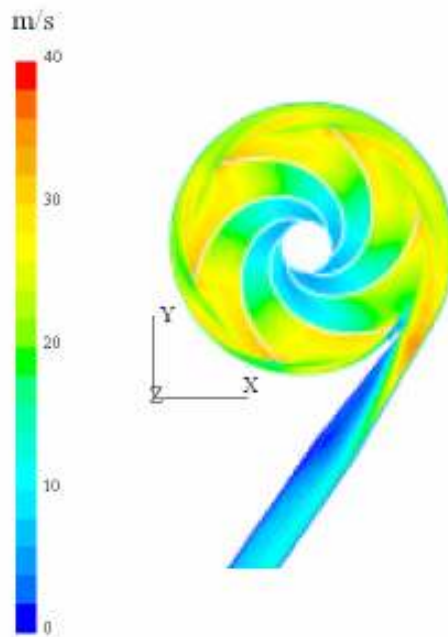


Figure (5): Velocity contour for impeller and volute at 120 m³/hr

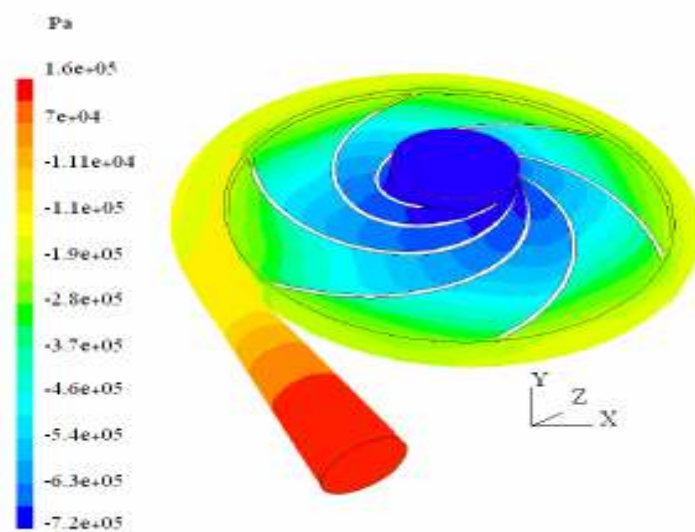


Figure (6): Static pressure contour for impeller and volute at 80 m³/hr

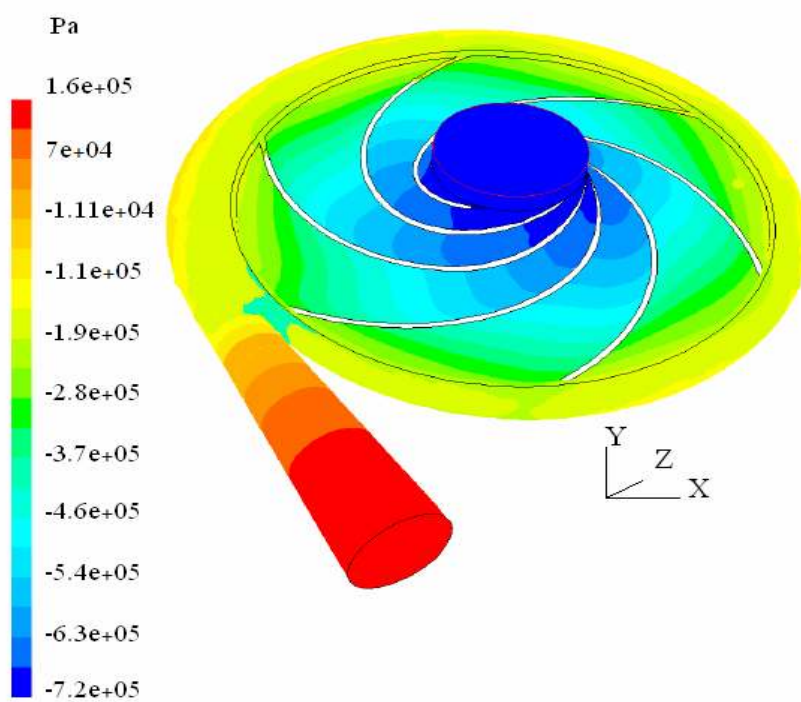


Figure (7): Static pressure contour for impeller and volute at 100 m³/hr

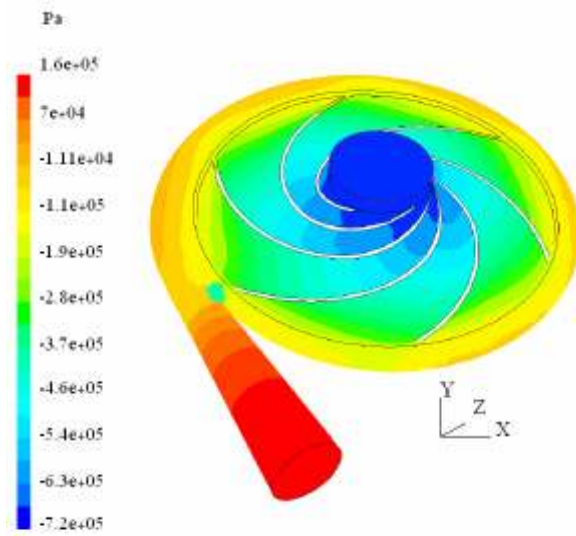


Figure (8): Static pressure contour for impeller and volute at 120 m³/hr

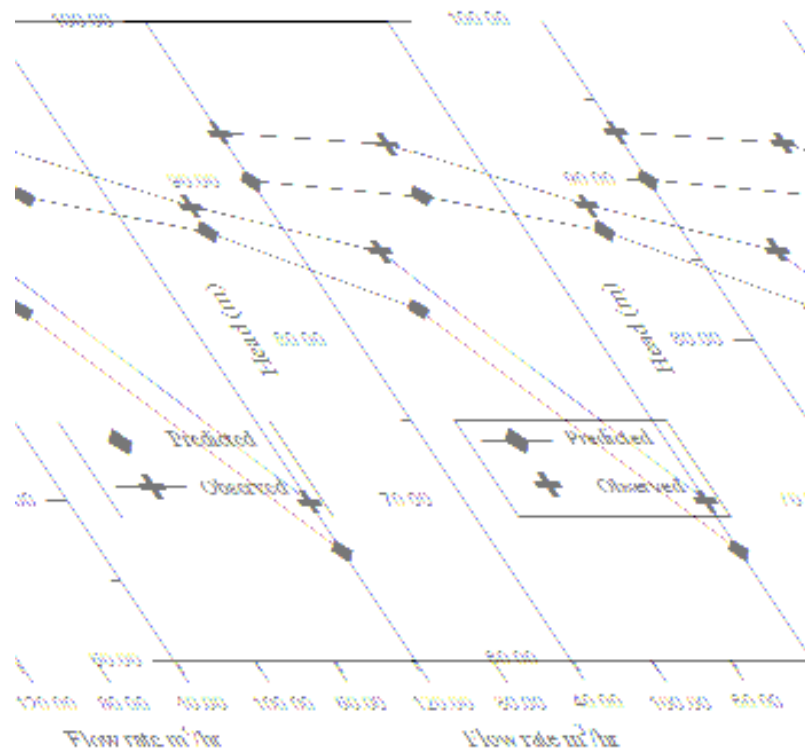


Figure (9): Variations of the predicted and observed head vs. flow rate

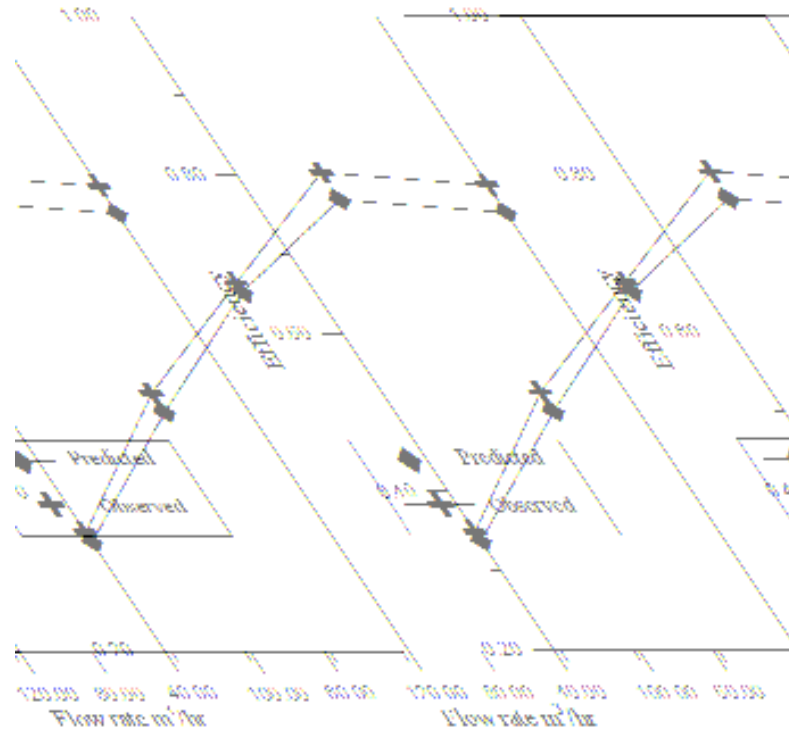


Figure (10): Variations of the predicted and observed efficiency vs. flow rate

CONCLUSIONS

A numerical model of 3D flows, based on the commercial CFD code CFX-TASCflow V. 2.12, has been used to predict head and efficiency characteristic curves as well as pressure and velocity contours for flow inside the impeller and a volute.

The predicted results of the head and efficiency characteristic versus flow rate curves are validated by the observed results at constant rotational speed. The characteristics curves are properly predicted at design and below the design flow rate, but above the design flow rate; the CFX-TASCflow prediction starts to deviate and over predicts the head and efficiency increase. The recirculation zone appears in the diffuser near the tongue at off-design conditions. The maximum deviation between predicted and observed results is equal to 5%. The predicted results give a confidence in CFD code CFX-TASCflow to predict the performance characteristics for impeller and a volute.

NOMENLATURE

g	gravitational constant	m/s^2
H	head	m
N	rotational speed	min^{-1}
n_s	specific speed	-
P	pressure	N/m^2
Q	Flow rate	m^3/hr
r	radial coordinate	m
t	time	s
u_r, u_θ, u_z	components velocity vector	m/s
z	axial coordinate	m
θ	angular coordinate	rad
μ	molecular viscosity	$Pa \cdot s$
ρ	density	kg/m^3

REFERENCES:

- 1- Zhou W. D. and Ng E. Y. K., "3-D Viscous Flow Simulation of Mixed-flow Water Pump Impeller with Tip-clearance Effects", *Proceedings of the Fourth International Conference and Exhibition on Pump and Systems*, USA, 1998.
- 2- Mishina H., Gyobu I., "Performance Investigations of Large Capacity Centrifugal Compressors", *ASME Paper 78-GT-3*, 1978.
- 3- Ayder E., and Van den Braembussche R.A., "Numerical Analysis of the Three-Dimensional Swirling Flow in Centrifugal Compressor Volute", *ASME Journal of Turbomachinery*, Vol. 116, pp 462-468, 1994.
- 4- Hagelstein D., Hillewaert K., Van den Braembussche R. A., Engeda A., Keiper R., and Rautenberg M, "Experimental and Numerical Investigation of the Flow in a Centrifugal Compressor Volute", *ASME Journal of Turbomachinery*, Vol. 122, 2000.
- 5- Al-Karama Company, "Radial Flow Pump with Volute", Feasibility report, Baghdad, Iraq, 2002.
- 6- CFX-TASCflow V2.12 Primer Documentation, Waterloo, Canada.