Materials Today: Proceedings xxx (xxxx) xxx

Contents lists available at ScienceDirect

Materials Today: Proceedings

journal homepage: www.elsevier.com/locate/matpr

Impeller design and CFD analysis of fluid flow in rotodynamic pumps

Chandrasekaran M. [a,*], Santhanam V. [b,*], Venkateshwaran N. [b]

[a] Vels Institute of Science, Technology and Advanced Studies, Chennai 600117, India
[b] Rajalakshmi Engineering College, Chennai 602105, India

ARTICLE INFO

Article history:
Received 27 June 2020
Received in revised form 23 July 2020
Accepted 27 July 2020
Available online xxxx

Keywords:
Impeller design
Rotodynamic pumps
MATLAB
NX CAD
ANSYS CFX

ABSTRACT

The intricate internal flows in the rotodynamic water pump impellers can be well predicted by using computational fluid dynamics and mathematical modeling. An impeller is a radial type rotor that increases the acceleration of the fluid due to the centrifugal action and the fluid is pumped through the casing to a higher potential head. The efficiency and performance of the centrifugal pump can be enhanced by optimizing certain design parameters of the impeller. In this work, the impeller design and development of potential flow calculations for the liquid flow using the MATLAB software was performed. NX-CAD was used to model the impeller and pump. CFD analysis in ANSYS CFX showed the flow of liquid through the variable cross-section of the volute casing of the pump. Modification in impeller design showed that the overall performance of the pump can be improved. The program developed in the MATLAB also served as a type of calculator for determining the type of pump that can be used for various industrial purposes based on the velocity of the liquid, head and discharge rate.

Selection and peer-review under responsibility of the scientific committee of the International Conference on Newer Trends and Innovation in Mechanical Engineering: Materials Science.

1. Introduction

The requirements for energy efficient and low cost pumps had driven the scientific community to develop new models for pump design. Due to increasing demand for low carbon footprint several researchers [1–6] have attempted to develop innovative materials, new processes and modifications in designs. In this work a modification in design and selection process had been attempted to reduce the energy waste in the rotodynamic pumps. The Impeller of the Centrifugal pump usually has vanes fitted between the shrouds or plates. The crown plate has a suction eye and the base plate is mounted which is keyed to a shaft. The impeller without the crown plate is known as the non-clog or semi-open impeller. Several researchers [7–10] have attempted to optimize the performance and improve the overall efficiency of the rotodynamic pumps by using computational fluid dynamics (CFD) analysis and varying the impeller design parameters suitably. Junaidi et al [11] had shown that the vortices and non-uniformity in the fluid flow can be reduced by introducing inner guide vanes at the entrance of the centrifugal pump, they performed CFD analysis with various design parameters of the guide vanes. They showed that the overall size of the pump can be reduced for the same capacity due to improved aerodynamics.

The impeller parameters significantly affect the overall operating performance of the rotodynamic pumps as they provide the necessary pressure to the fluid. Careful analysis of the impeller profile can significantly improve the performance and efficiency of the pump. Raj et al [12] had reviewed the application of various CFD techniques studied by the researchers, it was reported that two equation k-ε turbulence model along with unsteady Navier–Stokes equations (Reynolds-averaged) were giving closer predictions on the centrifugal pump. It was also reported that further studies on impeller volute interaction and fluid flow study in volute casing can improve the performance of the centrifugal pumps. The work done by Gurupranesh et al [13] predicted the performance of the pump by using virtual prototyping and CFD analysis by fluid flow simulation package. The impeller vane profile was created by using solid works software using point by point method, and the parameters such as pressure distribution, static pressure head, turbulence, fluid velocity and direction of flow. The results were closely in agreement to that of the actual values.

Zhou et al [14] had investigated the three different types of centrifugal pumps with different blades by using CFX software. The

* Corresponding authors.
E-mail address: santhanam.v@rajalakshmi.edu.in (M. Chandrasekaran).

https://doi.org/10.1016/j.matpr.2020.07.637
2214-7853/
Selection and peer-review under responsibility of the scientific committee of the International Conference on Newer Trends and Innovation in Mechanical Engineering: Materials Science.

Please cite this article as: M. Chandrasekaran, V. Santhanam and N. Venkateshwaran, Impeller design and CFD analysis of fluid flow in rotodynamic pumps, Materials Today: Proceedings, https://doi.org/10.1016/j.matpr.2020.07.637

problem was simulated by using standard two equation turbulence κ – ε model. It was reported that the comparison of computational results was in good agreement with that of actual values for twisted blade arrangement but vary significantly with straight blade arrangement. Nigussie et al [15] had used 3d modeling and CFD simulation to determine the velocity profiles and pressure distribution. Rajendran et al [16] had used ANSYS CFX to obtain the pressure plots, flow pattern and blade loading in the impeller passage. The delivery head predicted by CFD for the design flow rate was 9.45 m. Tan et al [17] reported the use of direct and inverse iteration methods to solve the continuity and boundary equation of the fluid flow. Three types of pumps with different wrap angles were numerically simulated, and results indicated that highest efficiency and larger head was obtained for the larger blade wrap angle. The work done by Rajmane et al [18] exhibited that modeling and simulation of impeller can result in precise prediction of pump parameters for better efficiency. Kim et al [19] used response surface methodology (RSM) to design optimized impeller that satisfied design specifications, and it was verified by numerical analysis. Minggao et al [20] studied the effect of different impeller outlet diameters on the flow field in six different types of centrifugal pumps. It was reported that the change of outlet parameters directly affected the total pressure and static pressure at the volute and at the impeller. Also, the predicted simulation results were in accordance with the cutting law with good accuracy. The above literature shows that the studies are focused on the improvement in the performance of the machines [21–25] by using different approaches [26–32] for various applications and processes.

In the present study the 3D model of the impeller in a centrifugal pump had been created using the NX CAD software using the circular arc method. MATLAB coding was used to plot the characteristic curve of a centrifugal pump at different speeds. Finally, CFD analysis was performed by using ANSYS CFD to predict the velocity profile and pressure distribution in the impeller.

2. Centrifugal pump impeller design

2.1. Design procedure

The CAD model of the impeller was done using NX CAD software. the design procedure involves the computation of required

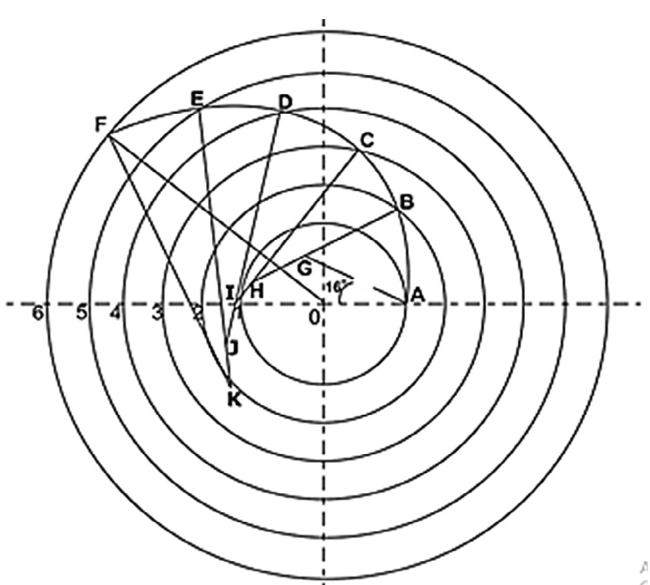

Fig. 1. Generation of blade profile.

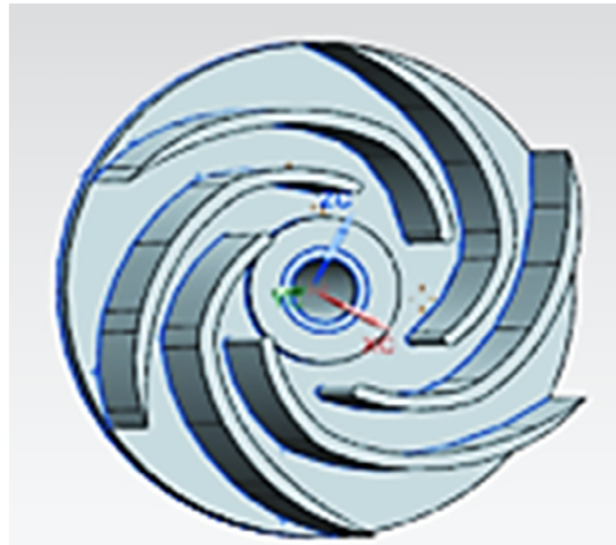

Fig. 2. Impeller 3D model in NX CAD.

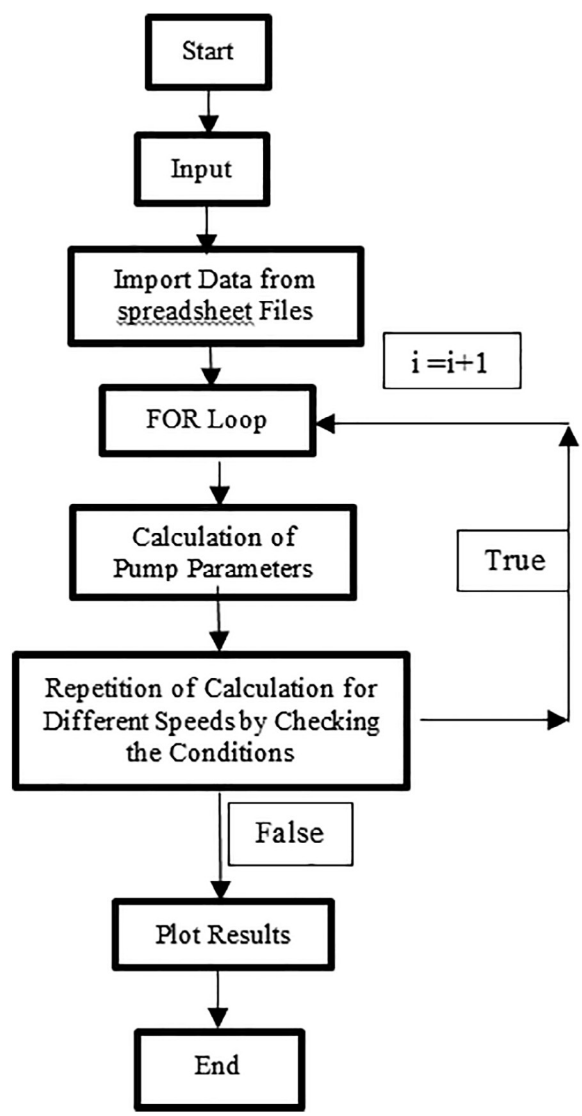

Fig. 3. Flowchart for re-plot of the pump curve.

Table 1
Boundary Conditions Used in CFD Analysis.

S. No.	Type of Boundary Condition	Values
1	Mass flow rate	0.35 m^3/kg
2	Inlet Pressure	1 bar
3	Rotation speed	2000 rpm
4	Number of Iterations	625

water path by using the laws of fluid motion. Fig. 1 shows the generation of blade profile. Initially two concentric circles representing the outer and inner diameter of the impeller were drawn. In the next step, five concentric circles were drawn at equal distance starting from the inner circle to the outer circle. Then, a line was drawn from point A at an angle of the inlet blade angle till the distance of first radius of curvature value. With G as center and AG is radius, an arc was drawn till the arc meets the next circle [29–34]. Another line was drawn from B, which passes through the previous center and the length is equal to the second radius of curvature, with H as center and BH is radius, an arc was drawn till the arc meets the next circle. This procedure was repeated until the curve meets the external diameter of the impeller. The curve was then off-set with internal blade thickness at the entry of the blade and external blade thickness at the exit of the blade as shown in Fig. 2.

2.2. MATLAB programming for selection of pump

MATLAB program was used to plot the pump characteristic curve which can be used as a tool for selection of pump based on the speed and delivery head. The pump performance curves were converted into a digital data using Engauge digitizer software, then, the data points were derived from the pump curve and exported to the excel sheet using .csv extension. This file was then loaded into the MATLAB program in the form of an array and the data was re-plotted again in the form of a graph. The program also calculates all the parameters of the pump using the formulae

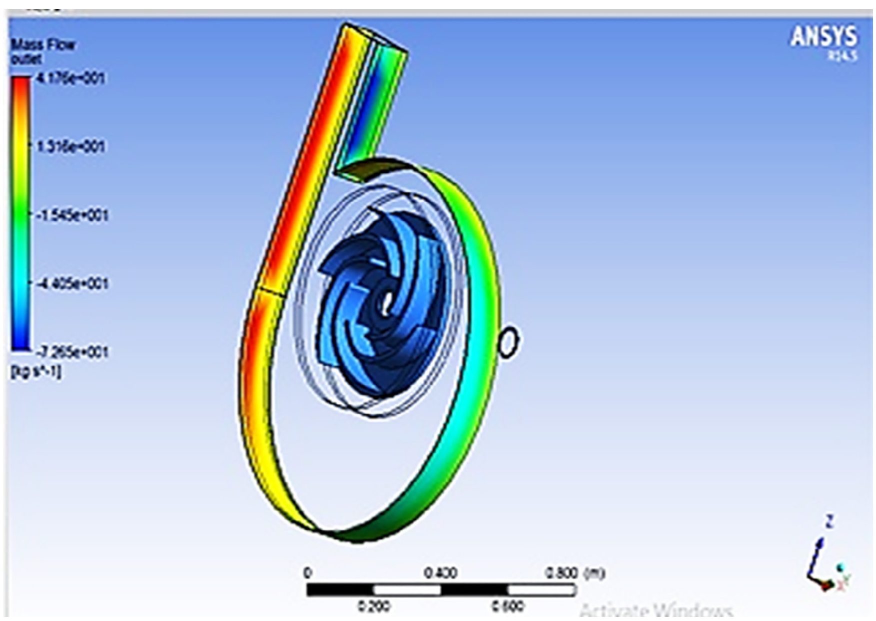

Fig. 4. Mass flow rate distribution.

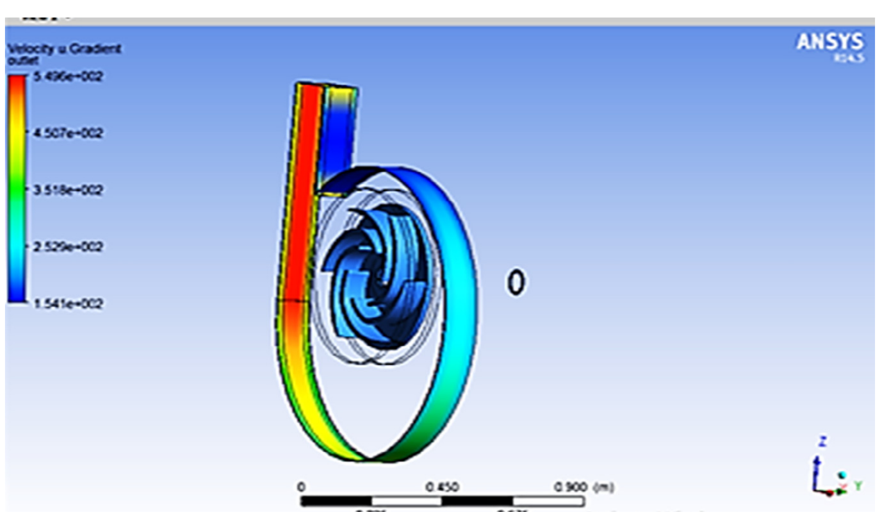

Fig. 5. Velocity gradient of the liquid flow.

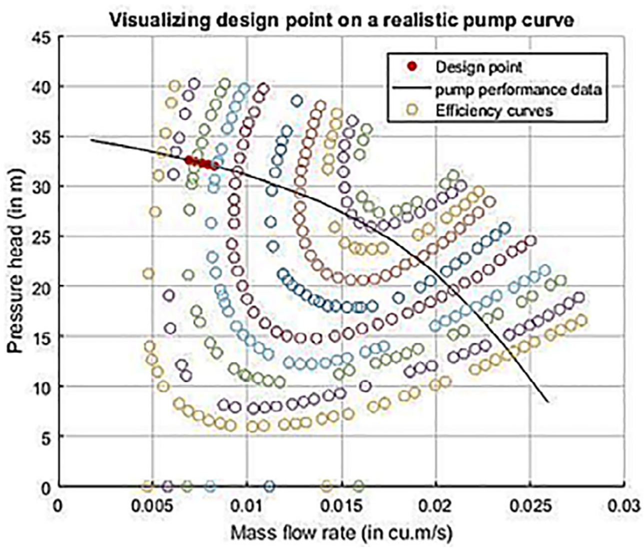

Fig. 6. Re-plot of pump curve in MATLAB.

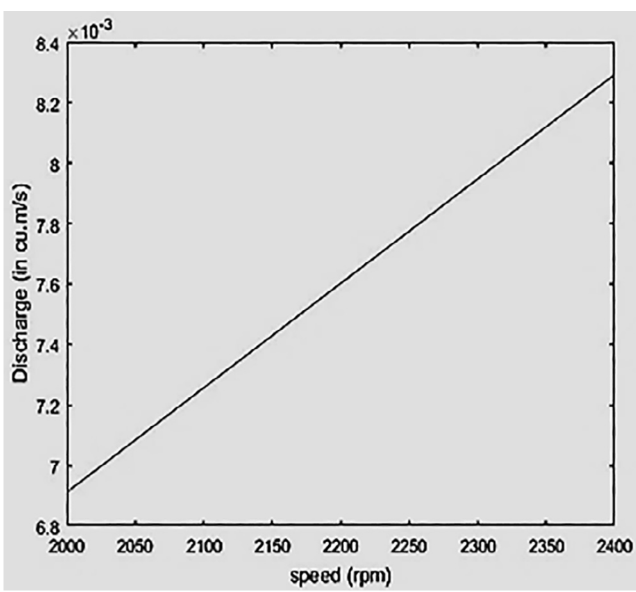

Fig. 7. Plot of speed versus discharge.

obtained from the velocity triangle and then links the data points obtained with the pump performance curve and the data obtained from the formulae. The final graph obtained was used to choose the pump with best efficiency, mass flow rate and pressure head. The flow chart of the program used in this study is presented in Fig. 3.

2.3. CFD analysis of the fluid flow in the pump

ANSYS CFD had been used in this study to simulate the fluid flow in the centrifugal pump, the flow was assumed to be turbulent and the standard $\kappa - \varepsilon$ model was used to simulate the fluid flow. Standard wall conditions were considered for the volute casing, impeller and pump walls. The input velocity was assumed to be constant and the factors considered in the analysis is given in Table 1.

3. Results and discussion

The 3D model of the impeller was created using NX CAD software which was used as input in ANSYS simulation. Figs. 4–6 show the results given by the CFD simulation. The CFD analysis showed the mass flow rate distribution along various cross section of the pump casing, while the velocity gradient distribution depicts the changes in the velocity of the fluid along the various cross section of the pump casing.

Characteristic curves are essential to be familiar with for the selection of suitable pump. Fig. 6 shows the re-plot of the pump performance curve using MATLAB. The design spot on the diameter curve indicates the selection of the optimum kind of pump for a particular application with the best efficiency available. The Fig. 7 shows the plot of speed from 2000 rpm to 2400 rpm in steps of 50 rpm increment versus discharge rate. The design modification, which was done on the blades of the impeller showed an increase in the overall efficiency which was greater by 10% than the previous pumps.

The program that had been developed in the MATLAB helps the industrialists in the ease of choosing the best type of pump that can be used, according to the requirement of the customer. This also helps in reducing the time in choosing the pump.

4. Conclusions

The vane profile was developed in NX-CAD software by circular arc method. The CFD analysis showed the distribution of various parameters along various cross section of the pump which indicated the proper flow of the fluid. The MATLAB program helped in choosing the best suitable pump according to the requirement and the application. A framework was built to perform simplified pump calculation and for displaying actual pump performance curve data for manufacturer. A systematic procedure was developed to digitize experimental data from images to make use of in the program. Linear interpolation was used to calculate realistic pressure drop data based upon the simplified pump model and manufacturer's model. This framework can be applied to extract and store the performance model of 100 s of pump model that can be used for commercial purpose. Such a tool can be useful for designers and analyst in selecting the pump that meets the design and cost requirement.

Declaration of Competing Interest

The authors declare that they have no known competing financial interests or personal relationships that could have appeared to influence the work reported in this paper.

References

[1] S. Ramasubramanian, M. Chandrasekaran, S. Baskar, A. Dowhithamaran, Mater. Today (2020), https://doi.org/10.1016/j.matpr.2020.05.637.
[2] Jishuchandran, K. Manikandan, R. Ganesh, S. Baskar, Int. J. Ambient Energy (2020) 1–16, https://doi.org/10.1080/01430750.2020.1725637.
[3] S. Baskar, V. Vijayan, I.I. Premkumar, D. Arunkumar, D. Thamaran, Mater Today. (2020), https://doi.org/10.1016/j.matpr.2020.05.352.
[4] B.S. Kumar, V. Vijayan, N. Baskar, Mech. Mech. Eng. 20 (2016) 347–354.
[5] D. Arunkumar, M. Ramu, R. Murugan, S. Kannan, S. Arun, S. Baskar, Mater Today. (2020), https://doi.org/10.1016/j.matpr.2020.01.220.
[6] D. Arunkumar, M. Ramu, R. Murugan, S. Kannan, S. Arun, Sanjeevi Baskar, Mat Today. 1–5. https://doi.org/10.1016/j.matpr.2020.01.220
[7] S.R. Shah, S.V. Jain, R.N. Patel, V.J. Lakhera, Procedia Eng. 51 (2013) 715–720.
[8] L. Hedi, K. Hatem, Z. Ridha, Int. Renewable Energy Congress (2010) 300–304.
[9] N. Dash, A.K. Roy, K. Kumar, Mater Today 5 (2018) 4460–4466.
[10] J.P. Yuan, Y.W. Zhu, A.X. Ge, J. Adv. Mater. Res 945 (2014) 914–923.
[11] M.A. Junaidi, N.L. Kumari, M.A. Samad, G.S. Ahmed, Mater Today 2 (2015) 2073–2082.
[12] P.P. Raj, P.S. Gupta, P.P. Kumar, G. Paramesh, K. Rohit, Mater Today 21 (2020) 175–183.

[13] P. Gurupranesh, R.C. Radha, N. Karthikeyan, IOSR-JMCE, (2010) 2278–1684.
[14] W. Zhou, Z. Zhao, T.S. Lee, S.H. Winoto, Int. J. Rotating Mach. 9 (2003) 49–61.
[15] T. Nigussie, E. Dribssa, Int. J. Eng. Res. Gen. Sci. 3 (2015) 668–677.
[16] S. Rajendran, D.K. Purushothaman, Int. J. Eng. Res. Technol. 3 (2012) 2278–2281.
[17] L. Tan, B. Zhu, S. Cao, H. Bing, Y. Wang, Chin. J. Mech. Eng.-En 27 (2014) 171–177.
[18] S.M. Rajmane, S.P. Kallurkar, Int. Res. J. Eng. Technol. 2 (2015) 984–988.
[19] J.H. Kim, H.C. Lee, J.H. Kim, S. Kim, J.Y. Yoon, Y.S. Choi, J. Mech. Sci. Technol. 29 (2015) 215–225.
[20] T. Minggao, L. Houlin, W. Yong, W. Kai, D. Liang, J. Irrig. Drain E-ASCE 5 (2009) 314–318.
[21] V. S Shaisundaram, M. Chandrasekaran, Mohan Raj, S., & Muraliraja, R. International Journal of Ambient Energy, 41(2020), 98–104.
[22] V.S. Shaisundaram, M. Chandrasekaran, M. Shanmugam, S. Padmanabhan, R. Muraliraja, L. Karikalan, Int. J. Ambient Energy (2019) 1–5.
[23] V.S. Shaisundaram, M. Chandrasekaran, S. Mohan Raj, R. Muraliraja, T. Vinodkumar, Int. J. Ambient Energy 40 (2019) 699–703.
[24] V.S. Shaisundaram, L. Karikalan, M. Chandrasekaran, Int. J. Vehicle Struct. Syst. 11 (2019).
[25] R. Muraliraja, J. Sudagar, R. Elansezhian, A. Raviprakash, R. Dhinakaran, V.S. Shaisundaram, M. Chandrasekaran, Arabian J. Sci. Eng. 44 (2019) 821–828.
[26] R. Muraliraja, R. Elansezhian, Trans. he IMF 93 (2015) 126–132.
[27] R. Muraliraja, D. Sendilkumar, D. Elansezhian, Int J Electrochem Sci 10 (2015) 5536–5547.
[28] S. Baskar, V. Vijayan, S. Saravanan, A.V. Balan & A. Godwin Antony, International Journal of Mechanical Engineering and Technology, 9 (2018) 91–96.
[29] A. Godwin Antony, V.Vijayan, S.Saravanan, S.Baskar, M.Loganathan, International Journal of Mechanical Engineering and Technology, 9 (2018) 681–691.
[30] S. Saravanan, A. Godwin Antony, V. Vijayan, M. Loganathan, S. Baskar, International Journal of Mechanical Engineering and Technology, 1 (2019) 785–790.
[31] S. Dinesh, A. Godwin Antony, K. Rajaguru, V. Vijayan, Mech. Mech. Eng. 21 (2017) 17–28.
[32] S. Dinesh, K. Rajaguru, V. Vijayan, A.Godwin Antony, Mechanics and Mechanical Engineering , 20(2016) 451–466.