International Journal of Applied Engineering Research ISSN 0973-4562 Volume 13, Number 11 (2018) pp. 9982-9987

Design and Development of Blower Impeller by Reverse Engineering for Noise Reduction using CFD

Dr. Vinayak R. Naik [1], Vikas B. Magdum [2]

[1] Professor and Head, Department of Mechanical Engineering,
D.K.T.E. Society's Textile and Engineering Institute, Ichalkaranji, Maharashtra – 416115, India.

[2] Assistant Professor, Department of Mechanical Engineering,
D.K.T.E. Society's Textile and Engineering Institute, Ichalkaranji, Maharashtra- 416115, India.

Abstract

This paper presents a design methodology to examine various parameters of the centrifugal blower using computational fluid dynamics approach. The effects of blower geometry, blower speed, impeller geometry, blade height and impeller diameter have been assessed. Noise level and speed are the output parameters calculated. High rotating frequency of blower produces high level of noise. Thus noise reduction is key parameter in design of hand blower. Using Computational fluid dynamics (CFD) noise source is analyzed. Different combinations of blower geometry, impeller diameter, blade height, no of blades are carried out and is optimized for noise reduction. This method resolves the noise problem of blower in household and industrial appliances. The development is carried out using reverse engineering and rapid prototyping technique.

Keywords: - Noise reduction, blower impeller, orthogonal array, CFD analysis, reverse engineering, rapid prototyping.

INTRODUCTION

Portable blowers are in common use by homeowners and professionals to remove debris from yards and driveways without water or other equipment such as rakes or brooms. The portable blowers can be powered by either an electric motor or a gas-powered motor. Furthermore, some of the blowers can be converted into vacuum units in which leaves or similar debris can be vacuumed into a bag or other container.

When operated as blowers, the units provide a sweeping action using a fast moving stream of air produced by an impeller rotating within the housing of the blower. The impeller draws air into the unit through an inlet and forces the air out of the unit through an outlet. An exhaust or blower tube is typically fitted over the outlet to contain the air stream to a nozzle at the end of the blower tube. The length of the blower tube typically allows a user to stand and hold the portable blower while locating the nozzle near the ground. In addition, the nozzle outlet opening is typically smaller than the outlet at the housing to increase the velocity of the air as it exits the nozzle at the end of the blower tube. The parameters involved were identified and experiment are design using Taguchi technique was carried out to optimize noise level. For the present work CFD was used as a tool for analysis of noise produced due to various parameter. The impeller was developed using rapid prototyping technique. The results of improved model were compared with existing one.

LITERATURE REVIEW

Robert Paul. M studied the effects of blower geometry, blower speed, impeller geometry, blade design and fillet radius on discharge and efficiency [1].

V. H. Chaudhari et. al design centrifugal blower by comparative analysis of number of blades, blade outlet angle and radial gap between impeller and volute casing[2].

M. Natraj found that CFD analysis is in good agreement with the experimental results [3].

K. Vasudeva Karanth et. al develop PIV and CFD tools to arrive at a practical solution compatible with the physical nature of flow [4].

C. N. Jayapragasan et. al used Taguchi method for optimum cases, CFD for meshing and analyzed using FLUENT [5].

Yogesh R. Pathak et. al analyze the 3-D flow field. Fluid domain is created and simulation is done with the CFD software code ANSYS CFX. Three dimensional Navier-Stokes equations are used to analyze the flow [6].

Kusekar S. K. et. al used modal and CFD analysis using FEA for reducing vibrations and failure of the centrifugal blower fan [7].

V. Krishna et. al meshed blower by three dimensional tetra mesh by using HYPERMESH 9.0. Numerical analysis is performed using CFD Fluent software package for the three types of impellers [8].

Girish A. R. et. al used CFD Simulation approach to visualize the flow condition inside centrifugal blower and offers the variable design information of the centrifugal blower [9].

G.V.R. Seshagiri rao et. al model the system using CATIA software and CFD techniques are implemented for the analysis [10].

Seung Heo et. al using hybrid computational aero acoustic techniques of the CFD, the acoustic analogy and the boundary element method (BEM) [11].

The review shows that Taguchi method used for design of experiments and analysis is carried out using CFD FLUENT. The parameters such as no. of blades, blade height and impeller diameter used as input parameters. The characteristics properties like noise level and speed are optimized. Reverse engineering technique used for noise reduction.

METHODOLOGY

Blower is electrically operated. The existing impeller has 10 blades and it is operated by solenoid motor. Motor is of 700 watts capacity with nominal velocity of 30 m/s. Specification and dimensions of blower are given in Table 1 and Table 2 respectively.

Table 1. Specification of blower

Sr. No.	Parameter	Quantity
1	Blowing Air Quantity	2.8 m^3/min
2	Rated Frequency	50/60 HZ
3	Rated Power Input	700 w
4	No Load Speed	1600 rpm
5	Noise	110 db

Table 2. Dimensions of blower

Sr. No.	Parameter of Component	Dimension
1	Impeller Diameter	1100 mm
2	Blade Thickness	2 mm
3	Blade Angle	28°
4	Blade Height	28 mm
5	No of Blades	10
6	Diameter of casing	150 mm
7	Diameter of hole on casing	40 mm
8	Height of casing	30 mm

3D Scanner

3D scanner of FARO shown in Fig. 1 is most trusted source for 3D measurement and imaging. This is having manual operated master arm and automatically give response to convert in 3D modeling. This machine use laser scanner (red color light source). The scanned product file stored in STL file format. 3D Measurement Technology from FARO permits micron-level precision 3D measurement, imaging, and scanning for inspection, CAD-to-part analysis, alignment, reverse engineering and all other production and quality assurance applications.

Figure 1. Working of 3D scanner

The scanned model is imported to 3D builder to remove error and obtain uniformity in design which can be further imported to Catia V5 is shown in Fig, 2.

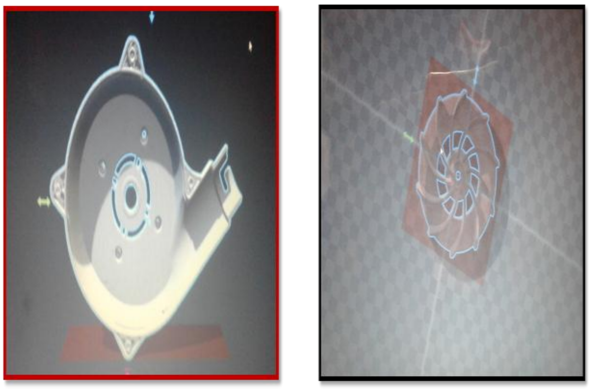

Figure 2. Casing and Impeller

Solid Modeling

Reverse engineering is the process in which the scanned model is converted into solid modelling. It is modern technique used in many design industries for accuracy of surfaces and dimensions. A scanned data collected by software. Scanned data is available in STL file format. Scanned data is used for further processes. Model is shown in Fig. 3.

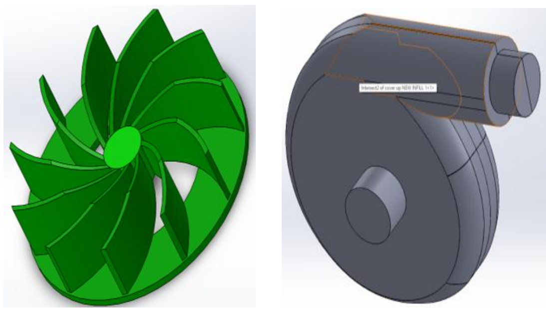

Figure 3. Assembly of blower

ANALYSIS OF PARAMETER

Depending upon results obtain by CFD analysis and Taguchi's philosophy of robust design is used to resolve the problem of noise reduction.

Step of parameter design are as follow-

1) Select different parameters to be evaluated.
2) Different parameter selected are no. of blades (A1, A2, A3), blade height (B1, B2, B3) and impeller diameter (D1, D2, D3).
3) Selection of appropriate orthogonal array.
4) Construct "inner array" consist of control array.
5) Construct "outer array "that contain noise factor and their setting which are under investigation.
6) Then combine "inner" and "outer" array to "product array" or design layout.
7) Conduct the experiment as per product array. Then mean is calculated.
8) Standard deviation is calculated.
9) Desired parameter are determined through analysis of signal to noise ratio where factor level that maximize the appropriate SN ratio are optimal
10) Construct the orthogonal array and is shown in Table 3.

Table 3. Orthogonal array

Model no.	No. of blades	Blade height (mm)	Impeller diameter (mm)
1	10	28	110
2	10	30	105
3	10	32	100
4	12	28	110
5	12	30	105
6	12	32	100
7	14	28	110
8	14	30	105
9	14	32	100

CFD ANALYSIS

To achieve the goals, it is necessary to do computational study of the following design parameters:

Noise, blade angles, angular velocity, Flow rates, number of blades.

Based on the CFD simulations blower noise level predicted using computational aero-acoustics models. It is able to predict noise generation due to turbulence and an unsteady blades interaction.

ANSYS CFD solvers are based on the finite volume method.

$$\frac{\partial}{\partial t}\int_V \rho\phi\, dV + \oint_A \rho\phi \mathbf{V}\cdot d\mathbf{A} = \oint_A \Gamma_\phi \nabla\phi \cdot d\mathbf{A} + \int_V S_\phi\, dV$$

Steps in CFD analysis are shown in Fig. 4:

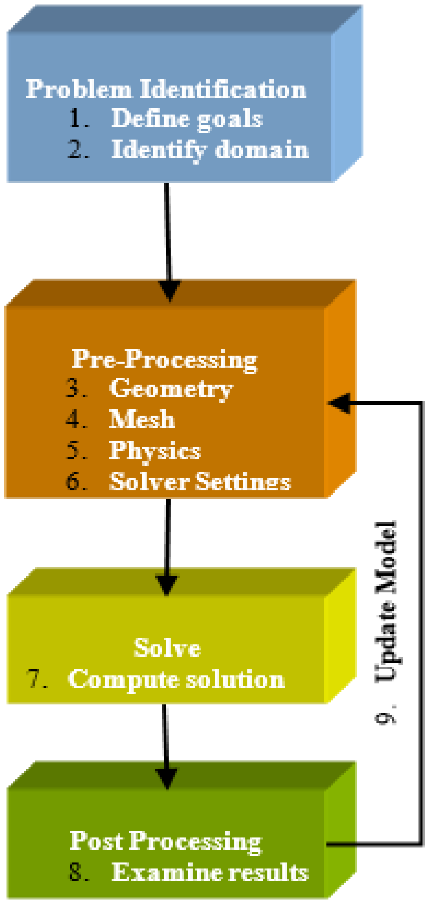

Figure 4. Steps in CFD analysis

CFD analysis was carried out for 9 models build based on Taguchi methodology. It was observed that model no. 4 exhibit low noise level. Hence the details of the same are presented in the paper. This model was taken further for actual impeller development using rapid prototyping technique. The results of noise measurement are compared with the existing one and that of CFD model.

Actual process of analysis

Firstly, geometry is imported in ANSYS Workbench 15.0 and analysis is carried out. Geometry is checked out properly and meshing is carried out providing boundary condition. Inlet and outlet condition like mass flow rate, static pressure, impeller rotating speed are setup. Then solution obtained along with graph. Result obtained are streamlined to obtained level of parameter like pressure, temperature. Animation for observing flow of fluid, Pressure streamline and temperature streamline are shown in Fig. 5 and Fig. 6 respectively.

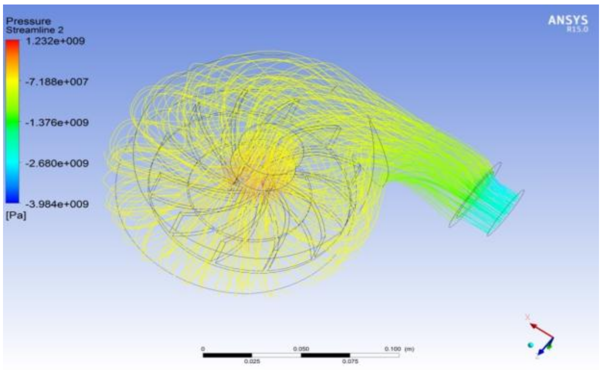

Figure 5. Pressure streamline

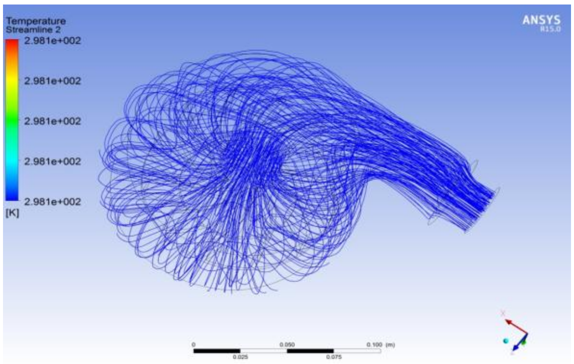

Figure 6. Temperature streamline

Solution graph

Pressure streamline are provided which specify change of pressure as it passes from inlet to outlet. It gives level of pressure high and low through colour. Temperature streamline are specified. Graph is obtained as shown in Fig. 7.

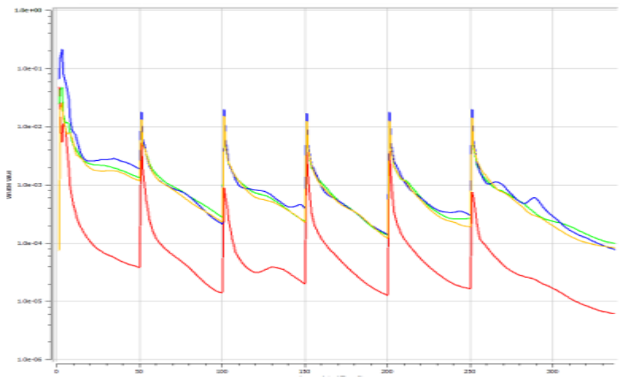

Figure 7. Solution graph in CFX

Procedure for fluent analysis

This is similar analysis such as CFX. It includes import geometry and meshing which provides selection such as inlet, outlet and wall enclosure. Provide state of casing and impeller as fluid and solid respectively. Setup includes providing boundary condition, material, model then run calculation by providing iteration. Contour of power level and velocity is obtained.

According to analysis of existing model and Taguchi model optimum results are obtained with streamline.

Fluent analysis of existing model

Velocity contour and power level contour of existing model are shown in Fig. 7 and Fig. 8 respectively.

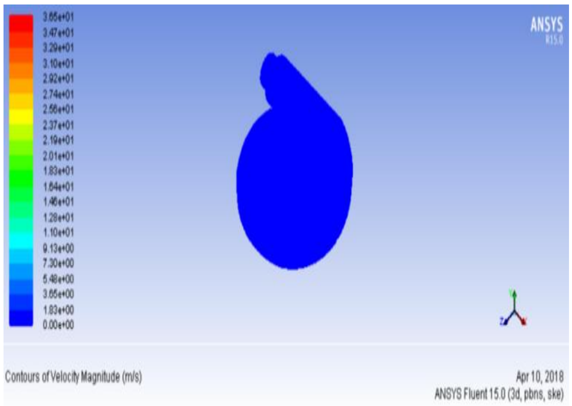

Figure 7. Velocity Contour

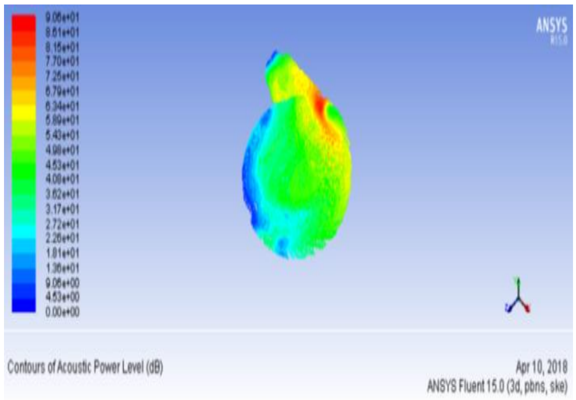

Figure 8. Power level contour

Fluent analysis of improved model

Velocity contour and power level contour of improved model are shown in Fig. 9 and Fig. 10 respectively.

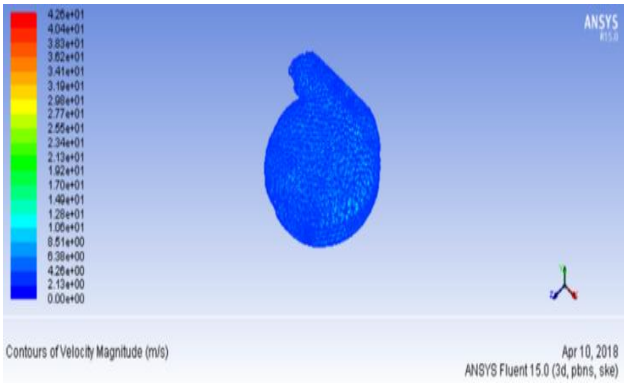

Figure 9. Velocity Contour

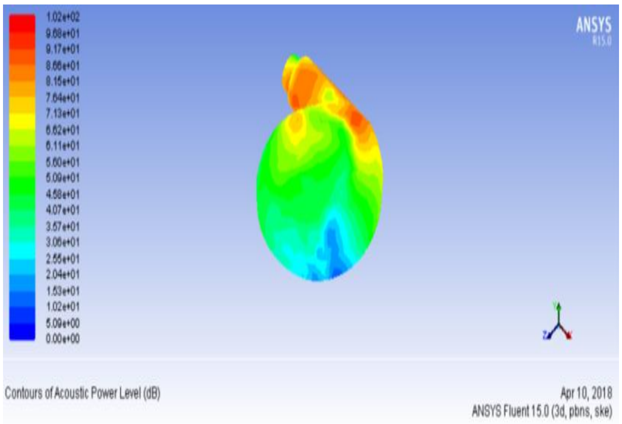

Figure 10. Acoustic Power level contour

All above figure shows contour of noise level (db) and velocity (m/s). Process of analysis includes level of db and velocity according region. It is representing in form of streamline in rainbow colour. Result obtained are optimum. Solution graph obtained by FLUENT is shown in Fig. 11.

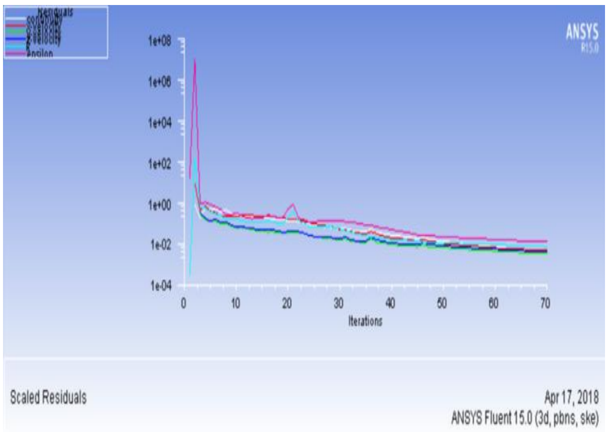

Figure 11. Solution graph in FLUENT

RESULT AND DISCUSSION

Velocity and noise level using FLUENT are shown in Table 4.

Table 4. Velocity and noise level

Model no.	Velocity (m/s)	Noise level (db)
1	36.5	102
2	16.55	90.86
3	16.58	87.41
4	15.59	79.28
5	42.6	90
6	14.06	87.6
7	35.56	93
8	35.567	93.79
9	29.46	91.04

Material selection for 3D printer

The material selected for 3D printer is ABS.

ABS (Acrylonitrile butadiene styrene) is used in very large variety of applications in the industry. ABS used in manufacturing of pipes (drain, waste, vent pipes), automotive components, electronics assemblies, kitchen appliances, protective headgear, toys etc.

3D printing process is shown in Fig. 12.

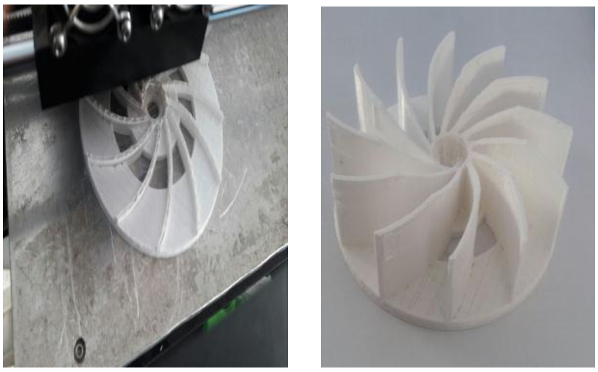

Figure 12. 3D printing process of impeller

This impeller showing excellent performance according to CFD FLUENT workbench.

- Impeller blades- 12
- Impeller blade height-30 mm
- Impeller diameter-105 mm

The trial is taken with help of decibel meter and anemometer as shown in Fig. 13.

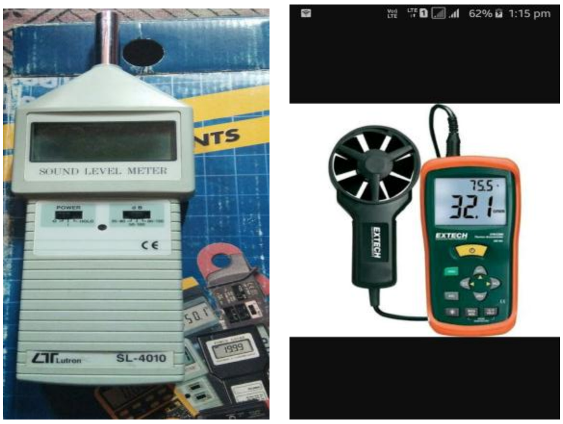

Figure 13. Decibel meter and Anemometer

The noise level and speed of improved rapid prototyping model are measured and is shown in Table 5.

Table 5. Noise level and speed

Decibel meter (db)		Anemometer (m/s)	
Existing	Improved with RP	Existing	Improved with RP
110.4	107.6	30.6	36.4

CONCLUSION

The CFD analysis provide guideline for optimization of noise level of blower impeller. The experimentation was carried out using Taguchi L9 orthogonal array. The optimum combination was chosen from development. It was observed that, improvement in noise level was observed, with the model actually using rapid prototyping. Also the velocity was substantially improved. The noise level decreases from 110.4 db to 107.6 db. The speed of the air velocity increases from 30.6 to 36.4 m/s.

REFERENCES

[1] Robert Paul. M, "Analysis and optimization of centrifugal blower CFD, ICMEET 2015, Issue 2, Volume 4, ISSN 2319-8753, PP.239-244, Published in IJIRSET.

[2] V. H. Chaudari, Varun Parmat "Parametric study and geometrical optimization of centrifugal blower" ICRISET2017, Volume 1, Page no 96-102 Kalpa publication.

[3] Ragoth Singh, R.Natraj M, (2012) "Optimising impeller geometry for performance enhancement", IJEST, Volume 4 No 10, ISSN 0975-5462.

[4] Karanth K.V and Sharma N.Y (2009) "CFD analysis on effect of Radial gap on impeller", IJRM, Volume 2, PP.1-8.

[5] N. Jayapragasan, K. Janardhan Reddy; "Design optimization and experimental study on the blower for fluffs collection system"; Journal of Engineering Science and Technology; Vol. 12, No. 5 (2017), PP. 1318 – 1336.

[6] Yogesh R. Pathak, Beena D. Baloni, Dr. S. A. Channiwala; "Numerical simulation of Centrifugal blower using CFX"; International Journal of Electronics, Communication & Soft Computing Science and Engineering; ISSN: 2277-9477, Apr-2012; PP. 242-247.

[7] Kusekar S.K., Lavnis A.K.; "Optimization of critical parts of centrifugal blower by Modal & CFD Analysis"; International Journal of Innovative Research in Advanced Engineering (IJIRAE) ISSN: 2349-2163; Volume 1 Issue 12 (December 2014); 2014, IJIRAE- PP. 69-82.

[8] V. Krishna, K. Naresh Kumar, M. Prasanth Kumar; "Numerical Analysis of Centrifugal Blower Using CFD"; International Journal of Engineering Research & Technology (IJERT); Vol. 2 Issue 8, August – 2013; ISSN: 2278-0181; PP. 1461-1464.

[9] Girish A. R., Prajwal Sandyal, Lokesh. K. S. Varun N.; "CFD simulation of centrifugal blower using ANSYS CFX"; National Conference on Advances in Mechanical Engineering Science (NCAMES-2016) ISSN: 2231-5381 http://www.ijettjournal.org PP. 34-38.

[10] G. V. R. Seshagiri rao, Dr. V. V. Subba rao; "Design of cooling fan for noise reduction using CFD"; International Journal of Scientific & Engineering Research (IJSER) Volume 2, Issue 9, September-2011; ISSN 2229-5518; PP.1-5.

[11] Seung Heo, Cheolung Cheong; "Development of low-noise centrifugal fans in a Refrigerator"; Proceedings of 20th International Congress on Acoustics, ICA 2010 23-27 August 2010, Sydney, Australia; PP. 1-5.