Modeling airflow inside and around hoods used for pneumatic control of pest insects. Part II: Application and validation of the model

M. KHELIFI1, C. LAGUË1 and J.-L. ROBERT2

1Soil Science and Agri-Food Engineering Department and 2Civil Engineering Department, Université Laval, Québec, QC, Canada G1K 7P4. Contribution No. CRH-159 from the Centre de recherche en horticulture. Received 3 November 1995; accepted 3 October 1996.

Khelifi, M., Robert, J.-L., and Laguë, C. 1996. Modeling airflow inside and around hoods used for pneumatic control of pest insects. Part II: Application and validation of the model. Can. Agric. Eng. 38:273-281. To optimize the design of pest control machines, the airflow inside and around blowing and suction hoods of different configurations for the specific case of pneumatic control of Colorado Potato Beetles (CPBs) was numerically modeled. Such predictions of airflow patterns can help in locating the optimal zones for CPB removal for a given configuration. Preliminary results show that two particular hood configurations (horizontal blowing and simultaneous oblique blowing and suction) that generate different airflows across the potato plant foliage are the most promising. Airflow inside and around these two configurations was further investigated using the contour element concept. This allowed airflow modeling at the wall level. Evaluation of the numerical model performance proved its validity and capability in adequately predicting airspeeds. Results are sufficiently accurate and detailed to serve as the first stage of the design process. Keywords: modeling, airflow, finite element, design, pneumatics, pest control, potato.

Pour optimiser la conception de machines destinées à contrôler les insectes nuisibles, l’écoulement de l’air à l’intérieur et autour des buses de soufflement et d’aspiration placées selon différentes configurations a été numériquement modélisé (à des fins de contrôle du doryphore de la pomme de terre (DPT)). La prédiction du patron d’écoulement de l’air pourrait aider à localiser les zones propices à l’élimination du DPT pour une configuration donnée. Les résultats préliminaires montrent que deux configurations, soit le simple soufflement horizontal, soit le soufflement et l’aspiration obliques simultanés, sont prometteuses. L’écoulement de l’air à l’intérieur et autour de ces deux configurations a été examiné en détail en introduisant la notion d’élément de contour pour modéliser l’écoulement de l’air au niveau des parois. L’évaluation des performances du modèle numérique prouve sa validité et son aptitude à prédire adéquatement les vitesses de l’air. Les résultats sont suffisamment précis et détaillés pour servir à la conception préliminaire de ces machines. Mots clés: modélisation, écoulement de l’air, éléments fins, conception, pneumatiques, contrôles des insectes nuisibles, pomme de terre.

INTRODUCTION

For a long time, the use of chemicals in agriculture has contributed to the increase of crop yields. However, some insects such as the Colorado Potato Beetle (CPB) have become resistant to many insecticides, including those that were very effective at one time (Forgash 1981; Boiteau et al. 1987). In some areas, these applications have contaminated the environment and created public health problems.

For the past few years, many attempts have been made at investigating alternatives to insecticide application. The use of pneumatic systems to remove the CPB from potato plants has attracted the attention of many growers. Some machines were designed and tested in potato fields; however, none of these machines currently provides complete satisfaction (Boiteau et al. 1991; Duchesne and Boiteau 1992; Puttré 1992). This is mostly attributed to the inadequate design of the operating units. Khelifi et al. (1992) also reported a lack of scientific data on pneumatic control of insect pests. Indeed, only very few studies have been done on this particular subject (deVries 1987; Misener and Boiteau 1991, 1992).

Pneumatic control of the CPB is a complex process. Its efficiency depends on many factors including: the variability in the gripping ability of the insects at different growing stages, the potato plant geometry, and the resistance of crop foliage to airflow. The success of this technique appears to depend upon an appropriate design of the control units, i.e. the hoods. The main factors that have to be taken into consideration are the geometry or the shape of the hoods, which greatly affects the airflow pattern at the plant foliage level, the dimensions of the hoods, their position relative to the plants, and obviously the airflow rate that the hoods can deliver.

The selection of the optimal hood geometry also depends on many factors such as the growing stage of the plants and the mode of action planned, i.e. total or partial coverage of the plants by suction, blowing, or a combination of both.

Experimentally, Dalla Valle in 1930 (Burgess et al. 1989) was the first researcher to investigate the capture velocity in front of hoods of different shapes. Dalla Valle derived for this purpose some empirical formulas. However, according to Flynn and Ellenbecker (1986), the capture velocity is not an adequate design parameter capable of providing the necessary information about the performance of hoods. Recently, many researchers like Flynn and Ellenbecker (1986, 1987), Ellenbecker et al. (1983), and Conroy et al. (1988) investi-
gated the performance of hoods in the presence of an airflow perpendicular to the air suction direction. They used a more appropriate index, the capture efficiency. It was simply defined as the proportion of particles captured among those released by the process to be controlled. According to Burgess et al. (1989), the use of the notion of capture efficiency as a design tool is very limited. Indeed, the research at this level is still at a preliminary stage and has only dealt with low air velocity, i.e. the terminal velocity of dust particles which is not appropriate for pneumatic control of pest insects.

The objective of this study was therefore to numerically model, by the finite element method, the airflow inside and around hoods arranged in simple geometries (air suction or blowing) and in more complicated ones (combinations of two to three hoods). This presents the advantage of predicting airflow patterns which allows for the detection of the most propitious zones for a better removal of CPB. The best combination, and consequently the most adequate geometry ensuring an efficient control of CPB, could therefore be selected as the working unit for subsequent field testing. The numerical model used for these simulations was developed by Khelifi et al. (1996). The performance of this model was preliminarily tested in a simple case, the simulation of airflow inside a round 0.4 m long x 0.15 m radius duct.

**PROBLEM DESCRIPTION**

**Airflow combinations**

In this study, the following combinations of airflow were investigated: a simple air suction from the top, a simultaneous horizontal air suction and blowing through the plant foliage, a simultaneous oblique ascending air blowing and suction, two horizontal air jets blowing across the plant and air suction provided at the top, and two oblique ascending air jets blowing and air suction at the top. All these airflow combinations are illustrated in Fig. 1.

**Hood geometries**

The different configurations required to ensure the necessary airflows for this study were obtained from some combinations of hoods already tested in the laboratory. The static pressure measured at the inlet and the outlet of these hoods was taken into consideration. Figure 2 presents the modeling domain of a hood in a suction position. The boundaries of the domain were selected sufficiently away from the vacuuming area in order to obtain the maximum information about the airflow fields and to avoid any possible interference with the expected results. All modeling domains for the other configurations were selected on the same basis. In axisymmetric situations like that of Fig. 2, the study was carried out on only half of the domain and null airflow flux conditions were imposed at the axis of symmetry.

**Fluid properties**

This study involves the flow of air at ambient temperature (20°C) and atmospheric pressure (101.3 kPa). In this case, the two necessary fluid properties, namely the kinematic viscosity, \( \nu \), and the density, \( \rho \), have the magnitudes (Giles 1984):
\[
\nu = 1.4888 \times 10^{-5} \text{ m}^2/\text{s} \quad \text{and} \quad \rho = 1.2047 \text{ kg/m}^3.
\]
MODEL VALIDATION

To validate the numerical model, two of the tested configurations (horizontal blowing, and simultaneous oblique blowing and suction) were carefully reproduced in the laboratory using the same hoods simulated by the finite element method. The semicircular shield of the combined configuration was made from 0.4 x 0.8 m sheet metal. All the tests were carried out on the test bench described by Khelifi et al. (1992). Airspeed was measured at the inlet (suction) and outlet (blowing) of the hoods using a 2% precision telescopic anemometer (Multi-purpose Solomat Instrument, Solomat Instrumentation Division, Norwalk, CT) and a Prandtl tube along with a pressure anemometer. However, before taking the speed measurements, a marking of the measuring points corresponding to the simulation nodes was performed as described in Fig. 4. This allowed a sweep of the cross section of the hoods (inlet/outlet) with either the anemometer or the Prandtl tube to record the necessary measurements at specific points located on the longitudinal central axis.

For each configuration, measured and predicted airspeeds were plotted.

Variables or degrees of freedom

The dependent variables for this particular problem are the pressure, $p$, and the two components ($u$, $v$) of the air velocity vector along the two directions of an orthonormal Cartesian reference plane ($x$, $y$).

Boundary conditions

All boundary conditions were imposed on $u$, $v$, and $p$ over the boundaries such as those presented in Fig. 3 for the simultaneous air suction and blowing configuration. The boundary conditions of the other configurations were imposed on the same basis. When only $p$ was imposed without specifying any value for $u$ or $v$, the normal velocity gradient was set to zero.

Modeling of the hoods

The numerical part of this study was completed at the Numerical Analysis and Computing Research Center of the Department of Civil Engineering of Université Laval. The model developed by Khelifi et al. (1996) was used for this purpose. The resolution of the system of equations was achieved by the MEFL3D code (Robert 1994) and the pre-and post-treatment by the MOSAIC software (Compiègne Science Industrie, Compiègne, France). Three different elements were principally used to mesh the modeling surfaces and domains: a nine-node quadrilateral (Q9-4), a six-node triangle (T6-3), and a three-node linear (L3-2).

Fig. 2. Schematic representation of a suction hood modeling domain.

Fig. 3. Example of the domains and frontiers for the simultaneous air suction and blowing configuration.

Fig. 4. Marking of different points to measure airspeeds along the longitudinal central axis of the hoods.
on the same graph in order to obtain an idea about the velocity profiles. Thereafter, they were plotted against each other in order to determine the degree of agreement between them to check qualitatively how well the model predicted the measured values. Also, to have a better idea about the numerical approach, the error made over measured values was estimated. Finally, the model performance was further evaluated using one of the criteria suggested by Fox (1981) and Willmot (1982), the mean absolute error (MAE):

$$\text{MAE} = N^{-1} \sum_{i=1}^{N} |P_i - O_i|$$  \hspace{1cm} (1)

where:
- $N$ = number of cases,
- $P_i$ = predicted value, and
- $O_i$ = observed or measured value.

The MAE gives a general idea about the mean difference between measured and predicted values, hence, of the accuracy of the model.

To get the most accurate approximation of the solution, especially inside and around the hoods, fine elements were used to mesh these areas. The elements were gradually increased in size toward the boundaries of the modelled domains. An example of one such finite element mesh for a suction hood is illustrated in Fig. 5. This figure shows only half of the domain of modelisation, thus half of the hood because it is an axisymmetric case.

As mentioned by Khelifi al. (1996), the mixing length “L” concept was used to ensure turbulent velocity profiles. For this purpose, “L” was calculated at each node for every configuration before solving the model. An example of the results, visualized as a vertical displacement, is presented on Fig. 6 for the same configuration as Fig. 5.

Figure 7 shows the approximated velocity field for the simple suction configuration. The presence of the flanges confirms the results reported by McDermott (1977). Indeed, it seems that they can effectively enlarge the flow pattern. However, we note that the velocity magnitude decreases drastically from inside the hood toward the ground. This shows that the air suction efficiency is limited to a zone close to the hood inlet. This configuration, alone, could not therefore be used to remove efficiently CPB because of their capability to grasp to potato foliage.

In Fig. 8 (the simple air blowing configuration), the air-
Figure 9 presents the results of a combination of the two previous configurations, i.e. a simultaneous air suction and blowing across the plant foliage. One of the advantages of this configuration is to keep the plant close to the suction hood under the effect of blown air. In this case, the efficiency of the suction can be kept at a high level. We can, however, note that there is some airflow escaping away between the two hoods. It would then be practical to provide a shield to prevent this air from escaping.

The configuration presented in Fig. 10 includes a semicircular shield having a 0.40 m radius centered at the intersection point of the y and horizontal hood axes. In addition, hoods were tilted upward 20° from their horizontal axis. This directs the airflow more toward the upper part of the plant where the CPB tend to feed first. Results show that the previous airflow loss is remarkably reduced. However, the perfect semicircular shape of the shield has to be slightly modified to keep airflow in contact with it and consequently prevent any eventual possible recirculation of air.

The configuration of Fig. 11 is similar to that of Fig. 10 except for the addition of a deflector inside the blowing hood.

Figure 12 shows the results of a two opposite air blowing across the plant foliage and an air suction from the top configuration (air velocities are in m/s). Flow maintains a considerable velocity even very far from the hood outlet. It is also wide enough to cover a large part of the plant. The air blowing efficiency appears to be far better than that of suction because the air can penetrate more in distance for improved dislodging and removing of CPB.
This was expected to enlarge the airflow pattern in the vicinity of the hoods resulting in more air covering the plants. However, results show that this deflector badly affected the airflow as an important zone of air recirculation is generated at the upper left side of the shield (not shown on the figure). This is mainly due to the boundary conditions of the deflector. Furthermore, the air velocity distribution is not improved by the introduction of the deflector although the airflow loss from the bottom can be totally eliminated.

The main idea behind the use of the configuration presented in Fig. 12 was to dislodge the CPB by two opposite air streams blowing across the plant foliage and to suck them out from the top. Unfortunately, the analysis of the simulation results show the presence of a stagnation point slightly under the blowing hood axis. This appears to be a result of the convergence of the two air streams blowing in opposite directions. The immediate consequence of this is the loss of a considerable amount of air velocity at the ground level.

To remedy this problem, the two blowing hoods were rotated 20° from their horizontal axis (Fig. 13). In this case, the air loss due to the interference of the two air streams is greatly reduced. Again, the simulation results show that a stagnation point is still present. In addition, a large amount of air is escaping through the space between the suction and blowing hoods as well as from underneath. This may considerably reduce the efficiency of this configuration.

One way to solve this problem consisted of sealing the space between the suction and blowing hoods (Fig. 14). As a result, the airflow loss at the upper part could be totally eliminated. However, a stagnation point was again present. In this case, either the blowing hoods have to be rotated more than 20° or the air suction power has to be increased which is costly.

Results of the numerical modeling using the contour element concept are presented in Figs. 15 and 16. In contrast to the previous results for the simple horizontal air blowing and shielded simultaneous oblique air suction and blowing configurations, airspeeds appear different from zero at the wall level (especially for the suction hood).

Figure 17 shows that predicted airspeeds for the simple horizontal air blowing configuration are very close to those measured in the laboratory. However, the difference between predicted and measured airspeeds seems more pronounced in Figs. 18 and 19. This is mainly due to the slight inclination of the hood flanges which did not allow for an accurate positioning of the anemometer at the measuring points level.
Airspeeds were then measured on a backward shifted axis (towards the outlet of the hood) in contrast to that initially designated. Airspeeds presented in Fig. 18 were measured with a Prandtl tube because of the simplicity of the configuration.

Figures 17 to 19 show that airspeed profiles are far from the classical turbulent profiles although the Reynolds number is greater than $2.1 \times 10^3$, the transition threshold between laminar and turbulent regimes. This could be explained by the fact that the ducts are very smooth and not subjected to many vibrations (Bird et al. 1963).

In Figure 18, a slight discrepancy between measured and predicted airspeeds at the central node levels appears. The minimum airspeed measured at those nodes was 12 m/s against a simulated airspeed of about 14 m/s. To investigate this disagreement, the model was subjected to many tests. A uniform airflow in a duct was used for this purpose. The model responded satisfactorily because the adjustment of airspeed profiles was easily made by only varying the friction coefficient. In our case, the airflow is both highly non-uniform due to the inclination of the hood walls (24°) and more complicated by the presence of a simultaneous blowing and suction process. These two phenomena could explain the difference between predicted and measured airspeeds at the central node levels for the suction hood. This discrepancy was not observed in the simple and combined air blowing cases.

On the other hand, the model adequately predicts airspeeds at the extreme nodes situated on the walls at the outlet and inlet of blowing and suction hoods, respectively. For the blowing hoods, airspeeds are high enough in the expansion zone that the airflow could not remain in contact with the internal walls of the diffuser. It lifts before leaving the hoods which gives null airspeeds at the extreme nodes (Figs. 16 and 19). Predicted and measured airspeeds near the wall at the inlet of the suction hood are higher (about 5 m/s). This is mainly attributable to the effect of the airflow jet coming from the blowing hood. Practically, an airspeed as high as 5 m/s in these locations greatly contributes to the suction of more CPBs.

Figure 20 shows that the model predicts adequately measured speeds for the simple air blowing configuration.
Fig. 21. Predicted vs. measured combined air suction speeds.

Fig. 22. Predicted vs. measured combined air blowing speeds.

CONCLUSIONS AND RECOMMENDATIONS

1. Numerical modeling can be used to investigate the airflow patterns for pneumatic control of CPB without the need of designing and building costly prototypes.

2. Among the eight configurations studied, two appeared promising: the horizontal air blowing and the shielded simultaneous oblique ascending air suction and blowing across the plant foliage.

3. The horizontal air jet could be efficiently used provided an adequate catching system is set to catch the CPBs blown out by the air.

4. The shielded simultaneous oblique ascending air suction and blowing configuration does not require a catching system because the suction hood ensures this task. However, the shield has to be improved to avoid any eventual possible air recirculation.

5. Results of the evaluation of the numerical model performance are satisfactory and prove the validity and the capability of the model to adequately predict measured airspeeds.

6. A future improvement to the model would require the introduction of a potato plant in the airflow domain. This could be accomplished by empirically introducing a friction zone equivalent to a plant.

ACKNOWLEDGMENTS

The authors gratefully acknowledge the financial support of the Natural Science and Engineering Research Council of Canada, Premier Tech Ltd., Provigo Inc., and the Conseil de recherche en pêche et en agro-alimentaire du Québec. Special thanks are extended to Philippe Savoie, research scientist at Agriculture and Agri-Food Canada, Sainte-Foy Research Station, and adjunct professor at Département des sols et de génie agroalimentaire, Université Laval, for revising the manuscript.

REFERENCES


Forgash, A. J. 1981. Insecticide resistance of the Colorado...


Robert, J.-L. 1994. MEFL3D manuel de référence. Dépt. de génie civil, Université Laval, Québec, QC.