Theoretical, Experimental and Numerical Investigations of the Effect of Blades Number on the Performance of Regenerative Blowers

T. Mekhail, O. Dahab, M. Fathy, M. EL-Gendi, and H. Abdel-Mohsen

Abstract—Regenerative blowers are those devices which allow high heads with small flow rates. Although the regenerative blowers are widely used in many industrial applications, they have low efficiency compared to other turbo-machines. This research contributes in increasing the efficiency by studying the effect of blade numbers on the performance. Four blade numbers are studied 21, 31, 41 and 51 at a rotation speed of 3000 rpm and at different flow rates. A one-dimensional mathematical model based on the momentum theory is proposed. Steady three-dimensional CFD calculations are carried out using CFX-ANSYS 16.1 software. Experiments are carried out using straight blades. Pressures at inlet, outlet, and around the circumference of impeller are measured at different flow rates. The proposed mathematical model has a good agreement with the experimental results with accuracy ranging from 86 % to 99.7 %. The numerical CFD analysis showed that the areas of vortices and shock losses increase as the number of blades are decreased, which leads to the decrease of the blower efficiency. The increase of angle of attack (AOA) leads to the increase of the incidence loss (Shock losses) between the incoming air and the blade and in turn leads to the decrease of the blower efficiency. The experiment results showed that the pressure head depends strongly on the number of blades. The design and the best efficiency case occur for this blower at the maximum inlet flow rate of 56.73 × 10^{-3} kg/s. As a whole, the performance improves with increasing the blade number and 51 blades give the optimum efficiency.

Keywords—regenerative blower, blade number, CFD, momentum theory, AOA, Shock losses.

I. INTRODUCTION

Regenerative turbo-machines are classified as radial flow machines. They show performance curves with very stable features. Regenerative blowers are smaller and simpler to be constructed than the other equivalent volumetric blowers. In addition, they have lightweight design, less heat and noise, excellent performance range, long-life reliability, and oil-free air delivery. Although, it has fairly low efficiency [1]. Regenerative blowers have many vacuum/pressure applications such as bag packaging equipment, which can be used to open, hold, and close bags in filling machines; vacuum conveying, such as plastic pellets, grain, powder, and other bulk dry materials can be transported from one container to another easily using vacuum; dust and chip collection in which vacuum power is used to collect dust and/or chips from saws, mills, and other cutting machines; soil vapor extraction in which contaminated soil is often treated by pulling the contaminants, usually gasoline or other hydrocarbons, out of the soil with vacuum; automated product feeding in which vacuum force is used to automatically stack product as it is received or to move the product from one process to another and more others.

In order to improve the efficiency of regenerative machines, previous investigations have been carried out to determine the influence of geometrical parameters on their performance, e.g. shape and dimensions of the flow channel, angle of vanes and clearance between impeller and casing. Sixsmith and Altmann [2] introduced a new type of regenerative flow compressor with airfoil blades, which produced more head rise than any other shape of impeller blades. They assumed the deceleration of the flow in the channel as a diffusion process. Senoo [3] investigated experimentally the influences of various suction nozzles on the characteristic of a peripheral pump. He concluded that by using larger channel area in the inlet region, both higher head and efficiency can be achieved. In addition, large channel area can prevent cavitation. Choi et al. [4] carried out experimental work to investigate the influence of blade angle on the regenerative pump performance. They found that the head performance of the forward and backward blades decrease as the inclined blade angle increases. Some researchers studied the effect of different blade angles or blade shape on the performance of regenerative blower and pump by using CFD [1, 5].

Four theories found in literature, explain the behavior of regenerative machines namely: momentum exchange theory, shear stress theory, theory of airfoil blading, and compressible flow theory [6]. Senoo [7] compared the momentum exchange theory with the shear stress theory and concluded that the two approaches were compatible with the physics of the problem. In the following years, different researchers like Meakhail et al. [8] accredited the momentum exchange theory as the most correct. Most previous researchers used mathematical models based on exchange theory and adopt one inlet angle and one exit angle. Due to the variations of impeller geometry, Meakhail and Park [1] proposed another mathematical model for regenerative pump based on momentum exchange theory and adopt one inlet angle and two exit angles. This model showed some discrepancy with experimental data when applying it on regenerative blower.
In this paper, this mathematical model has been modified to become more convenient to be investigated experimentally and numerically by using CFX-ANSYS 16.1 software, and applied in case of using regenerative blower.

The effect of blade numbers on the regenerative pump performance was investigated by Iversen et al. [9], Badami [10] and Grabow [11]. Choon and Jong [12] studied the influence of blade number and extension angle on the regenerative blower performance. That blower was considered an open channel impeller type. They concluded that increasing of blade numbers leads to increase of the pressure and improves the efficiency.

The previous works didn’t include the study of effect of blades number by using CFD technique. Thus, the aim of the present study is to investigate the influence of blade number of regenerative blower impeller on its performance. In addition, a mathematical model for regenerative blower based on momentum exchange theory is proposed and adopt one inlet angle and two exit angles. CFD analysis is also carried out to explain current experimental results.

II. THEORETICAL MODELS

Further information about quantitative influence of geometrical parameters on the regenerative blower performance is obtained by formatting two one-dimensional models. The measured blower performance characteristics, expressed in dimensionless flow coefficient \( \varphi \) and the head coefficient \( \psi \), are defined by the following equations (1) and (2) Choi et al. [4].

\[
\psi = \frac{\Delta \varphi}{\Delta \varphi_{\text{crit}}} \tag{1}
\]

\[
\varphi = \frac{Q}{uA_c} \tag{2}
\]

Since the efficiency of the regenerative blower is defined as the ratio between the hydraulic power transferred to the working fluid and the power introduced into the system by the impeller (Fig. 1 shows a schematic diagram of regenerative blower), thus, the blower efficiency can be estimated by using Eq. [13]:

\[
\eta = \frac{\varphi_{\text{in}}}{\varphi_{\text{in}} + K_{\text{P}} \varphi^2} \tag{3}
\]

Where \( \varphi_{\text{in}} \) is real dimensionless flow rate coefficient [13].

* Badami and Mura [13] Mathematical Model (The First Model)

The impeller rotates with an angular speed \( \omega \), while the fluid motion inside the machine can be described by means of two components: a tangential component that determines the effective flow rate \( Q \) and a meridian component which determines the circulatory flow rate \( Q_m \). According to Badami and Mura [13], the working fluid enters into the vane grooves several times, thus augmenting its angular momentum due to the centrifugal field action; the fluid is then projected into the side channel, where its static pressure increases.

\[
\psi = 2 \frac{Q_m}{Ac} \left( \frac{r_{\text{2t}}}{r_{\text{1t}}} \frac{C_{\text{2t}} - C_{\text{1t}}}{C_{\text{1t}}} \right) - K_{\text{P}} \varphi^2 \tag{4}
\]

\[
\frac{u_{\text{2t}}}{u} = \frac{C_{\text{2t}}}{C_{\text{1t}}} \varphi, \quad \frac{u_{\text{ct}}}{u} = \frac{r_{\text{ct}}}{r_{\text{ct}}} \varphi \quad \text{and} \quad \frac{u_{\text{ct}}}{u} = \frac{r_{\text{ct}}}{r_{\text{ct}}} \left(1 - \frac{\Delta u_{\text{2t}}}{u} \right) + \frac{A_c}{A_{\text{ct}}} \left( \frac{Q_m}{Ac} \right) \cot \beta_{\text{2t}} \tag{5}
\]

Where \( u_{\text{ct}} \) is the tangential velocity of the fluid in the side channel [14] and \( C_{\text{1t}}, C_{\text{2t}} \) are the tangential components of the absolute velocities of the fluid at the impeller inlet/outlet .and \( u \) is rotational blade speed that can be determined by Eqs. (5).

The term \( \Delta u_{\text{2t}} \) is the slip factor which permits the deviation of the relative current with respect to theoretical one taken into account. In this work, the following formulation of the slip factor is used [13]:

![Fig. 1 Regenerative blower](image-url)
\[
\frac{\Delta P_P}{u} = \frac{1.5 + 1.1(2 - 2\beta_2^* / \pi)}{z \left(1 - \left(\frac{r_2}{r_1}\right)^2\right) + 1.5(2 - 2\beta_2^* / \pi)}
\]  
(6)

Where \(z\) is the number of vanes.

The circulatory flow rate can be determined by:

\[
\frac{1}{2} K_m + \frac{1}{\sin(\beta_2^*)} - 1 + \left(\frac{A_{21}}{A_1}\right)^2 \left(\frac{\cot(\beta_1^*)^2}{A_1} - \frac{Q_m}{A_1}\right)^2 + \left(\frac{r_2 - u_2}{u}\right) \cot(\beta_1^*) \left(\frac{Q_m}{A_1}\right) + \frac{1}{2} \left(\frac{r_1 - r_2}{r_2}\right)^2 + \left(\frac{u_2}{u}\right)^2 - \left(\frac{u_1}{u}\right)^2 = 0
\]  
(7)

The Proposed Mathematical Model (The Second Model):

Applying the angular momentum equation per one circulation to the fluid in the side channel that is subjected to torque transmitted by the impeller, the expression for the head rise of the blower per one circulation, \(H\), can be written as; Meakhail and Park [1]:

\[
H = n h_{circ} - \frac{1}{2} K_p \left(\frac{Q_m}{A_1}\right)^2
\]  
(8)

The total head of the blower and the head coefficient can be expressed as

\[
H = nh_{circ} - \frac{1}{2} K_p \left(\frac{Q_m}{A_1}\right)^2
\]  
(9)

\(K_p\) is the loss coefficient related to the friction forces on the side channel wall and is assumed to be equal to 0.01[1].

Where the number of circulation \(n\) can be simply calculated from

\[
n = \frac{\theta_{eff}}{360^* z}
\]  
(10)

Where \(\theta_{eff}\) is the effective angle of the blower from the inlet port to the outlet port.

Eq.(8) is the same equation obtained by Wilson et al. [15] and Badami [10] but it differs in that it contains the term \((A_2/C_{m2f2})\), which represents the momentum of the fluid leaving at the side of the impeller blade as well as the number of circulation. This term is a function of the side exit angle of the impeller since \(C_{u2s}\), is given by:

\[
C_{u2s} = \sigma_s (u_{2s} - C_{m2s}(\cot(180 - \beta_{2s})))
\]  
(11)

\[
C_{u2st} = \sigma_t (u_{2st} - C_{m2st}(\cot(180 - \beta_{2st})))
\]  
(12)

Where \(\beta_{2s}\) and \(\beta_{2st}\) are the blade exit angles at the blade side and blade tip, respectively as in Fig. 2. And the slip factor at the tip and side is calculated as:

\[
\sigma_s = \frac{1}{1 + 2.5(1 + \sin(\beta_2)^*)}
\]  
(13)

\[
\sigma_t = \frac{1}{1 + 2.5(1 + \sin(\beta_2)^*)/2}
\]  
(14)

Under the hypothesis of constant angular speed of the fluid in the side channel, the kinetic ratio \(\frac{C_{u2}}{u}\) can be calculated by means of the following Eq. [13]:

\[
\frac{C_{u2}}{u} = \frac{r_2}{r_c} \phi
\]  
(15)

The circulatory velocity at the tip and side \((C_{m2t} and C_{m2s})\) calculated as:

\[
\frac{\xi_1}{2 \sin(\beta_2)^*} (C_{m2t})^2 + C_{m2t} \sigma_t \left(\frac{u_{2t} + u_1}{u_1} - \frac{r_2}{r_1}\right) = 0
\]  
(16)

\[
C_{m2s} \sigma_s (u_{2s}(-\cot(180 - \beta_{2s})) + \xi_2 \left(\frac{Q_m}{A_1}\right)^2 + 2u_1 \frac{r_1}{r_1} - \sigma_s \left(\frac{u_{2s}}{u_{2s}}\right) = 0
\]  
(17)

Where \(\xi_1\) is the loss coefficient and is evaluated experimentally [10] and ranged between 0.0042 and 0.0063 for all impeller blades. \(\xi_2\) is the channel skin friction loss and calculated as in [1] and it is equal to 0.011 for the rotational speed of 3000 rpm.

The inlet circulatory flow rate, \(Q_{m1}\) can be calculated by continuity equation as:-

\[
Q_{m1} = Q_{m2s} + Q_{m2t}
\]  
(18)

### III. CFD METHODOLOGY

A CFD simulation has been carried out by using ANSYS 16.1 software in order to increase the knowledge of the flow field inside the regenerative blower and to analyze the velocity vectors around the impeller. The steady state conditions are considered in the simulation and a compressible air ideal gas with constant specific heat is assumed. The standard k-\(\varepsilon\) model has been tailored specifically for recirculating flows Lettieri et al. [16], so turbulence is modeled using the standard k-\(\varepsilon\) turbulence model Fan et al. [17], the energy equation is solved additionally and all the walls are considered as adiabatic and smooth. The mass flow rate normal to the boundary is set at the inlet duct as boundary conditions, while the total pressure and temperature are imposed at the outlet.

Some of the main characteristics of the fluid at the boundary conditions are summarized in Table 1.
Grid Independence Check

In this section four runs of the same problem, same boundary conditions and different grids are done for all tested impellers (21, 31, 41 and 51) as shown in Table 2. An investigation of grid independence is carried out to check for the proper mesh.

For number of blades 21, as an example, four different grids are checked (498342 cells, 520269 cells, 546551 cells, and 922615 cells). The maximum value of the pressure difference between inlet and outlet occurs at grid no.4 (922615 cells) which is greater than the value of the designed pressure difference at grid no.3 (546551 cells) by a small difference of 6.2 %. This indicates that both of the two grids are approaching grid independence. All results in the following discussions are done for grid No.3 case.

IV. EXPERIMENTAL TEST RIG

In the present experimental study, a regenerative blower system with impellers of different number of blades (21, 31, 41 and 51) is designed and manufactured as shown in Figs. 3 and 4. This is selected for two reasons, firstly, the maximum manufacturing limit (by using CNC 3D Machine) that is produced by the impeller of maximum number of blades, is 51. Secondly, 10 blades reduction rate of number of blades between each impeller is chosen to get a remarkable effect on the blower performance. Fig. 2 shows the scheme of a regenerative blower that consists of an impeller with radial blade (c) and a casing (j) with bearing (k) where the side channel (d) is machined. There is a stripper block (b) to separate the inlet port (a) and outlet ports (e) that’s connected by

<table>
<thead>
<tr>
<th>Table 1</th>
<th>The main characteristics of the fluid at the boundary conditions.</th>
</tr>
</thead>
<tbody>
<tr>
<td>No.</td>
<td>No of blades</td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
</tr>
<tr>
<td>1</td>
<td>21</td>
</tr>
<tr>
<td>2</td>
<td>31</td>
</tr>
<tr>
<td>3</td>
<td>41</td>
</tr>
<tr>
<td>4</td>
<td>51</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Table 2</th>
<th>Global mesh statistics for different number of blades</th>
</tr>
</thead>
<tbody>
<tr>
<td>Global Mesh Statistics</td>
<td>21 blades</td>
</tr>
<tr>
<td>Global Number of Nodes</td>
<td>546551</td>
</tr>
<tr>
<td>Global Number of Elements</td>
<td>2221636</td>
</tr>
</tbody>
</table>
the air flow rate control valve (f) and calibrated air velocity measuring instrument (g). This blower is driven by 1 hp. 3 phase induction motor (h) in which the motor rotor is coupled with the blower rotor by coupling (i). The impellers have blades of diameter 300 mm, height 50 mm, width 23 mm and thickness 3 mm.

V. EXPERIMENTAL PROCEDURE

The experimental work is carried out at a rotational speed of 3000 rpm and at eight different outlet flow rates which is expressed as dimensionless flow coefficient $\varphi$ and varied from 0 to 0.7. The blower rotational speed is measured by digital tachometer, and controlled by changing the feeding electrical frequency to the induction motor that drives the blower by using adjustable frequency driver. The air flow rate is controlled by using the valve (f) in Fig. 4 and by using a calibrated Thermo-Anometer air meter unit with vane probe (g). It is adopted to be connected to a computer for accessing data. In order to evaluate the performance of the regenerative blower, the pressure at inlet, outlet, and around the circumference of impeller are measured at different flow rates by using a calibrated digital manometer, to calculate the dimensionless head coefficient $\psi$.

The pressure was measured by a digital manometer with accuracy of ±0.25% of full scale, including the combined effects of temperature, linearity, repeatability, hysteresis and resolution. The accuracy of the Thermo-Anometer air velocity meter unit is ±(3% + 1 digit). The rotational velocity is monitored using a digital tachometer whose accuracy falls within ±(0.5% + 1 digit). The scattered data are evaluated from repeatability tests and sensitivity analyses.

VI. RESULTS AND DISCUSSION

The results are subdivided into three parts, the first part; presents the experimental results of different number of
blades; the second part presents CFD results, and finally, comparison experiments with the first 1-D model, second 1-D model, and CFD results are presented.

Experimental Results
The measured experimental characteristic curves for the four impellers with numbers of blades 51, 41, 31, and 21 are shown in Fig. 5. In most cases, the maximum flow coefficient in the present experiments was about 0.7. Fig. 5 declares that, the pressures increases as the flow rates decrease. Owing to the instability of the pressures and velocities inside the blower for each flow rate, lines has been drawn connecting the average values of the head coefficients and efficiencies. The minimum head for the blower performance occurs at $\phi = 0.1$ for 21 blades number and increases gradually with increasing blades number to reach the maximum value of 51. The slope for the performance curve increases slightly with the increase of flow coefficient for the impeller of 41 blades number and decreases for the impeller of 21. The performance curves lines for the impellers of blades numbers 31 and 51 are parallel. At $\phi = 0.1$ the efficiencies for the blades number (21, 31, 41, and 51) are equal. With the increase of flow coefficient, efficiency increases gradually ascendingly as the blades number increases (Fig. 6). The compression increases as the number of blades increases and leads to the improvement of the regenerative blower performance and efficiency, so the impeller of 51 blades has a better efficiency than the other impellers (21, 31 and 41).

The efficiency is generally low at lower flow rates and increases as the flow rate through the blower increases [18], thus in order to obtain good efficiency; the blower should be operated at a higher flow rate. The reason for low efficiency at a low flow rate is that the circulatory power is greater because of the increased number of circulations at the low flow rates, which is a cause of increasing circulatory head loss and increasing shock loss. As the flow rate increases through the blower, there will be fewer vortices circulations which means better efficiency [18], and as the number of blades increases, the number of flow circulation in the side channel increases. This leads to the increase of the transferred energy from impeller blade to the flow inside the channel and leads in turn to the improvement of the blower performance and efficiency. Fig. 6 shows the relation between the number of blades and maximum efficiencies at the maximum flow rate ($\phi = 0.7$). The efficiencies differences between each blades number appear gradually and increase as the flow coefficient increases, to reach the maximum values at $\phi = 0.7$. At $\phi = 0.7$ the efficiency is improved by 0.8, 1.1, and 1.64 % as the blades numbers were increased from 21 to 31, from 31 to 41, and from 41 to 51 respectively. The maximum efficiencies were found to be 46.33, 44.68, 43.59, and 42.97 % for impellers of blades numbers 21, 31, 41, and 51 respectively.

![Fig. 5 Comparison of characteristic curves with different number of blades](image-url)
CFD Results

*Velocity Field*

From the CFD analysis, the flow velocity between any two blades can be deduced and analyzed. Fig. 7 shows the velocity vectors at the middle section between two blades of the blower impeller. This approves that the second theoretical [1] one-dimensional model is more compatible to the experimental results than the first model. The regenerative blower operation is affected by different types of losses [6]. Thus, when designing the regenerative blower, it is important to take into account how to minimize the losses as much as possible in order to improve its efficiency [19]. So, these losses and their impact on the performance of each fan should be well studied. A shock loss is one of these losses and appears usually at the blade inlet. The increase of Angle of attack (AOA) which is the angle between the incoming air and blade angle leads to the increase of the incidence loss (Shock losses) between the incoming air and the blade; and in turn leads to the decrease of the blower efficiency. Fig. 8 represents the velocity distribution of the entering flow (at the plan A-A in Fig. 7) to the impeller for different blades number and shows that the area of vortices increases as the number of blades is decreased which causes the increase in the energy loss between the blades, thus, resulting in a decrease of the blower efficiency. These results agree very well with that obtained experimentally.

*Pressure Evolution*

One of the factors affecting the pressure in the regenerative blower is the increase of blades number. Fig. 9 shows the pressure distribution for the regenerative blower at constant flow rates of $8.1 \times 10^{-3}$ kg/s and different blades number. It can be observed that the average pressure rises with increasing the number of blades and this increase in pressure is shown at the exit port in an area of pressure ranging of 2.1 kPa to 2.61 kPa. This area increases with the increase in pressure as the number of blades increases to reach another pressure range at an area ranging from 2.61 kPa to 3.13 kPa, that appears in case of using number of blades 41 and 51. It can also be noted that the rate of decline in pressure increases at the inlet port with decreasing the number of blades, as shown by the area of pressure which ranges between $45.8 \times 10^{-3}$ kPa and 0.468 kPa, and increased gradually as the number of blades is decreased to reach an area of pressure that is less than -2 kPa in case of number of blades 21.

The pressure variation of air circulates through the blower in several flow rates (expressed as the flow dimensionless coefficient) for the impeller of 51 blades and is shown in Fig. 10. The curves obtained suggest four regions in the blower operation, which are discussed below as follows.

i. Inlet region: The flow experiences some pressure loss through the inlet region.

ii. Compressing region: The flow enters the working section of blower with a velocity and pressure dependent largely on the inlet region. Until the flow reaches fully developed pattern of linear region, the circulatory velocity will change. A deceleration occurs and the kinetic energy of the circulatory velocity is changed as the pressure rises.

iii. Outlet region: A loss similar to that at the inlet region occurs at the outlet region.

iv. Stripper region: Between the discharge and the inlet, the casing clearance is reduced to block the high-pressure discharge from the low-pressure inlet. The stripper forces the air to go out through the discharge port. These regions are described in detail in ref. [6]. Compared to the previous studies, the pressure distribution has the same trend as in ref [6, 15].
Comparing the Experiment with 1st 1-D model, 2nd 1-D model, and CFD Results

The verification of the experimental results accompanied with numerical results obtained from both the first and second one-dimensional models, and CFD results has been carried out through by quantitative comparison. From Figs. 11 and 12, the comparison of performances and efficiencies between the experimental results, first and second one-dimensional model and CFD results for different number of blades can be performed. The second theoretical one-dimensional model is more compatible and closer to the experimental results because it depends on the flow entering the impeller from the side of the blade and part of it leaves the impeller from the blade side and another part leaves from the blade tip, as in Fig. 7. The CFD analysis shows that there are some differences between CFD results and the experimental results, as shown in Figs. 11 and 12. The reasons for these
differences are due to the gaps leakage that have only partially taken into account on the CFD calculation; this leakage is proportional to the pressure difference between inlet and outlet [20]. A second reason for the difference between CFD and experimental results is the flow inside the regenerative blower that is already unsteady and the CFD simulations are computed as steady state flow.

Fig. 9 Pressure along the side channel of regenerative blower at constant flow rate (8.1E-3 kg/s) for different number of blades

Fig. 10 Pressure distribution in the regenerative blower at various flow coefficients for the impeller of 51 blades
Thirdly, the standard k-ε turbulence model, which is used in the CFD simulation caused some deviations in case of forced flow, which appear clearly in Fig. 11 for the impellers of 21 and 31 blades. Finally, the blower walls are considered smooth in the CFD simulation, however, different unknown surface roughness exist due to some defects in manufacture.

Considering the differences mentioned previously, Fig. 11 indicates that the difference raised to about 0.66 at ϕ = 0.1 for blades number 21, and decreases to 0.2 as ϕ increases to 0.5. The difference drops for blade number 31 to about 0.6 at ϕ = 0.1 and becomes closer to the experimental results to be 0.23 at ϕ = 0.7 and 0.36 at ϕ = 0.1, for blades number 41 and 51 respectively. Whereas, Fig. 12 indicates that the efficiencies obtained from the CFD analysis exceed the experimental efficiencies by a semi-constant value in a range between 4.95 and 6.24 % for all blades numbers.

An exception occurs in case of blades numbers 21 and 31 at ϕ equal 0.5 and 0.7 to reach a maximum of 10.26 % at the blades number 31 and ϕ = 0.5.

The proposed one-dimensional model, shown in Fig. 11, and the theoretical and CFD efficiencies shown in Fig. 12 are all converged to the experimental efficiencies in most cases. At the flow rate of $32.42 \times 10^{-3}$ kg/s ($\phi = 0.4$) the second theoretical one-dimensional model performance match the experimental performance for different number of blades while at the flow rate of $40.52 \times 10^{-3}$ kg/s ($\phi = 0.5$) the first theoretical 1-D model performance match the experimental performance for different number of blades. The design and the best efficiency case occurred for this blower at the flow rate ranging between $48.62 \times 10^{-3}$ kg/s and $56.73 \times 10^{-3}$ kg/s (at $\phi$ between 0.6 and 0.7) and with different numbers of blades.
VII. CONCLUSION

The aim of this research is to study the effect of blade number on the performance of regenerative blower by three different methods, experimentally, numerically and theoretically. A one-dimensional mathematical model is proposed to be more convenient to the experimental results for the designed blower. Consequently, some concluded points are summarized as follows:

1. The CFD 3D analysis showed that, as the blade number decreases, the AOA increases, which leads to the increase of vortices, circulation and shock losses and consequently decreases the blower efficiency.

2. The proposed theoretical one-dimensional model is more compatible and closer to the experimental results other than Badami and Mura [13] mathematical model.

3. The impeller of blade number of 51 has the best efficiency among the rest of impellers. As the blade number is increased the transmitted energy from the impeller to the working fluid increases, leading to increasing the blower head and consequently increases the blower efficiency.

In the future work the performance of regenerative blower should be studied by using high accuracy transient measuring instruments that are improved to measure the vortices and flow circulation inside the regenerative blower and comparing these measurements with that obtained from the CFD simulations.

VIII. REFERENCES

[6] M. Raheel, and A. Engeda, Current status, design and performance trends for the regenerative flow...


Notation

A  cross sectional area
C  absolute velocity
h  head loss
H  head
n  number of circulations (number of times the fluid passes through the impeller blades)
N  rotational speed (rpm)
p  pressure
K_p  loss coefficient related to the friction forces on the side channel wall
K_m  loss coefficient that takes into account all the losses due to enlargements or restrictions of the flow and to turbulence share forces in the vanes related to the circulatory flow rate.
Q  blower flow rate
r  radius
u  peripheral velocity
z  number of impeller blades
Δp  increase in pressure in the side channel
φ  dimensionless flow coefficient
ψ  dimensionless head coefficient
ρ  density
η  efficiency
ω  angular speed
β'  blade angle (the theoretical angle of the flow)
β  flow angle (the actual angle of the flow)
σ  slip factor
θ_{eff}  effective angle from the inlet to the outlet of the blower

Subscripts

l  position of inlet flow to the impeller blade
2s  position of outlet flow from the impeller side
2t  position of outlet flow from the impeller tip
C  channel
cir  circulation
m  meridional (circulatory) component, mean value
s  side
t  tip
u  tangential component