CFD parametric simulation of low specific speed centrifugal pump

Mahmoud Fouaad[1], Mohamed Adel[2] and Ahmed Ashmawy[3]

[1] professor, Mechanical Power Engineering department, faculty of engineering, Cairo University.
[2] professor, Mechanical &Electrical Research Institute, National Water Research Center.
[3] postgraduate student, Mechanical Power Engineering department, faculty of engineering, Cairo university.
Eng_ahmed_ashmawy86@yahoo.com

Abstract: In the present work CFD simulation is used to investigate the effects of adding splitter blades and change in volute cross section shape on the performance of low specific speed centrifugal pump. Low specific speed centrifugal pumps are high demand in numerous applications, this type of pumps, however they are very sensitive to changes as they are characterized by relatively low efficiency. A three dimensional simulation was used to predict the flow pattern inside centrifugal pump. Standard k-ε turbulence model with enhanced wall function and **PISO** (Pressure-Implicit with Splitting of Operators) were chosen for turbulence model, wall treatment and pressure-velocity coupling respectively. This simulation has been carried out for model that has the following properties (head (H) =77.6 m, discharge (Q) =225 m^3/h, specific speed (N_s) =14.2, inlet blade angle (β_1) = 10.2°, outlet blade angle (β_2) = 15.1°). For pump model simulation was done at five different operating points (0.1Q, 0.4Q, 0.7Q, Q, 1.2Q). The difference between the head from CFD simulation less than the head from experimental with average value of (2.7%). Adding splitter blades to the impeller lead to increase the head of the pump with (14.4%) than the head from experimental. Changing volute case cross section shape to rectangular leaded to reduce the head by (2.5%) compared with the head from experimental, reduce the efficiency by (3.45%). Changing volute cross section shape to trapezoid leaded to reduce the head by (3%), reduce the efficiency by (4.8%).

[Mahmoud Fouaad, Mohamed Adel and Ahmed Ashmawy. **CFD parametric simulation of low specific speed centrifugal pump.** *J Am Sci* 2014;10(12):315-323]. (ISSN: 1545-1003). http://www.jofamericanscience.org. 41

Keywords: CFD, Impeller, centrifugal pump, volute, spilleter blades and pump performance.

1. Introduction:

Centrifugal pumps are used in a wide range of industrial and domestic applications. They have many advantages as; they have variable capacity control over operating range at constant speed; can handle liquids containing catalyst, dirt solids; can pump liquids with poor lubricity and does not normally require overpressure protection over operating range. The design of centrifugal pumps still mainly depends on experiences especially model test. The impeller and volute geometry is chosen according to several optimization criteria. Various parameters affect the pump performance and energy consumption. The blade angle, adding splitter blades and volute cross section shape have influence on pump efficiency. Using Computational fluid dynamics to predict pump Performance and effect of change in outlet blade angle, adding splitter blades and volute cross section shape in pump performance.

Dick E., ***et al.*****[2001]** study the pump performance using Computational fluid dynamics to show the best methods of fluent methods(the multiple reference frame method, the mixing Plane method and the sliding mesh method), the sliding mesh technique was able to predict performance of the pump with confidence. **Thin K.C.,** ***et al.*** **[2008]** study the full design of centrifugal pump and analyzed to get the best performance point, they calculate the different losses of centrifugal pump theoretically (shock losses, impeller friction losses, volute friction losses, disk friction losses and recirculation losses*)* at various operating conditions and its effect on pump performance. **Bacharoudis E.C.,** ***et al.*** **[2008]** study the performance of impellers with the same outlet diameter having different outlet blade angles, when pump operates at nominal capacity, the gain in the head is more than 6% when the outlet blade angle increases from(20 deg to 50 deg), however the above increment of the head is recompensed with 4,5% decrease of the hydraulic efficiency, the numerical simulations predict reasonably the total performance and the global characteristics of the laboratory pump. **Kaewnai S,** ***et al.*** **[2009]** predict the performance of a radial impeller of centrifugal pump using CFD, the analysis was done based on grid quality and various turbulence model in computational fluid dynamics program (k - ε, k - ω and RNG k - ε), the analysis show that head values computed from all three turbulence models are similar, with only 0.3% difference. **Mustafa T. M.,** ***et al.*** **[2009]** predicted the head, efficiency, pressure and velocity characteristics of the three dimensional flow inside the centrifugal pump with volute using computational fluid dynamics, the simulation was performed with the impeller fixed in one position, the predicted and measured head and efficiency curve versus flow rate at constant rotational speed was in agreement with the measurements. **I.Shahin, *et al.* [2010]** study pressure fluctuations on the blade and stator surfaces. Three different pump constructions are performed to study the rotor-stator interactions; pump with volute casing, pump with van less diffuser and pump with vaned diffuser. **Shah S. R., *et al.* [2010]** study flow and Numerical simulation for centrifugal pump with different turbulence models (k - ε) and (RNG k - ε) at six operating point from (30% to 110%), it was found that k-ε turbulence model provides better results compared to RNG k- ε model. The head predicted by CFD analysis was 5 to 7 % lower than the mode test results, **A. Manivannan [2010]** study the flow pattern inside mixed flow pump and study impeller vane angles effect on pump efficiency, by changing the outlet angle the efficiency of the impeller was improved, for the second case the efficiency of the impeller improved but not like increase in efficiency in change in outlet angle. **Aman A., *et al.* [2011]** 2-D simulation of turbulent fluid flow used to visualize the flow in a centrifugal pump, including the pressure and velocity distributions, was observed that FLUENT simulation results give good prediction of performance of centrifugal pump and may help to reduce the required experimental work for the study of centrifugal pump performance.

Mustafa Gölcü, *et al.* [2010] study the effect of splitter blades on a deep well pump for three impellers have different numbers of blade number (z = 5, 6, 7) and low blade discharge angle and splitter blade of (25, 35, 50, 60 and 80% of the main blade length), when adding splitter blade adding splitter blades to the deep well pump impellers of (z = 6 and 7) under investigation decreased the head and as splitter blade length increased, the overall efficiency of the pump also decreased splitter blades were not seen to have positive effects on the pump performance characteristics, when the blade number was five, the efficiency increased with increasing splitter blade length. **G. Kergourla, *et al.* [2007]** study the influence of adding splitter blades on the performance of a centrifugal pump, adding splitters has a positive and negative effect on the pump performance, the splittered impeller head is approximately (10–15%) higher than the original impeller but the efficiency is not improved since the hydrodynamic losses are greater. **S. Chakraborty, *et al.* [2011]** study centrifugal pump performance of impellers with the same outlet diameter having different blade numbers with (4,5,6,7,8,9,10,12 numbers of blades), two-dimensional study of steady static pressure distribution, total pressure distribution, the changes in head and efficiencies, simulation was steady and moving reference frame used to take into account the impeller-volute interaction, with the increase of blade number, the head and static pressure of the model increases **Sunsheng Yang, *et al.* [2011]** study volute main geometric parameters including volute throat area, volute cross-section shape(round, horseshoe, rectangular and trapezoid), design rule of spiral development area and radial gap between impeller and volute tongue influence of volute main geometric parameters to pump performance, From the simulation the round-shaped volute cross section generates the highest pressure head, the second highest pressure head is pump with horseshoe shaped cross-section, the pressure head of rectangular and trapezoid-shaped cross-section pump is almost the same.

2. Pump Model:

Pump model used in the present research has the following properties:

Speed (N)	1490 rpm	Outer diameter of Impeller(D_2)	489 mm
Head (H)	77.6 m	Inlet diameter of Impeller(D_1)	202.72 mm(8")
Discharge (Q)	225 m^3/h	Inlet blade angle(β_1)	10.2°
specific speed(N_s)	14.2	Outlet blade angle(β_2)	15.1°
Number of blades (Z)	6		

Changes were made to pump design

- Adding six splitter blades to the impeller.
- Change volute cross section shape (rectangular cross section shape and trapezoid cross section shape.

3. Mesh Generation

Table (1): number of cells and nodes for each mesh

Mesh	Advanced size function	**Number of cells**	Number of nodes	**Pressure(pa)**
A	-	**279667**	56283	**726129**
B	-	**1482776**	295141	**743858**
C	✓	**6760918**	1268336	751782

Mesh (C) was used in all the tests in this research work.

Structured tetrahedral mesh cells was created, advanced size function at curvature and Proximity, fine relevance center, high smoothing and element mid side node settings were used. A mesh independent test was made to. Two meshes were created without advanced size function and one with advanced size function. Table (1) shows the number of cells and nodes for each mesh.

4. Pump performance prediction without changes: Pump has been tested experimentally to verify the result obtained from CFD, the measure values of discharge (Q) and head (H) are listed in table (2).

Table (2): values of discharge (Q) and head (H) from experimental measurements

Q (m^3/hr)	0	49	105	154	202	227	292
H (m)	82.52	83.15	82.17	81.84	79.7	77.12	71.77

Calculations have been performed with CFD software, that solves the finite volume method for the solution of the unsteady 3D incompressible Navier-Stokes equations. Standard k-ε turbulence model with enhanced wall function and PISO (Pressure-Implicit with Splitting of Operators) were chosen for turbulence model and pressure-velocity coupling respectively.

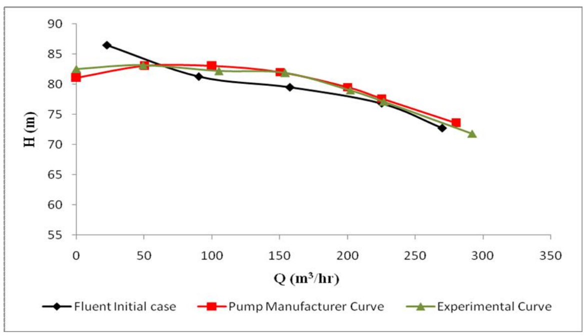


Figure (1): (H, Q) curve for pump manufacture's, experimental and fluent simulation.

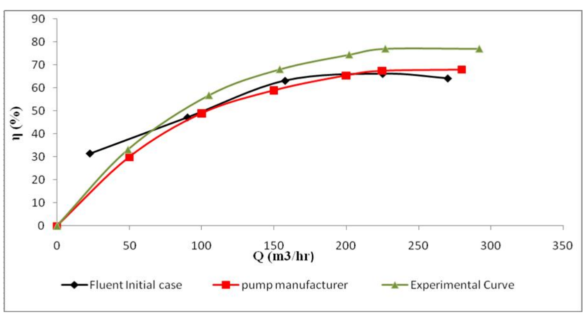


Figure (2): (η, Q) curve manufacture's, experimental and fluent simulation.

Simulation was done at five operating points (0.1Q, 0.4Q, 0.7Q, Q, 1.2Q) it was found that, difference between the head obtained from fluent simulation were (5.62%, -2.41%, -1.92%, -1.06%, -2.47%) respectively to operating points. Difference between the efficiency obtained from fluent simulation were (4.98%, -3.7%, 6.9%, 0.87%, -4.99%).

Figure (1) shows the (H, Q) curve for pump manufacture's, experimental and fluent simulation. Figure (2) shows the (η, Q) curve manufacture's, experimental and fluent simulation. Figure (3) show the pressure distribution through the pump at five simulation points.

5. Effect of adding splitter blades:

Adding splitter blades to the impeller help to increase the head delivered due to reducing turbulence at impeller outlet and there is an increase in slip factor due to adding splitter blades. When splitter blades were added to the impeller it increases the head with average value of (14.4%), it also lead to decease the efficiency by average value (4.2%) and that due to the increase in blade blockage at impeller outlet and increase in fraction at impeller surface.

Figure (4) shows the (H,Q) curve for pump manufacture's, fluent simulation and fluent simulation for splitter blades impeller.Figure (5) shows the (η, Q) curve for pump manufacture's, fluent simulation and fluent simulation for splitter blades impeller. Figure (6) show the pressure distribution for five simulation points for splitter blades impeller.

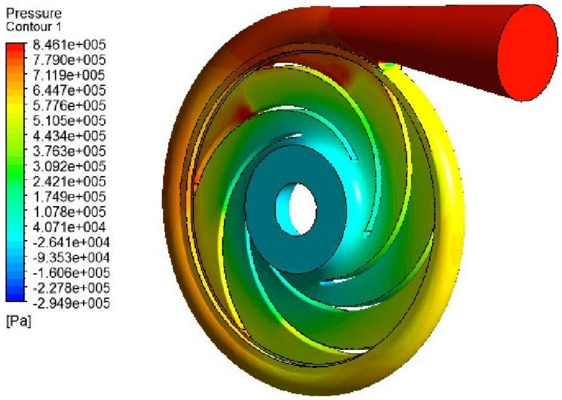
Change in volute cross section shape:

Changing volute cross section shape to rectangular leads to reduce the head by (2.53%, -0.35%, -3.9%, -2.23%, -3.5%),at (0.1 Q) the head increased than the head from the head from simulation of round volute. Changing volute cross section shape to rectangular reduce the efficiency by (-4.6%, -3.5%, -2.17%, -2.02%, -4.9%).

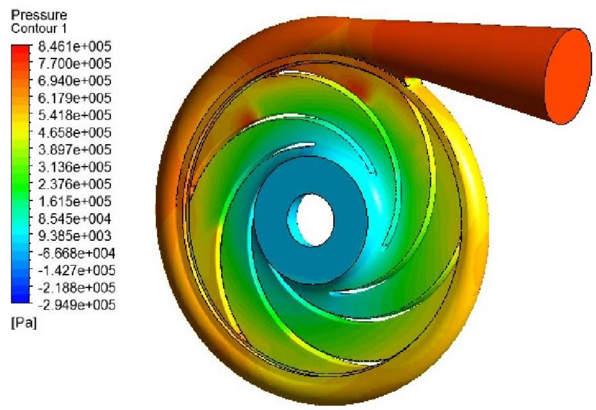
Changing volute cross section shape to rectangular leaded to reduce the head by (0%, -2.6%, -5.4%, -2.6%, -4%), changing volute cross section shape to rectangular reduce the efficiency by (-5.8%, -6.5%, -4%, -3.33%, -4.6%).

The best volute cross section shape was round cross section witch give the higher head and efficiency. The second was rectangular cross section and the last was the trapezoid cross section.

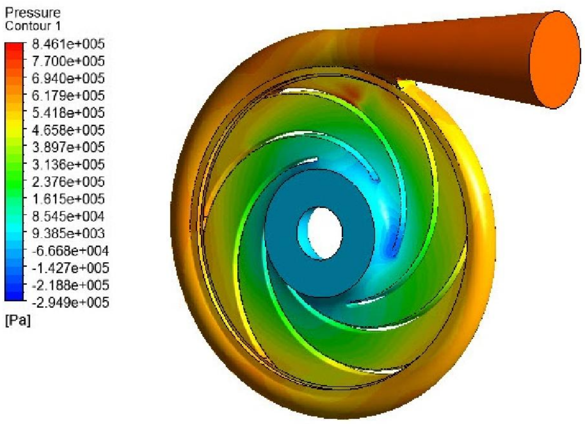
Figure (7) shows the (H, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular volute. Figure (8) shows the (η, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular volute.



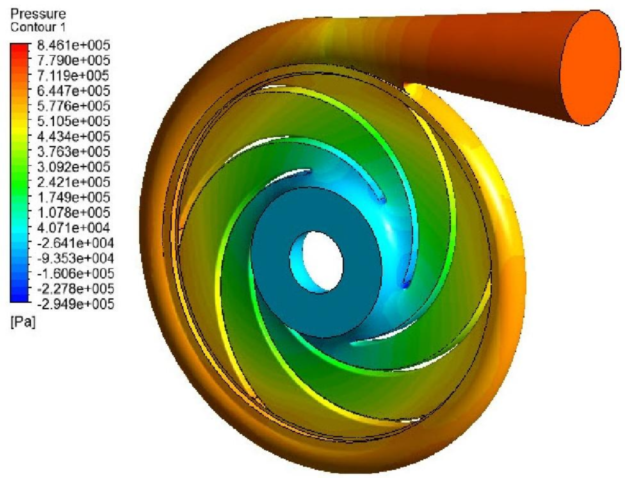
0.1 Q



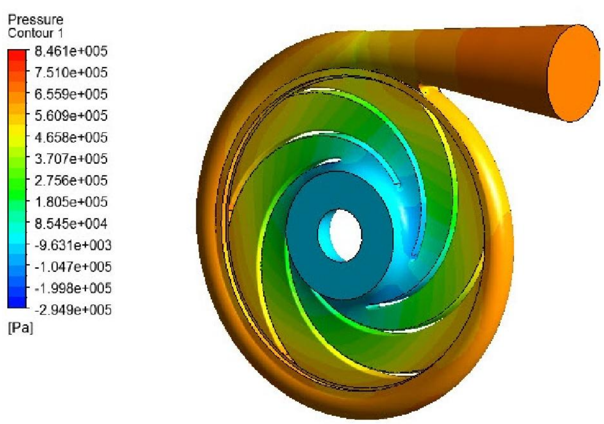
0.4 Q



0.7 Q



Q



1.2 Q

Figure (3): pressure distribution for five simulation points.

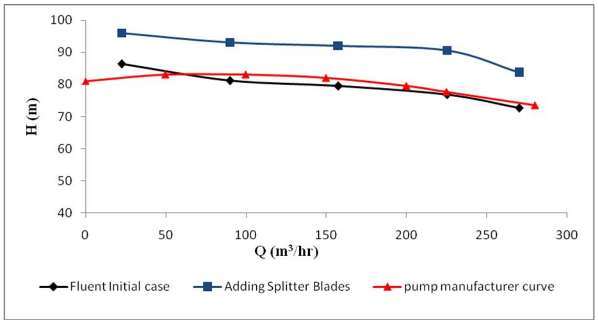


Figure (4) : (H,Q) curve for pump manufacture's, fluent simulation and fluent simulation for splitter blades impeller

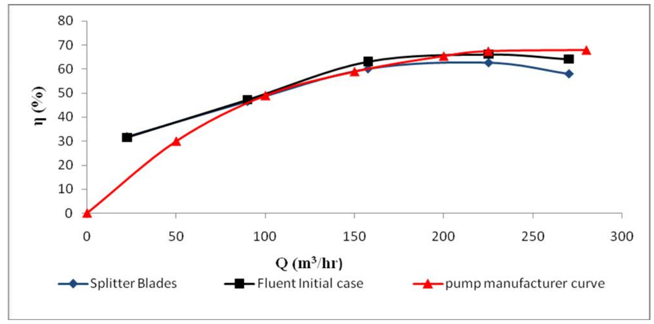


Figure (5): (η, Q) curve for pump manufacture's, fluent simulation and fluent simulation for splitter blades impeller

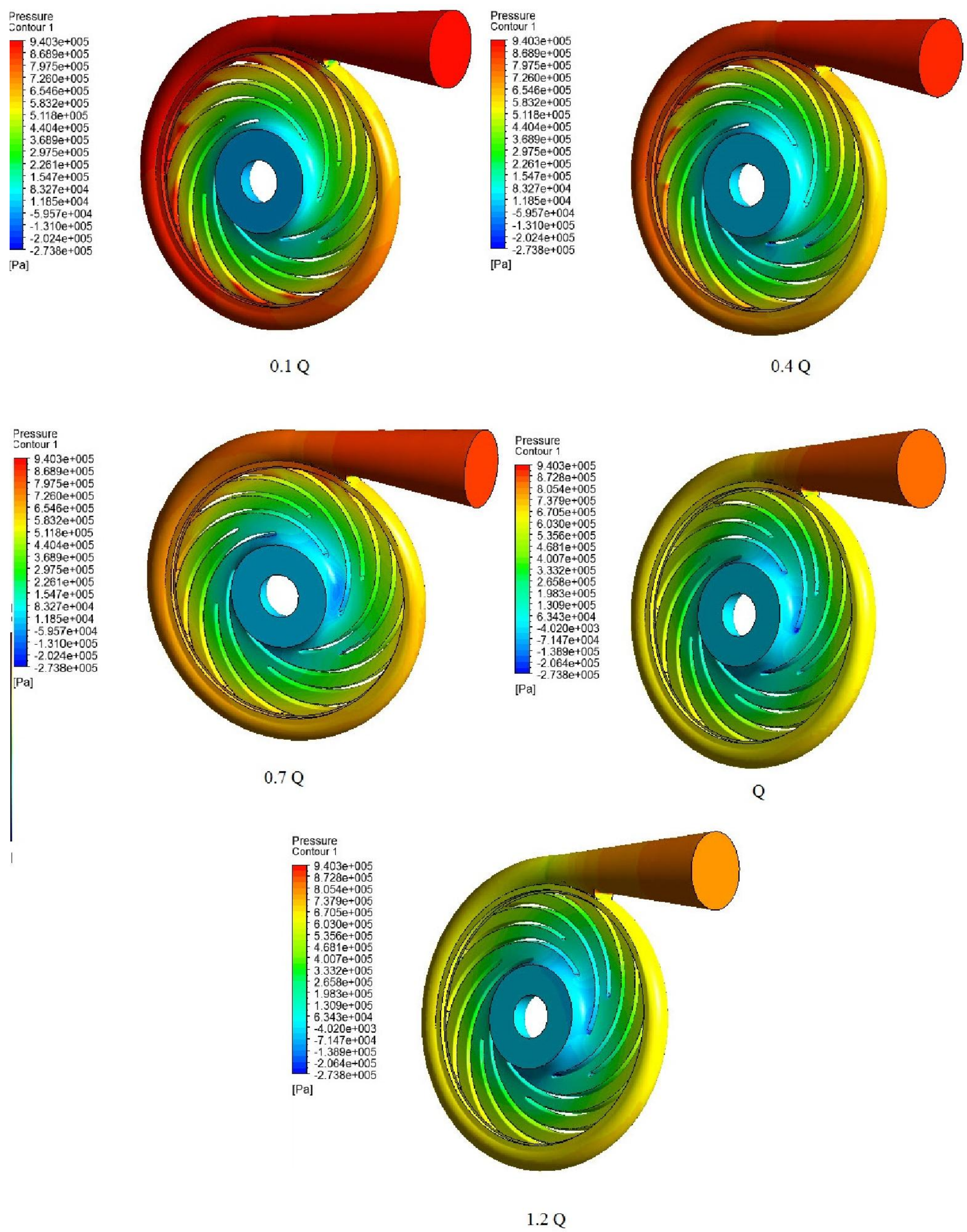


Figure (6): pressure distribution for five simulation points for splitter blades impeller

Figure (9) shows the (H, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular trapezoid volute. Figure (10) shows the (η, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular trapezoid volute.

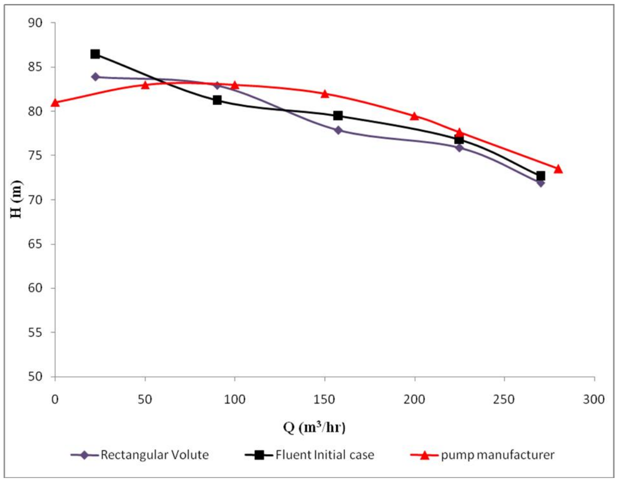


Figure (7): (H, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular volute.

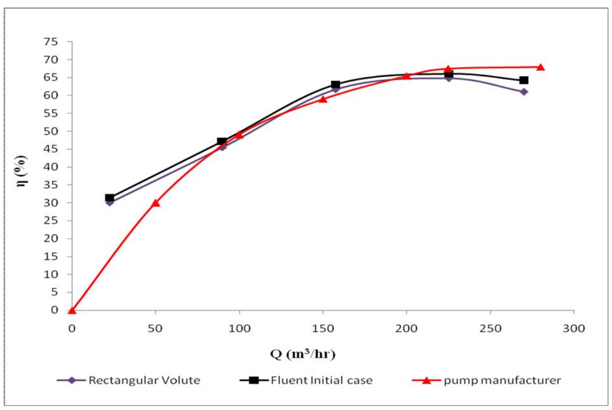


Figure (8):(η, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular volute

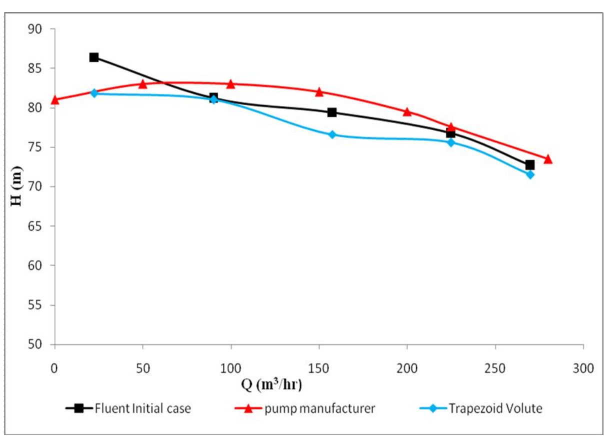


Figure (9):(H, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular trapezoid volute

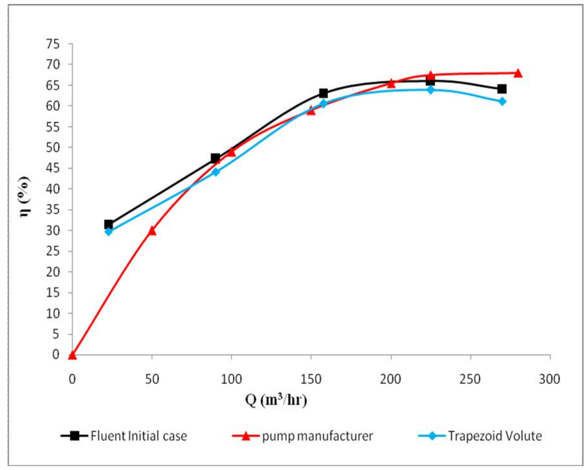


Figure (10):(η, Q) curve for pump manufacture's, fluent simulation and simulation with rectangular trapezoid volute

Conclusions:

Applying CFD simulation that solves the finite volume method to the unsteady 3D incompressible Navier-Stokes equations to simulate flow through pump. Standard k-ε turbulence model with enhanced wall function and PISO (Pressure-Implicit with Splitting of Operators) were chosen for turbulence model and pressure-velocity coupling respectively.

- CFD simulation give good accuracy in prediction pump performance with difference of (3.4%) in head and (3%) in efficiency for present case study.
- Using advanced size function in creating mesh give better simulation results.
- When splitter blades were added to the impeller it increases the head with average value of (14.4%), it also lead to decease the efficiency by average value (4.2%).
- Changing volute cross section shape to rectangular leads to reduce the head by (2.53%, -0.35%, -3.9%, -2.23%, -3.5%), changing volute cross section shape to rectangular reduce the efficiency by (-4.6%, -3.5%, -2.17%, -2.02%, -4.9%).
- Changing volute cross section shape to rectangular leads to reduce the head by (0%, -2.6%, -5.4%, -2.6%, -4%), changing volute cross section shape to rectangular reduce the efficiency by (-5.8%, -6.5%, -4%, -3.33%, -4.6%).

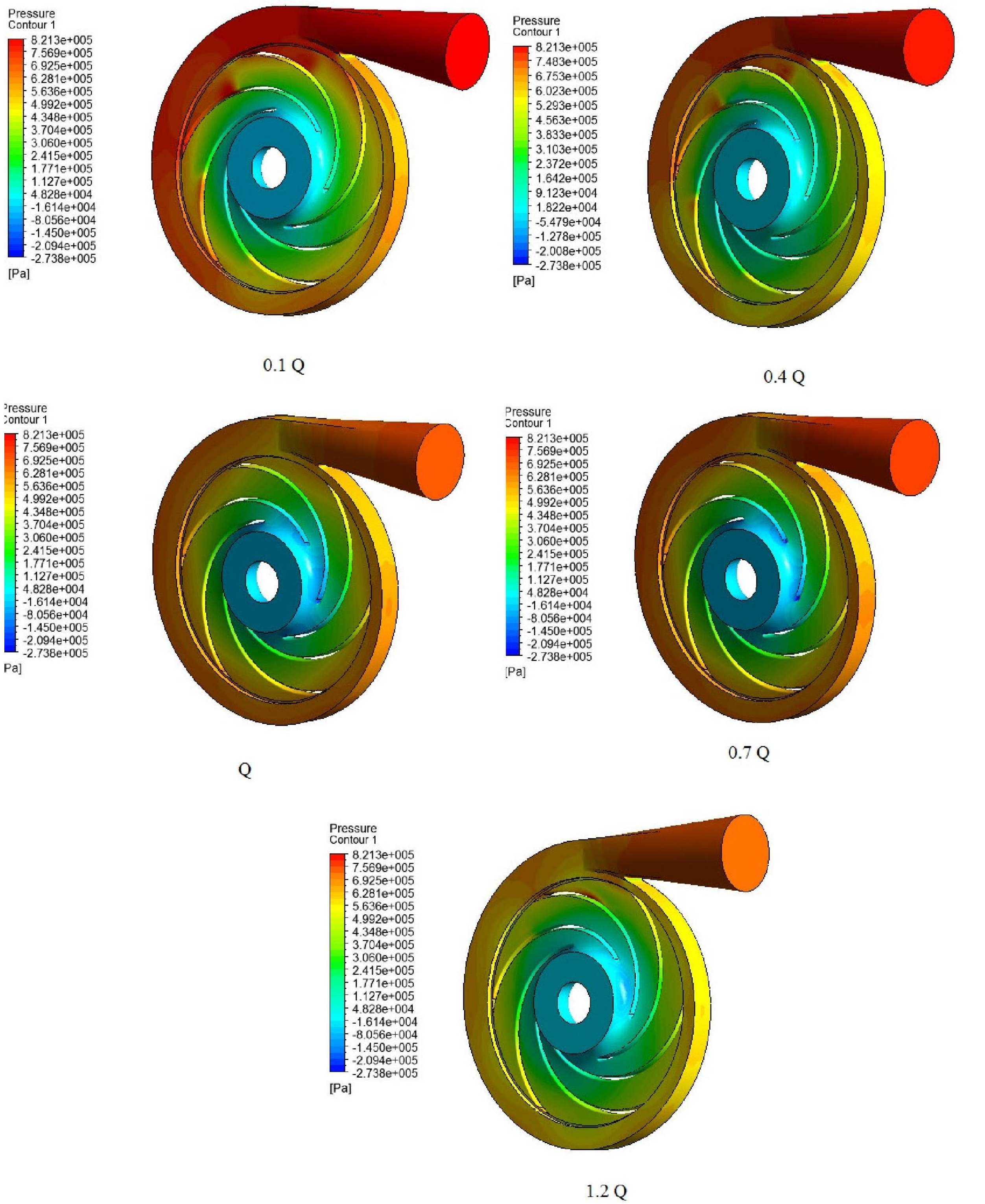


Figure (11): pressure distribution for five simulation points for rectangular volute.

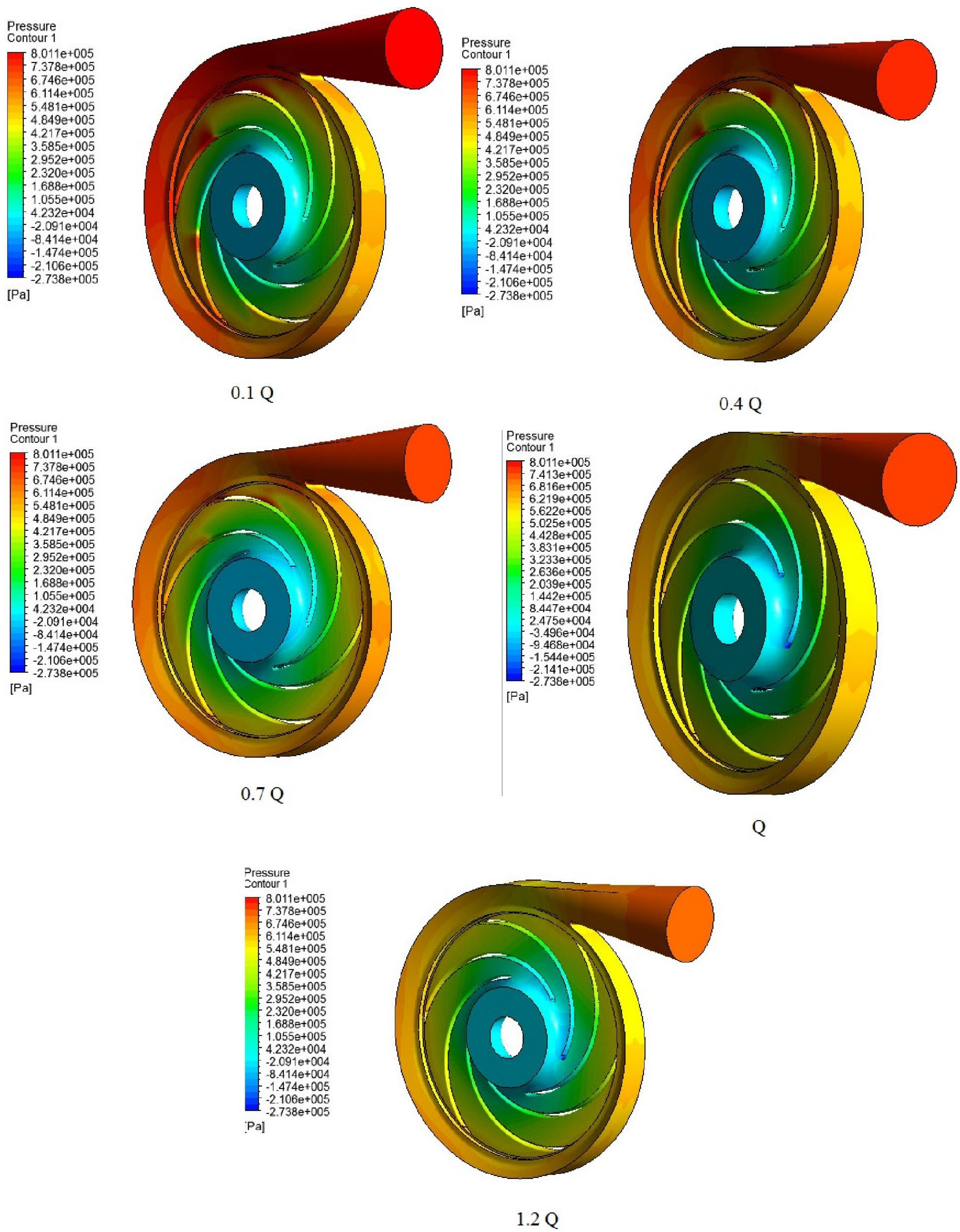


Figure (13) : pressure distribution for five simulation points for trapezoid volute.

References:

1. Aman A., Kore S. and Dribssa E. (2011), "Flow Simulation and Performance Prediction of

Centrifugal Pumps Using CFD-Tool". Journal of EEA, Volume 28, pp (59–65).

2. Bacharoudis E.C., Filios A.E., Mentzos M.D. and Margaris D.P. (2008), "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle". The Open Mechanical Engineering Journal, volume 2, pp (75–83).
3. Chakraborty S. and Pandey K.M. (2011), "Numerical Studies on Effects of Blade Number Variations on Performance of Centrifugal Pumps at 4000 RPM". International Journal of Engineering and Technology, Volume 3, pp (410–416).
4. Dick E., Vierendeels J., Serbruyns S. and Voorde J. V. (2001), "Performance Prediction of Centrifugal Pumps with CFD-Tools". Task Quarterly, Volume 4, pp (579–594).
5. Gölcü M. and Pancar Y. (2005), " Investigation of performance characteristics in a pump impeller with low blade discharge angle". World pumps**,** volume 2005, pp (32–40).
6. Gölcü M., Pancar Y., Ergür H. S. and Göral E. Ö. (2010), " Prediction of Head, Efficiency, and Power Characteristics in A Semi-Open Impeller". Mathematical and Computational Applications, Volume 15, pp (137–147).
7. Gülich J. F. (2010), "Centrifugal Pumps Second edition". Springer, New York.
8. Jinfeng Z., Shouqi Y., Yuedeng F. and Jianping Y. (2013), " Influence Of Splitter Blades On The Total Flow Field Of A Low-Specific Centrifugal Pump".Technology and Research Center of Fluid Machinery Engineering, Jiangsu University, Zhenjiang.
9. Kaewnai S., Chamaoot M. and Wongwises S. (2009), "Predicting performance of radial flow type impeller of centrifugal pump using CFD". Journal of Mechanical Science and Technology, volume 23. pp (1620-1627).
10. Kergourlay G., Younsi M., Bakir F., and Rey R. (2007), "Influence of Splitter Blades on the Flow Field of a Centrifugal Pump: Test-Analysis Comparison". International Journal of Rotating Machinery, Volume 2007, pp (1–13).
11. Kim J.H., Oh K.T., Pyun K.B., Kim C.K., Choi Y.S. and Yoon J.Y. (2012), "Design Optimization of a Centrifugal Pump Impeller and Volute Using Computational Fluid Dynamics". 26th IAHR Symposium on Hydraulic Machinery and Systems.
12. Lobanoff V. S., Ross R. R. (1992), "Centrifugal Pumps Design & Application" Second Edition. Gulf Publishing, Houston, TX.
13. Manivannan A. (2010), "Computational fluid dynamics analysis of a mixed flow pumps impeller". International Journal of Engineering, Science and Technology, volume (2), pp (200–206).
14. Mustafa M. T. and Ibrahim A. M. (2009), "The Performance Characteristics Evaluation Of An Impeller Of Centrifugal Pump With A Volute Using Computational Fluid Dynamics Technique".
15. PPatil P. M. and Todkar R.G. (2013), "An Overview of Effect of Splitter Blades on Centrifugal Pump Performance". International Journal of Engineering Research & Technology, volume 2, pp (2249–2252).
16. Shah S.R., Jain S.V., Patel R. N. and Lakhera V. J. (2010), "CFD Based Flow analysis Of Centrifugal Pump". 37th National & 4th International Conference on Fluid Mechanics and Fluid Power, Madras, Chennai, India.
17. Shah S.R., Jain S.V., Patel R. N. and Lakhera V. J. (2013), "CFD for centrifugal pumps a review of the state-of-the art". Chemical, Civil and Mechanical Engineering Tracks of 3rd Nirma University International Conference.
18. Singh R. R. and Nataraj M. (2012), "parametric study and optimization of centrifugal pump Impeller by varying the design parameter using Computational fluid dynamics". Journal of Mechanical and Production Engineering, volume 2, pp (87–97).
19. Shahin I., Abd Elganny M., Ayad S. and Abd Rabbo M.F. (2010), "Performance and Unsteady Flow Field Prediction of a Centrifugal Pump with CFD Tools". Tenth International Congress of Fluid Dynamics Conference. Ain Soukhna, Red Sea, Egypt.
20. Shojaeefard M.H., Tahani M., Ehghaghi M.B., Fallahian M.A. and Beglari M. (2012), "Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid". Computers & Fluids, volume 60, pp (1–10).
21. Thin K. C., Khaing M. M., and Aye K. M. (2008), "Design and Performance Analysis of Centrifugal Pump". World Academy of Science, Engineering and Technology, volume (22), pp (422–429).
22. Versteeg H. K., Malalasekera W. (2007), "An Introduction To Computational Fluid Dynamics". Pearson Education Limited, England.
23. Wendt J. F. (2009), " Computational Fluid Dynamics An Introduction" Third Edition. Springer-Verlag, Berlin.
24. Yang S., Kong F. and Chen B. (2011), " Research on Pump Volute Design Method Using CFD". International Journal of Rotating Machinery, Volume 2011, pp (1–7).

12/17/2014