VOL. 12, NO. 7, APRIL 2017 ISSN 1819-6608
ARPN Journal of Engineering and Applied Sciences
©2006-2017 Asian Research Publishing Network (ARPN). All rights reserved.

www.arpnjournals.com

VALIDATION FRAMEWORK FOR FLUID-PARTICLE-STRUCTURE-INTERACTION SOLVER

Nazri Huzaimi Bin Zakaria[1], Mohd Zamani Bin Ngali[2] and Eng Pei Ying[2]
[1]Faculty of Mechanical Engineering, Universiti Teknikal Malaysia Melaka, Hang Tuah Jaya, Durian Tunggal, Melaka, Malaysia
[2]Faculty of Mechanical and Manufacturing, Universiti Tun Hussein Onn Malaysia, Parit Raja, Johor, Malaysia
E-Mail: nazrihuzaimi@utem.edu.my

ABSTRACT

Verification and validation (V&V) framework has always been an essential part in solver development. In Computational Fluid Dynamics (CFD), fluid, fluid-particle and fluid-structure solvers have already embraced specific benchmark case studies for the V&V framework. For the more complex fluid–particle-structure- interaction (FPSI) solver however, the V&V framework is still on its infant phase. Therefore, a standard framework to validate the FPSI is thoroughly developed in this work. Granular fertilizer boom sprayer is presented as the case study since the system design includes all the important interaction characteristics. The V&V framework has to be obliged to include all V&V procedures for fluid, fluid-particle and fluid-structure solvers earlier before the fully integrated FPSI solver is evaluated. Therefore the benchmark studies of driven flow in a square cavity, the experiment of particle trajectory in a lid-driven cavity and two-dimensional flow over a thin elastic beam attached to a rigid and fixed square block are used to validate the fluid solver, fluid-particle solver and fluid-structure solver respectively. Once the supposed initial procedures are verified, the framework is concluded with the validation for the FPSI solvers. The developed V&V framework is extremely practical to assure the accuracy of the solvers involved without compromising the accuracy at any of the two-way couple interactions.

Keywords: fluid solver, fluid particle, fluid structure, solver benchmark.

INTRODUCTION

Nowadays in engineering industries, simulations are facing higher demand than ever. Fluid flow is not only coupled with particle distribution or structure deflection as a standalone two-way interaction solver anymore. Industrial players such as those in oil and gas industry are craving for solvers that capable of simulating the interaction between fluid, particles and structures altogether. In crude oil transportation within their piping system, the momentum of sand particles carried across the pipe streamlines impinges the wall of the fittings. This observable fact results in erosion damage. Erosion of the fittings may result in failure of the piping system, which can be dangerous and expensive. The prediction of erosion is not only allows us to estimate service life, but also enables the detection of locations in the geometry where severe erosion is likely to occur. Elbows and plugged tees are common geometries used in piping systems to transmit fluids. Both elbows and plugged tees are exposed to erosion when sand particles are present because particles deviate from the fluid streamlines and impact the wall when they pass through the geometries [1]. Based on these cases, it is essential to have a method to determine the erosion rate for a given set of operating conditions to prevent any failures from occurring.

In software engineering, the engineers and researchers attempt to solve problems in several different kinds of methods. To do so, they produce several dissimilar types of results, and they should develop appropriate evidence to validate these results. Verification and validation (V&V) framework have always been an essential part of the stage in engineering process, because they offer the only way to judge the success based on the project development. In this paper we present and discuss a concrete framework for validation of Fluid-Particle-Structure-Interaction (FPSI) solver.

Significant efforts have been taken by researchers and engineers to study the problem in fluid solver, fluid-particle-interaction solver, fluid-structure-interaction solver and many prediction models have been developed. For example [2] studies the fluid analysis for 2D viscous flows. [3-8] studies about the fluid of the incompressible flow thru one body by using Navier Stokes Equation. [9]–[14] investigated the particle movement by using the Lagrange Mutiplier/ fictitious domain method. [15]-[19]are used Eulerian-Eulerian approach and Eulerian-Lagrangian approach [20]-[29] to investigate and simulate all the problem involving the fluid-particle-interaction flows. [30]developed the splitting scheme for Eulerian-Lagrangian technique in the analysis of the particle. In another study, [31] proposed the Arbitrary Lagrangian-Eulerian (ALE) methods to solve and investigate the problem in fluid-structure-interaction. This method also used by others researchers such as [32], [33], [34] and [35].

According to the previous contribution, we found that many researchers and engineers study the problem in the fluid solver, fluid-particle-interaction solver and fluid-structure-interaction solver only but still not widely studied the Fluid-Particle-Structure-Interaction (FPSI) problem. Due to insufficient of that knowledge, the studies begin on the benchmark validation for fluid solver, fluid-particle-interaction solver, and fluid-structure-interaction solver. Later, come out with the validation framework for fluid-particle-structure-interaction (FPSI) solver.

©2006-2017 Asian Research Publishing Network (ARPN). All rights reserved.

METHODOLOGY

Currently, by validating the benchmark for fluid solver, fluid-particles-interaction solver and fluid-structures-interaction solver. Past years, Fluid-Particle-Structure-Interaction (FPSI) has been the very large thing to concern due to no validation benchmark data's. Therefore, all four phases of developing the validation framework of FPSI which is consists of fluid solver with the benchmark case study driven flow in a square cavity by Ghia, lid driven cavity by Tsorng for validation benchmark for fluid-particle-interaction solver, bluff body by Wall for validation benchmark for fluid-structures-interaction solver were discovered and lastly for FPSI, the case of granular boom sprayer design used as the reference to develop the validation framework for Fluid-Particle-Structure-Interaction (FPSI).

FLUID SOLVER

Fluid solver is the basic solver that must to consider initially before continue with the validate proses for Fluid-Particle-Structure-Interaction (FPSI). In this part, the driven flow in a square cavity by [7] used as the benchmark to validate fluid flow problem. This case study enable to be made as the validation reference due to this model is not complicated but complete with the characteristic in fluid flow problem. By using this case study, allowed to investigate the vortex formation of the primary and secondary vortices in the driven-cavity flow. From this investigation, we qualify to conclude the regions that have low or high velocity. In addition, we also enable to observe the interaction of fluid with boundaries wall and from that we obtained the streamline pattern of the interaction. However the streamline pattern of vortices is depending to the meshing size of the grid and Reynolds number.

FLUID-PARTICLE-INTERACTION SOLVERS

The validation process for fluid-particle-interaction solver capable to run parallel with the fluid-structures-interaction solvers validation. This process almost has the similar step with the validation for the fluid solver but the different between both processes are the particle elements.

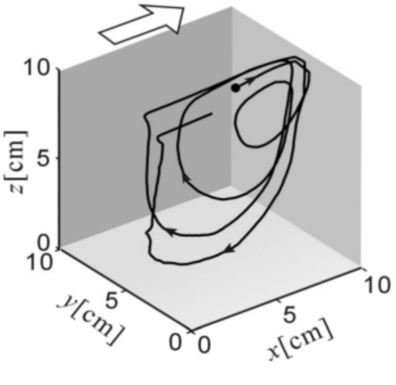

Figure-1. Three-dimensional trajectory of solid particle in lid-driven cavity flow for Re = 470, result from [36].

Particles response when submerged in fluid flow that has numerous applications to discover. Particles equivalence to fluid flow is a vital in various phenomena such as sedimentation, deposition, ventilation and waste management [30].Therefore a little complicated experimental setup to study particles behaviour submerged in a lid-driven cavity by [36] is referred as the benchmark to validate the fluid-particle-interaction solver in this work. Few works on two-dimensional simulations of similar arrangements show substantial difference due to the complexity of particles behaviour and three-dimensional inconsistencies by Tsorng et al. Figure-1 illustrates the experimental three-dimensional trajectory of a solid particle in lid-driven cavity by Tsorng for fluid Reynolds number 470.

FLUID-STRUCTURE-INTERACTION SOLVER

Hence, for fluid-structures-interaction solver used at two-dimensional flow over a thin elastic beam attached to a rigid and fixed square block and this work are popular among researchers [32], [33], [35], [37] as benchmarking to validate their works. This benchmark was proposed by[38] in order to test accuracy and robustness of newly emerging fluid-structures-interaction solver procedures. The problem setup is illustrated in Figure-2. The flow is driven by a uniform velocity of magnitude 51.3cm/s prescribed at the inflow. Lateral boundaries are assigned zero normal velocity and zero tangential traction. A zero traction boundary condition is applied at the outflow.

The fluid density and viscosity are set to 1.18×10−3 g/cm3 and 1.82 × 10−4 g/(cm s), respectively, resulting in flow at Reynolds number Re = 100 based on the edge length of the square block. The density of the elastic beam is 0.1g/cm3, and the Young's modulus and Poisson's ratio are 2.5 × 106 g/(cm s2) and 0.35, respectively.

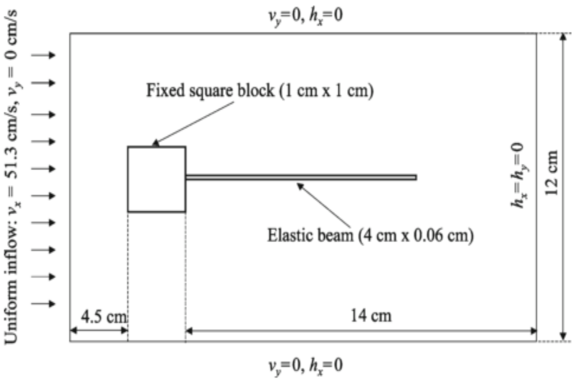

Figure-2. Flow over an elastic beam attached to a fixed square block [38].

FLUID-PARTICLE-STRUCTURE-INTERACTION SOLVER

In this work, the granular boom sprayer design used as the case study to performed the validation experiment. Granular boom sprayer is an example of applications that widely used in agriculture industries and have many problems when related to granular flow in the pipe like in the fluid, granular (particles) and pipe vibration (structure). Therefore, there will have continuously solution with exploit the granular flow in the pipe to improve the performance of the fluid flow, particles and structure. Concentration will be focus more on variable such as pressure, friction, vibration and many other variables proportional to the time.

Here, we introduce an experimental setup designed and performed to measure the amount of particle deposition and the structure analysis on the design of granular boom sprayer. In this experiment, the design must be considering in order to get an effective functional of granular boom sprayer. The most important design that needs to consider are the boom pipe and blow head which is the distribution of granular fertilizer is depends on the design of both of it. [39]focused on boom sprayer design consideration and their control variables are granular fertilizer flow from the inlet to the outlet. The analysis will be start with identify the task that number of blow head that need at the boom pipe will use for simulation in order to achieve the proper granular distribution with all variables or properties that needed.

In order to design pipe system for the boom sprayer machine, there are constraint side that the shape must be round and the design layout was influence by curve between pipes. However, the installation of the pipe at the machine frame also can influence the effect of sprayer. This is because the installation of the pipe in vertical position. Part that, it needs to consider the gravitational effect for manipulate it to fall down the material to the ground. The boom pipe as shown in Figure-3.

At the angle between the pipe corner or type of joint can effects the movement of granular material flow through the pipe. The granular materials will in-compressed so the cause of vibration between other is high and give direct impact to the material particle velocity. Furthermore, the kinetic energy will lost and effect the flow defection in the pipe. The flows shown that particle force to wall suddenly increase due to the bending. It will make the granular particle motion about to slow and defect the granular flow. To get the uniform distribution, the four blow heads can be adjustable. The blow head were assigned numbers, one to four in the order of the airflow directions but the last blow head without the deflecting plate.

The most critical part in design the boom sprayer machine is the blow head because it placed at final stage of process to distribute the granular fertilizer. Blow head is very different from nozzle that use for liquid fertilizer. Nozzle is controlled by the tip installed in it and manipulate by the pressure drop to get the spray type then achieve the uniform particle distribution [39]. Blow head used for granular fertilizer and not consist of any tip then it will directly drive by the force among the granular material itself. The blow head required some modification when the output results not in satisfy. The blow head in Figure-4 is simply mounts to the pipe as shown in Figure-3.

The blow head directly connect to the boom pipe that has 120º scattering angle and 185mm long. The opening in boom pipe is 35x35mm and the wide reflector is 35mm. The connecting plate was inserting into the boom pipe at 30º respectively to longitudinal direction of the boom pipe. Then the deflecting plate was a blunt edge formed by change a 0.6mm thick plate over the collecting plate at the boom pipe opening.

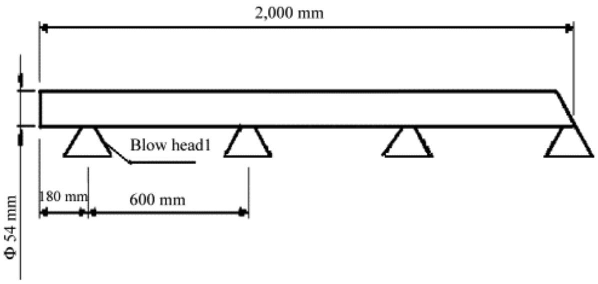

Figure-3. Placement of blow heads in a boom section [39].

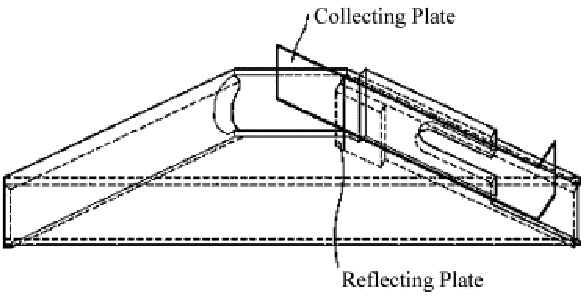

Figure-4. Blow head of the prototype granule applicator [39].

RESULTS AND DISCUSSIONS

Basis of Fluid-Particle-Structure-Interaction (FPSI) is the fluid solver, particle and structure distribution are only determined by the framework created by the fluid solver itself. Thus, in this Figure-5 shows the validation of framework for Fluid-Particle-Structure-Interaction (FPSI) solver flow which presents the fluid solver is the main phase of this work and the[7] procedure will be used as a validation.

Then, the validation of both fluid-particle-interaction and fluid-structure-interaction validation are managed parallel in one time due to the validation proses for both procedures or methods are not interrelated. These validations being utilized to assure the two ways coupling of the fluid-particle-interaction and structure use of the highest accuracy. Once both two ways coupling successful, the interaction of all these components must be used to evaluate the Fluid-Particle-Structure-Interaction (FPSI). Thus, to validate fluid-particle-interaction part, experiment of lid-driven cavity by[36] used while for fluid-structure-interactionwork followed the method by[38] as reference. Wall presented the flow over an elastic beam attached to a fixed square block as the model in his research and this model become are reference to many researchers.

At the final stage of this validation framework for FPSI solver, the granular boom sprayer design problem is used as the experimental apparatus to make the analysis. The granular boom sprayer problem has complete characteristic about FPSI problem where in this problem the fluid (air form blower), particle (granular fertilizer) and structure (design of pipe) are utilized together. Therefore, this problem is the most suitable case study that will be used as the benchmark to validate any works in the FPSI field.

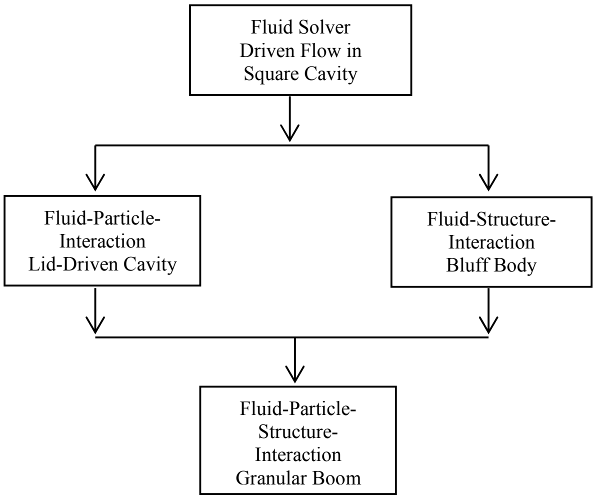

Figure-5. Validation framework for FPSI solver.

CONCLUSIONS

A validation framework for FPSI solvershas comprehensively presented in this paper. This broad procedure includes fluid solvers validation benchmark, fluid-particle-interaction validation benchmark, fluid-structure-interaction validation benchmark and last but not least the case study that will be used as validation benchmark for the FPSI solvers.Air-particle flows within a segmented passageway is used as the case study in this work and this selection is due to simplicity and comprehensiveness of the characteristic involved in this problem that include fluid, particle and structure elements. The focus of this paper which is to prepare overall procedure to validate FPSI solvers is clearly explained.

ACKNOWLEDGEMENT

This research is supported by the Fundamental Research Grant Scheme (FRGS), Grant No: FRGS/2/2013 Vot 1420 by Ministry of Education, Malaysia.

REFERENCES

[1] Chen X., McLaury B. S. and Shirazi S. A. 2004. Application and experimental validation of a computational fluid dynamics (CFD)-based erosion prediction model in elbows and plugged tees. Computers & Fluids. 33(10): 1251-1272.

[2] RamamurtR., GhiaU. and GhiaK. N. 1991. A semi elliptic analysis for 2D viscous flows through cascade configurations. Computers & Fluids. 20(3): 223–242.

[3] GhiaU., Ghia K. N., Rubin S. G., and Khosla P. K. 1981. Study of incompressible flow separation using primitive variables. Computers & Fluids. 9(2): 123-142.

[4] Ghia U., Ghia K. N., and Studerus C. J. 1977. Three-dimensional laminar incompressible flow in straight polar ducts. Computers & Fluids. 5(4): 205-218.

[5] DavisR. T., Ghia U., and Ghia K. N. 1974. Laminar incompressible flow past a class of blunted wedges using the Navier-Stokes equations. Computers & Fluids. 2(2): 211-223.

[6] Davis R. T., Ghia U., and Ghia K. N. 1974. Symmetric laminar incompressible flow past sharp wedges. Computers & Fluids. 2(2): 225-235.

[7] [7] Ghia U., Ghia K. N., and Shin C. T. 1982. High-Re solutions for incompressible flow using the Navier-Stokes equations and a multigrid method. Journal of Computational Physics. 48(3): 387-411.

[8] Osswald G. A., Ghia K. N., and Ghia U. 1991. Simulation of dynamic stall phenomenon using unsteady Navier-Stokes equations. Computer Physics Communications. 65(1): 209-218.

[9] Glowinski R., Pan T. W., and Periaux J. 1994. A fictitious domain method for Dirichlet problem and applications. Computer Methods in Applied Mechanics and Engineering. 111(3): 283-303.

[10] Pan T. W., Joseph D. D., and Glowinski R. 2005. Simulating the dynamics of fluid–ellipsoid interactions. Computers & Structures. 83(6): 463-478.

[11] Pan T. W., Chang C. C., and Glowinski R. 2008. On the motion of a neutrally buoyant ellipsoid in a three-dimensional Poiseuille flow. Computer Methods in Applied Mechanics and Engineering. 197(25): 2198-2209.

[12] Hao J., Pan T. W., Glowinski R., and Joseph D. D. 2009. A fictitious domain/distributed Lagrange multiplier method for the particulate flow of Oldroyd-B fluids: A positive definiteness preserving approach.

Journal of Non-Newtonian Fluid Mechanics. 156(1): 95-111.

[13] Glowinski R., Pan T. W., Hesla T. I., Joseph D. D., and Periaux, J. 1999. A distributed Lagrange multiplier/fictitious domain method for flows around moving rigid bodies: application to particulate flow. International Journal for Numerical Methods in Fluids. 30(8): 1043-1066.

[14] Pan T. W., Glowinski R., and Hou S. 2007. Direct numerical simulation of pattern formation in a rotating suspension of non-Brownian settling particles in a fully filled cylinder. Computers & Structures. 85(11): 955-969.

[15] Balakin B. V., Notøy I., Hoffmann A. C., and Kosinski P. 2012. The formation of deposit in a magnetic fluid: Numerical and experimental study. Powder Technology. 228: 108-114.

[16] Kosinski P. and Hoffmann A. C. 2005. Dust explosions in connected vessels: Mathematical modelling. Powder technology. 155(2): 108-116.

[17] Utkilen H., Balakin B. V., and Kosinski P. 2014. Numerical study of dust lifting using the Eulerian–Eulerian approach. Journal of Loss Prevention in the Process Industries. 27: 89-98.

[18] Balakin B. V., Hoffmann A. C., and Kosinski P. 2011. Experimental study and computational fluid dynamics modeling of deposition of hydrate particles in a pipeline with turbulent water flow. Chemical Engineering Science. 66(4): 755-765.

[19] Balakin B. V., Hoffmann A. C., Kosinski P., and Høiland S. 2010. Turbulent flow of hydrates in a pipeline of complex configuration. Chemical Engineering Science. 65(17): 5007-5017.

[20] Ilea C. G., Kosinski P., and Hoffmann A. C. 2008. Three-dimensional simulation of a dust lifting process with varying parameters. International Journal of Multiphase Flow. 34(9): 869-878.

[21] Kosinski P. 2007. Numerical analysis of shock wave interaction with a cloud of particles in a channel with bends. International Journal of Heat and Fluid Flow. 28(5): 1136-1143.

[22] Balakin B. V., Shamsutdinova G., and Kosinski P. 2015. Agglomeration of solid particles by liquid bridge flocculants: Pragmatic modelling. Chemical Engineering Science. 122: 173-181.

[23] Kosinski P., Hoffmann A. C., and Klemens R. 2005. Dust lifting behind shock waves: comparison of two modelling techniques. Chemical Engineering Science. 60(19): 5219-5230.

[24] Kosinski P., and Hoffmann A. C. 2010. An extension of the hard-sphere particle–particle collision model to study agglomeration. Chemical Engineering Science. 65(10): 3231-3239.

[25] Kosinski P., Kosinska A., and Hoffmann A. C. 2009. Simulation of solid particles behaviour in a driven cavity flow. Powder Technology. 91(3): 327-339.

[26] Kosinski P., and Hoffmann A. C. 2007. An Eulerian-Lagrangian model for dense particle clouds. Computers & Fluids. 36(4): 714-723.

[27] Kosinski P. 2008. Numerical investigation of explosion suppression by inert particles in straight ducts. Journal of Hazardous Materials. 154(1): 981-991.

[28] Kosinski P., and Hoffmann A. C. 2005. Modelling of dust lifting using the Lagrangian approach. International Journal of Multiphase Flow. 31(10): 1097-1115.

[29] Kosinski P. 2011. Explosion suppression by a cloud of particles: Numerical analysis of the initial processes. Applied Mathematics and Computation. 217(11): 5087-5094.

[30] Ngali M. Z. and Osman K. 2009. Splitting Scheme for Eulerian-Lagrangian Technique in the Analysis of Particle Conformity to Fluid Flow. Computer Modeling and Simulation, 2009. EMS'09. Third UKSim European Symposium on. pp. 333-338.

[31] Donea J., Huerta A., Ponthot J. P., and Rodriguez-Ferran A. 2004. Encyclopedia of Computational Mechanics Vol. 1: Fundamentals. Chapter 14: Arbitrary Lagrangian-Eulerian Methods.

[32] Dunne T., Rannacher R., and Richter T. 2010. Numerical simulation of fluid-structure interaction based on monolithic variational formulations. Fundamental trends in fluid–structure interaction, Contemp. Chall. Math. Fluid Dyn. Appl. 1: 1-75.

[33] Bazilevs Y., Calo V. M., Hughes T. J. R., and Zhang, Y. 2008. Isogeometric fluid-structure interaction:

©2006-2017 Asian Research Publishing Network (ARPN). All rights reserved.

theory, algorithms, and computations. Computational Mechanics. 43(1): 3-37.

[34] Legay A., Chessa J., and Belytschko T. 2006. An Eulerian–Lagrangian method for fluid–structure interaction based on level sets. Computer Methods in Applied Mechanics and Engineering. 195(17): 2070-2087.

[35] Habchi C., Russeil S., Bougeard D., Harion J. L., Lemenand T., Ghanem A., and Peerhossaini H. 2013. Partitioned solver for strongly coupled fluid–structure interaction. Computers & Fluids. 71: 306-319.

[36] TsorngS. J., Capart H., Lai J. S., and Young D. L. 2006. Three-dimensional tracking of the long time trajectories of suspended particles in a lid-driven cavity flow. Experiments in Fluids. 40(2): 314-328.

[37] Turek S., Hron J., Madlik M., Razzaq M., Wobker H., and Acker J. F. 2010. Numerical simulation and benchmarking of a monolithic multigrid solver for fluid-structure interaction problems with application to hemodynamics. Springer Berlin Heidelberg. pp. 193-220.

[38] Wall W. A. 1999. Fluid-struktur-interaktion mit stabilisierten finiten elementen.

[39] KimY. J., Kim H. J., Ryu K. H., and Rhee J. Y. 2008. Fertiliser application performance of a variable-rate pneumatic granular applicator for rice production. Biosystems engineering. 100(4): 498-510.