Modeling and optimization of suction fan for pneumatic conveyor for chickpea seed

Mostafa kolivand
M.Sc in Department of Agricultural Machinery Engineering. Faculty of Agriculture. University of Kurdistan. Sanandaj, Iran
Mostafakolivand98@yahoo.com

Hiwa golpira
Ph.d in Department of Agricultural Machinery Engineering. Faculty of Agriculture. University of Kurdistan. Sanandaj, Iran
h.golpira@uok.ac.ir

Abstract

In this paper computational fluid dynamics (CFD) and measurements using hot-wire anemometers were used to study the flow configuration and performance of a suction fan (SF) used for pneumatic conveyor for chickpea seed. A general two-dimensional simulation of turbulent fluid flow is presented to predict velocity and pressure fields for a suction fan. A commercial CFD code was used to solve the governing equations of the flow field. In order to study the most suitable turbulence model, three known turbulence models of standard K–ε, RNG and RSM were applied. Simulation results in the form of characteristic curves were compared with available experimental data, and an acceptable agreement was obtained. Additionally, special attention was paid on the effect of location of inlet on the efficiency of fan was studied. It was demonstrated that two-dimensional CFD model can predict fan performance up to an acceptable level. Moreover, it was shown in general that the location of inlet plays a crucial role for the performance of the SF. The location of inlet was changed in four section (top, below, left & right). The results show that the position of inlet in below section has the highest velocity inlet. Investigations of this kind can help to reduce the required experimental work for the development and design of such devices.

Keywords: suction fan, chickpea, CFD.

Introduction

Chickpea (*Cicer arietinum* L.) as important source of protein (18-23%) is a grain legume widely grown in Iran. According to Tabatabaeefar et al (2003) approximately 70-80% of chickpea production is used for human consumption, 14% for animal feed, 5-10% for seed, and 2-7% is lost during harvesting and processing. Iran is the 7th largest chickpea producing country and Kurdistan with 14.1% is the 4th largest chickpea producing in Iran.

At the present, harvesting and handling of the chickpea are carried out manually. This procedure is labor-intensive, time-consuming and also with difficulty and low efficiency. Therefore, mechanization of chickpea harvesting is objective of farmers in Iran.

In recent years, many efforts have been made to design and build a chickpea harvesting machines. Behroozilar and Huang (2002) designed and developed a stripper type chickpea combine harvester but Tado et al. (1998) expressed that this type of head for grain products with low yield is not suitable and reported that the losses were high. According to Mustafavand and Kamgar (2013) the main problem impeding mechanical harvesting of chickpeas is the excessive grain losses at initial cutting and feeding of the crop. Very little works has been done on the mechanical harvesting of chickpea. In addition, existing literature in this regard is very brief and scattered but the researcher expressed the chickpea harvest is similar to the lentil harvest. Different kinds of mechanism have been used on chickpea harvesting machines. There has not been much study, however, on the design and development of chickpea harvesting machines. Haffar et al (1991) and Siemens (2002) applied conventional combine harvesters for chickpeas but losses were high. Chakraverty et al (2003) reported that the optimum losses for mechanized chickpea harvesting were 5.5%.

From the literature review, it can be seen that the main reason for the high loss of chickpea harvesting machines is use of the mechanical system for conveying of chickpea. Therefore, redesign and modification of the conveying mechanism increase the work quality. The pneumatic conveyor is one of the conveying mechanism that had less losses than typical losses of conventional conveying methods in conveying of agricultural materials. Pneumatic convey is a technique to convey powder and granular material constantly by using air energy.

In recent years, researchers (golpira et al, 2011) used of pneumatic conveying for chickpea Fig. 1. It includes a suction fan and separator cyclone. A stream of air from the fan and separator cyclone is utilized to drag and separate seed from air. In order to obtain an effective drag action, the fan has to generate an even airflow with proper speed over. The proper air speed can be determined from aerodynamic properties of agricultural materials which are the terminal velocity and drag coefficient of the material (khoshtaghaza and mehdizadeh, 2006).

Fig. 1.Conveing system (a: cyclone, b: suction fan)[5].

Different techniques can be used for measuring aerodynamic properties. Carman (1996) and Konak et al (2002) used the free fall method to determine the terminal velocity of seeds while Joshi et al (1993) and Singh and Goswami (1996) determined the terminal velocity by using a wind column. Some engineering properties of chickpea seeds, such as density, terminal velocity and coefficient of drag, were reported by Kural and Carman

(1997).

The fan type used in conveying section are shown in Fig. 2. As solid particles are introduced in a flowing stream of air in a duct they are subjected to aerodynamic drag. If the generated suction air velocity is sufficiently high the particles accelerate and the drag is reduced, because the relative velocity between the particles and the air also is reduced (srivastava et al, 2006). When the particles are being conveyed, the drag overcomes the forces of gravity, particle-to-particle interaction, and friction between the particles and the conduit wall. As the number of particles in the airstream are increased as a result of a higher conveying rate, the resistance to airflow increases. If the conveying rate of the solids continues to increase, there comes a point when the particles no longer behave as discrete particles. The phase when the solids are in a uniform suspension is called the dilute phase (srivastava et al, 2006). Conveying of agricultural material is done in the dilute phase. So, increase in suction velocity plays a crucial role for the performance of the pneumatic conveyor.

Design and performance prediction process of fan is still a difficult task, mainly due to the great number of free geometric parameters involved. On the other hand the significant cost and time of the trial-and-error process by constructing and testing physical prototypes reduces the profit margins of the fan manufacturers. For this reason, CFD analysis is currently being used in hydrodynamic design for many different fan types (jafarzadeh et al, 2011).

For experimental validation, hot-wire anemometry (HWA) is a widespread measurement technique in the study of turbo machinery flow (mekonnen et al, 2010).

In the current study the effect of various turbulence models (k–ε, RNG and RSM) on the flow field and efficiency of a suction fan has been carried out. Using the suitable model the effect of location of inlet on the specific characteristic of the fan has been investigated.

Fig.2. Suction fan.

Materials and methods

The measurement of aerodynamic properties

Dry matter chickpea (kaboli) seeds were used for all the experiments in this study. The crop was grown in 2014 during the spring season at the farm University of Kurdistan, Iran.

The initial moisture content of seed, stem and pod were determined by following a standard method and the aerodynamic properties terminal velocity and drag coefficient of them were assessed at moisture levels of 8.51, 16.67, 34.45 and 71.15% db with ten replications at each level by using a wind column as shown in Fig.3 and using the following relationship given by Mohsenin (1997) respectively.

$$C_D = \frac{6\pi\mu d_p}{A_p \rho_f v_t} \tag{1}$$

Where μ is the fluid viscosity in $N.s.m^{-2}$, d_p is the geometric mean diameter of chickpea in mm, A_p is projected area in mm^2, ρ_f is fluid density in $kg.m^{-3}$ and V_t is terminal velocity in $m.s^{-1}$.

To determine the average size of the seed, a sample of 100 seeds was randomly selected. Measurements of the three major perpendicular dimensions of the seed were carried out with a micrometer to an accuracy of 0.01mm. The geometric mean diameter d_p of the seed was calculated by using the following relationship (m0stafavand et al, 2013).

$$d_p = \sqrt[3]{lwt} \tag{2}$$

Where l is the length, w is the width and t is the thickness in mm. The geometric mean diameter of chickpea between 13.55- 14 mm calculated.

Fig.3. The wind column for measure of terminal velocity.

The experimental measurement of fan

The experimental measurement of fan for validation of the simulation CFD method by using the hot-wire anemometer were conducted**.** The hot-wire anemometer measured the velocity across the inlet of fan. The measurements were conducted at eight different flow rates. Table 1, shown the results of the experimental measurement.

Table 1
The results of the experimental measurement of suction fan.

RPM_{motor}	$RPM_{impeller}$	V_{inlet}
800	549	10.08
950	680	13
1050	729	14.7
1150	818	15.7
1250	885	18.5
1350	940	18.93
1450	1000	21
1550	1085	21.09

Fan specifications

The simulated fan includes a circular inlet with diameter 13 centimeter, a 5-blade impeller with D_{in} and D_{out} equal 15 and 54 centimeter respectively and rectangular outlet with width and length equal 32 and 100 centimeter. The specific speed is defined as (jafarzadeh et al, 2011):

$$N_s = \frac{\Omega Q^{0.5}}{H^{0.75}} \tag{3}$$

where Q is volume flow rate in $m^3.s^{-1}$ and H is the fan head in m.

Governing equations

Since the fluid surrounding the impeller rotates around the axis of the fan the equations must be organized in two reference frames, stationary and rotating reference frames. To accomplish this, the Multiple Reference Frame (MRF) model has been used. In this approach, the governing equations are set in a rotating reference frame, and Coriolis and centrifugal forces are added as source terms. Continuity and momentum equations of an incompressible flow are as the following (jafarzadeh et al, 2011):

$$\nabla . u = o \tag{4}$$

$$\nabla . (\rho u u) = -\nabla_p + \nabla_\tau + s \tag{5}$$

In the above equations u is the relative velocity of fluid in $m.s^{-1}$, s the stress tensor in pa and s is the source term, which consists of Coriolis and centrifugal forces

$$s = -2\rho\Omega \times u - \rho\Omega \times (\Omega \times r) \tag{6}$$

Here Ω is rotational speed in $rad.s^{-1}$ and r position vector in m.

Grid generation

The fan is divided into two regions, casing and rotary. The rotary region is not discretized. Structured grids are used for casing and rotary regions. In Fig. 3 the grids in four case of fan are shown. In the present study, four sets of grids were used for grid study.

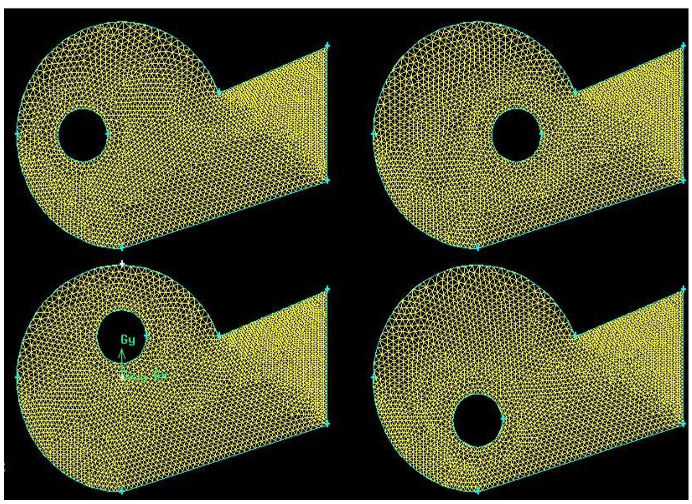

Fig.3. Gridding in fan for 4 cases.

Selection suitable turbulence model

For selection of suitable turbulence model, three turbulence models k-ε, RNG and RSM with the experimental data in the form of characteristic curves were compared. In this curves used from the non-dimensional head and flow coefficients. The non-dimensional head and flow coefficients are defined as (jafarzadeh et al, 2011):

$$\Psi = \frac{gH}{\Omega^2 r^2} \tag{7}$$

$$Q_c = \frac{Q}{\Omega r^3} \tag{8}$$

Boundary conditions

In the present study, velocity inlet and fan intake boundary conditions were used for the inlet and outlet, respectively. Outer walls were stationary but the inner walls were rotational. There were interfaces between the stationary and rotational regions. Non slip boundary conditions have been imposed over the impeller blades and walls, the volute casing and the inlet wall and the roughness of all walls is considered 100 lm. The turbulence intensity at the inlet totally depends on the upstream history of flow. Since the fluid in the suction tank is undisturbed, the turbulence intensity for all conditions is considered 1%. Air was used as a working fluid in ambient condition.

Numerical scheme

In order to calculate the flow field a commercial CFD code, FLUENT, was used. The governing integral equations for the conservation of mass, momentum and when appropriate, energy and other scalars such as turbulence were solved. Two numerical solvers of segregated and coupled employ a similar discretization process, but the approach used for linearizing and solving the discretized equations is different. The segregated solver solves the governing equations sequentially, while the coupled solves them simultaneously. In the present analysis, the segregated solver was used since the coupled solver is usually used in high compressible flows in which the flow and energy equations are coupled, and this method often results in a faster solution convergence. A trade-off involved in the use of the coupled solver is that it requires more memory (1.5–2 times) than the segregated solver.

The Pressure-velocity coupling methods recommended for steady-state calculations are SIMPLE or SIMPLEC [10,11]. For relatively uncomplicated problems in which convergence is limited by the pressure-velocity coupling, the convergence could be achieved more quickly using SIMPLEC. With SIMPLEC, the pressure-correction under-relaxation factor is generally set to 1.0, which aids in the convergence speed-up. In some problems, however, increasing the pressure-correction under-relaxation to 1.0 can lead to instability due to the high grid skewness. In the present simulation, SIMPLE algorithm was preferred considering the complexity of the flow and grid qualities (jafarzadeh et al, 2011).

FLUENT provides a powerful set of features for solving problems in which fluid rotates around an axis, such as flows inside turbo machineries in different methods. Some of these methods include multiple reference frames (MRF), mixing plane and sliding mesh models. Each method has a different accuracy and computational expenses. The first and second models are appropriate for steady flows and for cases in which the interactions between rotor and stator are negligible. For instance, for a fan with bladeless stators the MRF can be used as a suitable approach. The sliding mesh model is appropriate where the interaction between rotor and stators is noticeable and the unsteadiness of problem is supposed to be reproduced. Since in the present problem, the stator has no blade and the unsteadiness of the problem can be ignored, the MRF model was used.

Results and discussion

Modeling of turbulence

Fig. 3 shows the characteristic curve for the three turbulence models compared with available experimental data. The curves show that with increasing the flow coefficient, the head coefficient is decreased. Comparing various turbulence models data with experimental data, it has been concluded that each of these turbulence models provide acceptable results, but the models RNG show better agreement than the standard k–ε and RSM model.

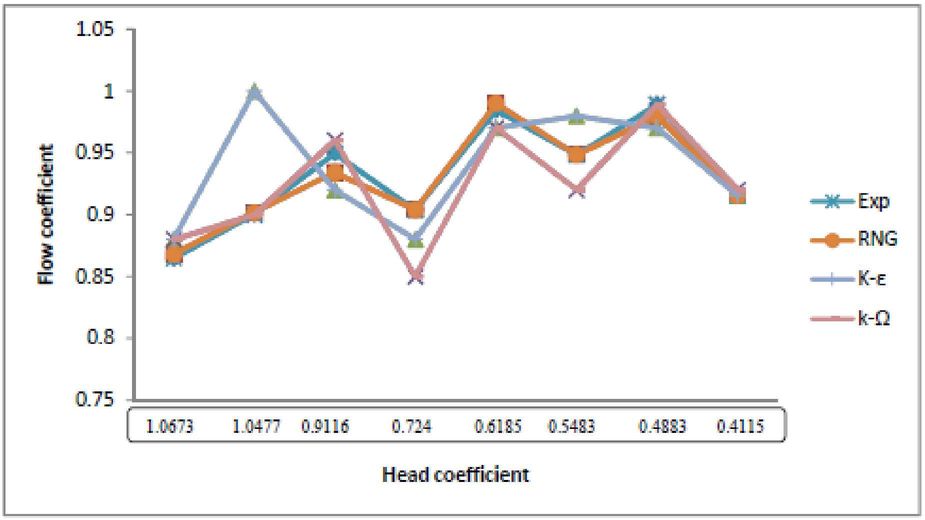

Fig.3. Head coefficient vs. flow coefficient with three different turbulence models and one available experimental data.

Effect of location of inlet on the fan characteristics

Fig 4 and 5, shown the simulation of flow for suction fan in five different time. In this figures, contours of velocity drawn for fan in four cases. The results of CFD shown when location of inlet is in the below section, suction of fan increased. The small eddy are the main reason for reducing of the suction. When small eddy put in the front of outlet flow, suction decreased.

Table 2

The experimental measurement of fan in for cases

$\mathbf{RPM_{motor}}$	800	950	1050	1150	1250	1350	1450	1550
$\mathbf{RPM_{impeller}}$	549	680	729	818	885	940	1000	1085
Below	13.39	14	16.3	17	18.5	19.7	21	25
Top	11.2	12.3	13.3	14.34	15.4	17.2	19.7	22.7
Left	10.08	12.83	13.3	14.3	15.92	17.73	19.64	22.3
Right	9.56	10.94	11.15	11.83	12.9	13.74	15.48	17.54

In fig 4, the small eddy is behind of outlet flow while in other cases small eddy are in the front of outlet flow. Table 2 shown the results of the experimental measurement of fan in for cases.

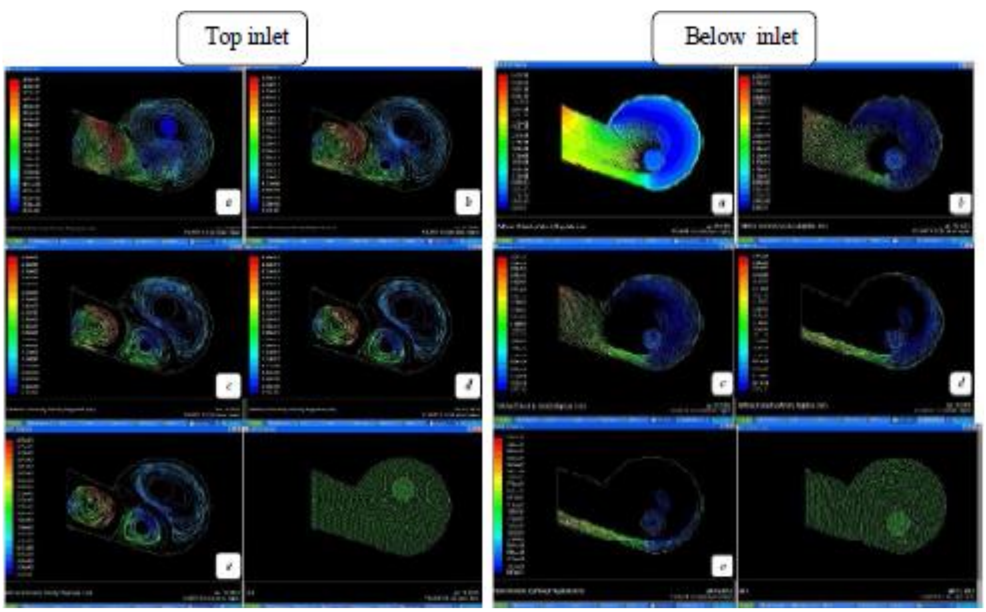

Fig.4. The contours of velocity for fan with the below inlet and top inlet.

Conclusion

In the present investigation numerical simulation of a suction fan was performed. At first the optimum turbulence model for the problem was found. Considering the available experimental data, the best result appears to be obtained by RNG model. Investigation on the effect of location of inlet on the increasing of suction fan shows that the fan with below inlet has the highest suction when compared with other cases at all ranges.

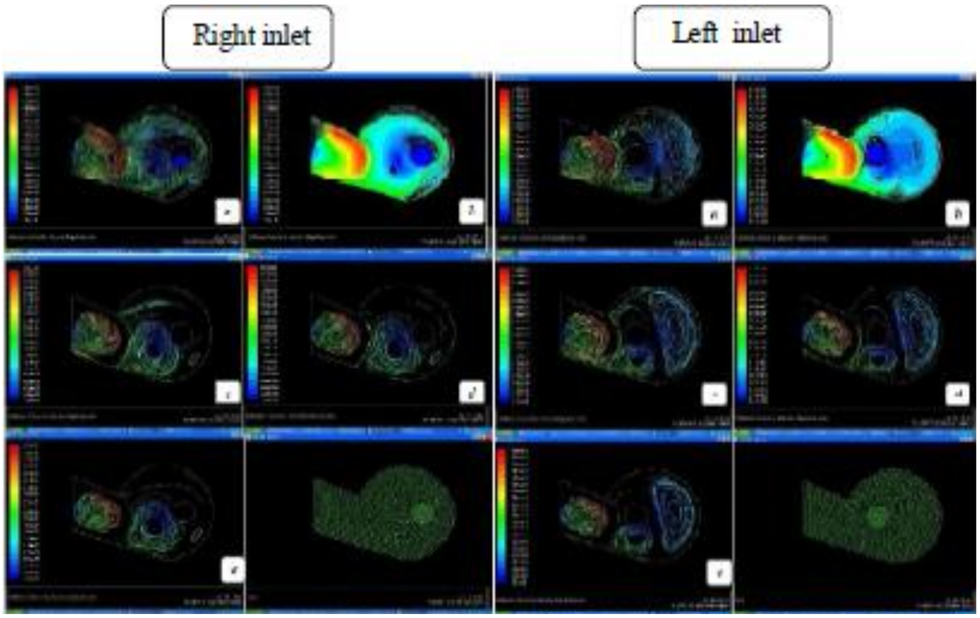

Fig.5. The contours of velocity for fan with the right inlet and left inlet.

References

Behroozi-Lar, M. and Huang, B.K. 2002. Design and development of chickpea combine, Agricultural Machines American, 33 (1): 35-38.

Carman, K. 1996. Some physical properties of lentil seeds. Journal of Agricultural Engineering Research, 63(2): 87–92.

Chakraverty, A., Mujumdar, A.S., Raghavan, G.S.V. and Ramaswamy, H.S. 2003. Handbook of postharvest technology cereals, fruits, vegetables, tea, and spices. Marcel Dekker Inc, NY.

Fluent, Inc. 2006. FLUENT 6.3 User's Guide. Lebanon, NH, 845P.

Golpira, H. and Tavakoli, T. 2011. The design and development of a chickpea harvester. I R Patent No. 68914. Date issued: 23 February. [In Farsi].

Haffar, L., Singh, K.B. and Birbari, W. 1991. Assessment of chickpea grain quality and losses in direct combine harvesting. T ASABE 34(1): 9-13.

Jafarzadeh, B., Hajari, A., Alishahi, M.M. and Akbari, M.H. 2011. The flow simulation of a low-specific-speed high-speed centrifugal pump. Applied Mathematical Modeling, 35: 242–249.

Joshi, D.C., Das, S.K. and Mukherjee, R.K. 1993. Physical properties of pumpkin seeds. Journal of Agricultural Engineering Research, 54(3): 219–229.

Khoshtaghaza, M. and Mehdizadeh, R. 2006. Aerodynamic properties of wheat kernel and straw materials. Manuscript FP 05 007. Agricultural Engineering International: the CIGRE journal, VIII.

Konak, M., Carman, K. and Aydin, C. 2002. Physical properties of chickpea seeds. Biosystems Engineering, 82 (1): 73–78.

Kural, H. and Carman, K. 1997. Aerodynamic properties of seed crops. National symposium on mechanization in agriculture Tokat, Turkey, 615–623.

Mekonnen, G.G., Baerdemaeker, J.D and Baelmans, M. 2010. Effect of a cross-flow opening on the performance of a centrifugal fan in a combine harvester: Computational and experimental study. Biosystems Engineering, 105: 247-256.

Mohsenin, N. N. 1970. Physical Properties of Plant and Animal Materials. Gordon and Breach Science Publishers, New York.

Mostafavand, H. and Kamgar, S. 2013. Comparison between feeder and cutter mechanism, Conventionally combine and hand pulling method on chickpea harvest at different grain moisture contents. International Journal of Agronomy and Plant Production, 4:9:2174-2178.

Siemens, M.C., Corp, M.K. and Correa, R.F. 2002. Effects of header modifications on garbanzo bean harvesting losses, Columbia Basin Agricultural Research Annul Report, SR 1040: 37-43.

Singh, K.K. and Goswami, T. K. 1996. Physical properties of cumin seed. Journal of Agricultural Engineering Research, 64(2): 93–98.
Srivastava, A., Goering, C., Rohrbach, R. and Buckmaster, B. 2006. Engineering Principles of Agricultural Machines, American Society of Agricultural and Biological Engineers, 2nd edition. 552P.
Tabatabaeefar, A., Aghagoolzadeh, H. and Mobli, H. 2003. Design and development of an auxiliary chickpea second sieving and grading machine. Agricultural Engineering International, the CIGRE journal, Vol. V, manuscript FP 03 005.
Tado, C.J., Wacker, M.P., Kutzbatch, H.D. and Suministrado, D.C. 1998. Development of stripper harvesters: a review. Journal of Agricultural Engineering Research, 71: 103-112.
USDA (1970) Official Grain Standards of the United States. US Department of Agricultural Consumer and Marketing Service Grain Division, Revised.
Vicar, R.M. 1999. Harvesting chickpea. Special crops, Saskatchewan agriculture and food.