Numerical analysis of a non-contact surface adhesion system based on vortex cup for wall climbing robots

A. Fallah\textsuperscript{a}, M. Rezasoltani\textsuperscript{a}, A. Riasi\textsuperscript{a} and H. Moradi\textsuperscript{b,\textdagger}

\textsuperscript{a} School of Mechanical Engineering, College of Engineering, University of Tehran, Tehran, P.O. Box 1155-4563, Iran.
\textsuperscript{b} Control and Intelligent Processing Center, School of Electrical and Computer Engineering, College of Engineering, University of Tehran, Tehran, P.O. Box 1155-4563, Iran.

Received 5 May 2014; received in revised form 22 September 2014; accepted 20 October 2014

**KEYWORDS**

Robot; Vortex cup; Wall climbing; CFD; Non-contact adhesion.

**Abstract.** In this work, an adhesion system for wall climbing robots using vortex cup has been designed and numerically simulated. A small scale model of the system has been designed and the flow patterns including pressure and velocity fields are computed using CFD analysis. The results are verified using mesh independency and validated through comparison with the available experimental data and show to have high correlation together. Then a large scale vortex cup based on the actual weight of a wall climbing robot has been designed and simulated. Furthermore, the effects of different parameters such as the number of nozzles, cup height and cup radius on the adhesion force have been studied. Finally, a new cup has been designed base on the optimum data obtained from the numerical results.

\textcopyright 2015 Sharif University of Technology. All rights reserved.

\textdagger Corresponding author. Tel: +98 21 82084018; E-mail addresses: ali.fallah@ut.ac.ir (A. Fallah);
m_rezasoltani@ut.ac.ir (M. Rezasoltani); arias@ut.ac.ir (A. Riasi); moradih@ut.ac.ir (H. Moradi)

1. Introduction

In recent decades, the interest for wall climbing robots has grown rapidly. Wall climbing robots with maneuverability on vertical surfaces are required in different industries for operations such as maintenance, inspections, reconnaissance, aboveground or underground petrochemical storage tank inspection, and assistance in firefighting and rescue operations. Consequently, this kind of robots can be used to elevate the performance in such dangerous environments with difficult access. Furthermore, these robots can eliminate the need for human operators in high risk operations [1].

Surface adhesion is the basic approach in wall climbing robots. The adhesion force should overcome the gravitational force due to a robot’s weight. Surface adhesion techniques are classified into various groups based on the nature of the adhesion forces: (1) Pneumatic, (2) Magnetic, (3) Mechanical, (4) Chemical, and (5) Electrostatic. Pneumatic technique is probably the most popular method for adhesion forces in wall climbing robots [2].

Pneumatic method is subdivided in three categories: (a) Vacuum cups, (b) Vacuum chamber, and (c) Vortex cup. In the first method, pneumatic force is produced by vacuum cups. This method is simple and inexpensive (albeit the system creating vacuum may become big and expensive), but they have slow movement because they need time to provide vacuum in cups, and cannot operate on rough surfaces [2-4]. Moreover, since the vacuum cup relies on suctioning the air, delicate surfaces, such as tiles on domes, may not be suitable for it. Finally, dealing with the dirt that is sucked into the system is a major problem.

The vacuum chamber technique uses a big chamber to create relative pressure difference between the
border of the chamber and the outside air. This pressure difference pushes the chamber toward the surface. Unlike the vacuum cup, vacuum chamber does not need to be fully sealed. A robot equipped with a vacuum chamber adhesion system can operate on different surfaces and can overcome small obstacles on walls. Furthermore, this robot is faster than a robot using vacuum cups. However, due to incomplete sealed condition, the vacuum chamber technique has higher energy consumption compared to the vacuum cups \[1,3\].

Vortex cup is a noncontact method that uses vortex phenomenon to generate a negative pressure at the center of the cup, producing a force toward the contact surface. One of the latest robots that uses this phenomenon is Alica VTX. This robot uses a centrifugal fan at the center of the cup to produce suction by creating vortex [4]. Vortex base adhesion system is also used in applications where noncontact levitation is very important such as optical disks, wafer, circuit wafer, LCD panel, and semiconductor manufacturing.

As can be seen in Figure 1, the cup is made up of a circular cylinder and a tangential nozzle inserted on top. A fillet is cut at the bottom to conduct the air out of the cup. Compressed air is blown through the nozzle into the cup and spins along the circular wall to create a negative pressure in the central area by centrifugal force. This negative pressure causes a lifting force applied to a work piece placed under the cup, and holds it in an equilibrium position. Since the air is continuously supplied, the cup will keep levitating with a gap of hundred micrometers from the work piece, through which air can be discharged into the atmosphere. For this reason, the work piece never contacts the cup. One more important fact is that the negative pressure inside the cup depends on the gap between the cup and the work piece. In a considerable narrow region below the vortex cup, the lifting force decreases if the work piece deviates from the equilibrium position to approach the cup, and increases if the work piece leaves the cup from the equilibrium position due to the weight of the work piece. As a result, the work piece can levitate stably at the equilibrium position [5].

In this paper, we propose two modifications to this new noncontact surface adhesion to increase its performance. The proposed modification includes equipping the vortex cup with a centrifugal fan instead of compressed air tank used in the experimental model of Li et al. [5], and also changing the nozzle configuration. To verify the effectiveness of the proposed modifications, a scaled up model based on the real weight of a robot has been designed and simulated.

2. Numerical simulation

This section includes the modeling of a vortex cup, mesh generation, turbulence modeling, code verification and validation.

2.1. Modeling of the vortex cup

The cup geometry and inlet conditions are taken from the experimental work of Li et al. [5-7]. These specifications are shown in Figure 1 and listed in Table 1. Two different models have been tested in this paper. It is worth mentioning that the models are scaled down considering the experimental limitation.

2.2. Mesh generation

For meshing the models described above, tetrahedral mesh is used near the walls and hexahedral mesh is generated in regions with small pressure gradient. A size function has been employed in order to lower the number of mesh elements. Size function starts from a small mesh size and increases with a rate of growth. This function is a desirable strategy to obtain refined mesh in regions where large pressure gradients are expected such as a boundary layer. Because of the dramatic flow changes in gap regions and nozzle inlet, a smaller mesh size is used in these areas to obtain more reasonable results.

![Figure 1. Mechanism of vortex cup (the air flow is shown by using the arrows).](image)

<table>
<thead>
<tr>
<th>Item</th>
<th>1st model</th>
<th>2nd model</th>
</tr>
</thead>
<tbody>
<tr>
<td>(H_1)</td>
<td>8.5</td>
<td>8.5</td>
</tr>
<tr>
<td>(H_2)</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>(H_3)</td>
<td>2</td>
<td>2</td>
</tr>
<tr>
<td>(R_1)</td>
<td>11.5</td>
<td>8.5</td>
</tr>
<tr>
<td>(R_2)</td>
<td>20</td>
<td>9</td>
</tr>
<tr>
<td>(R_3)</td>
<td>9</td>
<td>6.5</td>
</tr>
<tr>
<td>(d) (nozzle diameter)</td>
<td>1.5</td>
<td>1</td>
</tr>
</tbody>
</table>
2.3. Governing equation

The governing equations of incompressible flow including continuity and momentum equations are as follows:

\[ \nabla \cdot V = 0, \]

\[ \frac{DV}{Dt} = -\frac{1}{\rho} \nabla P + g + \frac{1}{\rho} \nabla \tau_{ij}. \]

These equations are solved by using Fluent, commercial software which is based on the finite-volume method for three dimensional flow fields. The discretization schemes used in this work are: Simple algorithm for pressure-velocity coupling, and second order accurate upwind scheme for momentum and turbulence equations.

2.4. Turbulence modeling

For turbulence modeling both Reynolds Stress Model (RSM) and \( k-\varepsilon \) can be used. Among different kinds of \( k-\varepsilon \) models the RNG \( k-\varepsilon \) is used because of its robustness. Because of high swirling in the cup, the flow field is anisotropic. So RSM method is the best and the most precise turbulent model for this case. Also results from previous works on cyclone, which has somehow similar swirling flow, show that RSM is the best model for the high swirling cases [8]. For boundary layer effects a semi-empirical formula, called wall function, is used to bridge the viscosity-affected region between the wall and the fully-turbulent region.

2.5. Assumptions and boundary conditions

To capture the actual physical phenomena, the following boundary conditions are taken into consideration to simulate the flow field inside the vortex cup:

- Ideal gas is adopted as a fluid;
- No slip boundary condition is utilized for the cup walls;
- Mass flow inlet is applied at the nozzle outlet;
- Boundary condition at the cup outlet is the pressure which is set to ambient pressure.

2.6. Mesh independency

In order to investigate mesh independency, the vortex cup is simulated using mesh with four different number of elements, and the suction force is computed for each. The results show that a system with 226745 grid nodes is acceptable for this work and more elements are not necessary.

2.7. Code validation

To validate the numerical model developed in this paper, the computed data are compared with the measured data of Li et al. [5-7], as shown in Figures 2 and 3. The gap clearance and the flow rate in the first model (Figure 2) are assumed to be 0.45 mm and \( 15.7 \times 10^{-5} \) m³/s, respectively, and for the second model (Figure 3) they are 0.65 mm and \( 20.8 \times 10^{-5} \) m³/s, respectively. The numerical results have a good correlation with the experimental data and also RSM model provides reasonable results.

3. New design using centrifugal fan

In the original work of Li et al. [5-7], the air flow is supplied by using a compressed air tank. This method is very expensive and causes several limitations for climbing robot’s movement and maneuverability. To solve this problem, the air tank has been replaced by an on-board centrifugal fan. For this purpose a centrifugal fan with forward curved blades has been designed. This type of fan provides greater total pressure in comparison to backward curved blades at the same size and rotational speed. In this design, the blade outlet angle is 147 degrees and the rotational speed is 5000 rpm. Figure 4 shows the meshed geometry of the impeller.

Dynamic pressure of the nozzle inlet in the previ-
ous work is about 50 kPa. Since this dynamic pressure is not achieved with a small centrifugal fan, the nozzle configuration is changed. To this end, the second model of Li et al. [5], in which the gap clearance is 0.65 mm, has been modified as follows.

**Design 1:** Nozzle diameter is increased at a constant flow rate, and as a consequent the nozzle outlet velocity decreases. In this case, the inlet dynamic pressure is about 3.8 kPa which is much smaller than that in the second model of Li et al. [5].

**Design 2:** Inlet dynamic pressure is the same as that in Design 1, but the flow rate is doubled by adding a nozzle to Design 1.

**Design 3:** The total flow rate of the two nozzles is set to be the same as that in Design 1. This means that the inlet velocity is half compared to the first and second designs.

The pressure distribution of these three designs is shown in Figure 5. With respect to Li et al. [5] report, the flow rate through the cup is a function of gap clearance, and it is independent of the gap in case that the gap thickness is more than 0.1 mm. In these designs the gap clearances were assumed to be greater than 0.1 mm. By analyzing the computed results of Designs 1, 2 and 3, it can be concluded that the generated negative pressure increases with the increase of the inlet flow rate at the same inlet velocity (the same inlet dynamic pressure). Also results show that by increasing the inlet velocity (inlet dynamic pressure), at the same flow rate, the generated negative pressure increases. Therefore, both the inlet flow rate and the inlet velocity are important in the creation of negative pressure. The pressure contours for Design 2 are shown in Figure 6.

**Design 4:** In this case, four symmetric nozzles with the same inlet conditions of Design 1 are used. Figure 7 represents the comparison of the fourth design with the experimental data of Li et al. [5]. It shows that the maximum negative pressure of Design 4 is increased by 20%.

4. **Parametric study**

In this section, a large scale version of Design 4, based on the real weight of a climbing robot has been...
Table 2. Specifications of large scale vortex cup (dimensions are in cm).

<table>
<thead>
<tr>
<th>$H_1$</th>
<th>$H_2$</th>
<th>$H_3$</th>
<th>$R_1$</th>
<th>$R_2$</th>
<th>$R_3$</th>
<th>$d$</th>
</tr>
</thead>
<tbody>
<tr>
<td>8.5</td>
<td>2</td>
<td>0</td>
<td>9</td>
<td>10</td>
<td>6.5</td>
<td>1.6</td>
</tr>
</tbody>
</table>

Figure 8. The axial velocity at $H_1/2$; $Q = 16.08 \times 10^{-3}$ (m$^3$/s).

Figure 9. The tangential velocity at $H_1/2$; $Q = 16.08 \times 10^{-3}$ (m$^3$/s).

4.1. The effect of clearance
The inlet dynamic pressure of 3.8 kPa is assumed for the numerical simulation. Figures 8 and 9 show axial and tangential velocity at $H_1/2$. The gap clearance is 1 cm and the flow rate is $16.08 \times 10^{-3}$ m$^3$/s. Figure 10 shows the generated force in different gap clearances. As expected, the generated force takes its maximum value at a certain gap clearance and further increase in the gap clearance reduces the generated force.

4.2. The effect of the cup height
Figure 11 depicts the result of the generated force versus mass flow rate for different $H_1$ (clearance in all cases is 1 cm). As shown in this figure, the suction force at the constant mass flow rate increases as $H_1$ decreases.

4.3. The effect of the cup radius
With the assumption of $H_1 = 2$ cm and $H_2 = 0$, the effect of different $R_3$ (the cup radius) is computed.

Figure 12 shows that the cup with $R_3 \approx 7.5$ cm provides the maximum suction force.

Finally, the modified vortex cup is designed based on the best values of the geometric parameters. Figure 13 shows the comparison of generated force between the modified cup and its initial configuration.

5. Conclusion
In this paper, a new vortex-based surface adhesion system is designed and numerically simulated. The new
Figure 13. The comparison between the modified and the first design of large scale vortex cup at different gap clearances.

design shows the following improved features:

- The adhesion force is increased by increasing the inlet flow rate at a constant velocity;
- By increasing the inlet velocity, at a constant inlet flow rate, the adhesion force increases;
- The reduction of the cup height results in the force increase.

The future work would focus on implementing this design to test vortex cup in a lab environment and also put it on our drone inspection robot to test the design on an actual robot.

References


Biographies

Ali Fallah received his MSc degree in Mechanical Engineering, Energy Conversion, from the University of Tehran, in 2012. His main research interest is design and simulation of wall climbing robots adhesion system. He is currently working in the turbomachinery industry as senior engineer.

Masoud Rezasoltani received his MSc degree in Mechanical Engineering, Energy Conversion, from the University of Tehran, in 2013. His main research interest is design and simulation of suction cup vortex attractor used in wall climbing robots. He is currently working in the turbomachinery industry as senior engineer.

Alireza Riasi received his BSc degree in Mechanical Engineering from Tehran Polytechnic, in 2002, and his MSc and PhD degrees in Mechanical Engineering, Energy Conversion, from the University of Tehran, in 2004 and 2009, respectively. He has worked for the hydropower industry as senior engineer for the supervision on supply and locally manufactured turbine components. He is currently Assistant Professor and also Head of Fluid Mechanics Laboratory at the School of Mechanical Engineering, University of Tehran, Iran. His main research interests include turbomachinery in renewable energy applications (water, wind, ocean and tidal), transient flows, fluid mechanics and Computational Fluid Dynamics (CFD) and hydroelectric power plants.

Hadi Moradi is an Assistant Professor at the Electrical and Computer Engineering Department of the University of Tehran. He is also a member of Control and Intelligent, Processing Center of Excellence. He received his BSc in Electrical Engineering from the University of Tehran, and his MS and PhD degrees in Computer Engineering (Robotics), respectively, in 1994 and 1999, from the University of Southern California. Before joining the Tehran University in 2008, he spent several years in industry as a research
scientist. He joined USC at 2000 as a faculty at the Computer Science Department. In 2004, he spent two years at the Intelligent Systems Research Center at Sungkyunkwan University, in South Korea till 2006. His current research interests include motion planning, service robotics, educational robotics, and intelligent tutoring system, game-based teaching/evaluation/rehabilitation. He was a co-chair of Service Robots Technical Committee at Robotics and Automation Society, IEEE, from 2006 till 2013.
كارگاه‌های آموزشی مرکز اطلاعات علمی جهاد دانشگاهی

کارگاه آنلاین بروزار نویسی و یادگیری نویسی

کارگاه آنلاین بررسی مقابله ای جشنون (مقدماتی)