Bulgarian Journal of Agricultural Science, 21 (No 3) 2015, 703-710
Agricultural Academy

DETERMINATION OF HEAD LOSSES IN DRIP IRRIGATION LATERALS WITH CYLINDRICAL IN-LINE TYPE EMITTERS THROUGH CFD ANALYSIS

H. K. CELIK[1,3*], D. KARAYEL[1], M. E. LUPEANU[2], A. E. W. RENNIE[3] and I. AKINCI[1]
[1] *Akdeniz University, Dept. of Agricultural Machinery, Faculty of Agriculture, Antalya, Turkey*
[2] *The Politehnica University of Bucharest, Dept. of Manufacturing Engineering, Bucharest, Romania*
[3] *Lancaster University, Lancaster Product Development Unit, Department of Engineering, Lancaster, United Kingdom*

Abstract

CELIK, H. K., D. KARAYEL, M. E. LUPEANU, A. E. W. RENNIE and I. AKINCI, 2015. Determination of head losses in drip irrigation laterals with cylindrical in-line type emitters through CFD analysis. *Bulg. J. Agric. Sci.,* 21: 703–710

Drip irrigation techniques have been used in the agricultural production industry as an advanced water-saving irrigation method in recent decades. However advantageous this method, there are still some difficulties in projecting the efficiency of such systems. Most especially, determination of the losses because of the emitters is very important in order to set up an efficient irrigation system. This study is focused on determining head losses for agricultural drip irrigation systems with cylindrical in-line type drip emitters using Computational Fluid Dynamics (CFD) techniques. In the study, three-dimensional solid models of five in-line type drip emitters, which have a volumetric flow capacity of 4.0 [l h^{-1}] and placed in a pipe with 250 [mm] intervals between each other, were considered and CFD analyses were carried out with different inlet pressure values. According to the results of the CFD analyses, hydraulic losses were calculated globally and locally and analysis outputs were presented to determine the head losses due to the inclusion of the emitters. Validations for results of the simulation were also achieved by using empirical equations taken from related literature. As a result, maximum simulation error rate of 8.824 [%] was observed between simulations and empirical equations taken from related literature results. This accordance between simulation and empirical results can be interpreted that CFD analyses could be used to calculate the critical flow parameters such as total head losses of drip irrigation pipes integrated with in-line emitters.

Key words: Computer Aided Engineering, Computational Fluid Dynamics, Drip Irrigation, Drip Emitter, Hydraulic Losses

Nomenclature: A: Cross-sectional area [m^2]; D: Diameter [m]; f: Friction coefficient [-]; g: Acceleration due to gravity [9.81 m s^{-2}]; H_f: Head loss due to insertion of emitter [m]; H_k: Friction loss of the pipe [m]; L: Length [m]; n: Kinematic viscosity of water [m^2 s^{-1}]; P: Pressure [Pa]; Q: Volumetric flow rate [m^3 s^{-1}]; R_e: Reynolds Number [-]; t: Coefficient in head loss eq. [-]; v: Velocity [m s^{-1}]

Subscripts: i - Numerical index; *max*- Maximum; *min*- Minimum

Abbreviations: CAD - Computer Aided Design; CFD - Computational Fluid Dynamics

Introduction

Drip irrigation systems are used to uniformly distribute water in agricultural fields. The main device of a drip irrigation system is the drip emitter (Wei et al, 2006; Wang et al., 2006; Zhao et al., 2009). It is used to dissipate pressure and to discharge a small uniform flow or trickle of water at a constant rate at several points along a lateral. It is designed in such a way that the flow rate does not vary significantly

*Corresponding author: hkcelik@akdeniz.edu.tr

with minor changes in pressure across the lateral. The properties of emitters that play a vital role in designing a drip irrigation system are: discharge variation due to manufacturing tolerance; closeness of discharge-pressure relationship to design specifications; emitter discharge exponent; operating pressure range; pressure loss in laterals due to insertions of emitters; and stability of the discharge-pressure relationship over a long period of time. However, it is possible to investigate flow parameters with the aid of physical experiments in practice; it is not easy to predict the losses locally and flow behaviour of the fluid inside laterals and emitters with very good accuracy. To address this issue, numerical methods based flow simulations may be very useful for investigation of the flow conditions in the irrigation units.

Computational Fluid Dynamics (CFD) applications have been used to solve complicated fluid flow problems in a very wide range of engineering disciplines. Recent research has also shown that CFD applications could be very useful in agricultural engineering developments (Sun, 2002; Norton et al., 2007; Wang and Wang, 2006; Celik et al., 2010). The CFD simulations can be set up for internal and external flow conditions and the results can be used for predicting related flow parameters, and to design/re-design and optimise the flow path of the products virtually with good accuracy and without excessive design costs and time losses. It also helps to reduce prototype numbers and validation procedures.

In this paper, a CFD simulation approach was utilised to predict head losses for a sample irrigation piping system with cylindrical in-line type drip emitters. In the study, five of these emitters have been placed equidistantly in a pipe at 250 [mm] intervals. The flow behaviour of water has been simulated three-dimensionally and pressure losses calculated for the whole piping system and for each emitter at the after and before points so as to determine the emitters pressure losses locally. Results from simulations and empirical data taken from related literature have been compared.

Materials and Methods

Determining head loss

The head losses in drip irrigation laterals can be divided into two groups: the head loss due to friction and the head loss due to insertion of an emitter (local loss). The head loss due to friction can be calculated using the Darcy-Weisbach equation (Giles et al., 1995):

$$H_k = f \times \frac{L}{D} \times \frac{v^2}{2g} \quad (1)$$

where; H_k is the friction loss of the pipe [m], L is the pipe length [m], D is the pipe's internal diameter [m], v is the velocity [m s^{-1}], g is the acceleration due to gravity (9.81 [m s^{-2}]), and f is the friction coefficient. For smooth pipes, the friction coefficient is characterised by the Blasius equation as (Giles et al., 1995):

$$f = 64/Re \qquad (Re < 2100) \quad (2),$$

where: $Re = v\,D\,n^{-1}$ is the Reynolds Number and n is the kinematic viscosity of water (n = 1.01 x 10^{-6} at water temperature of 20ºC).

The schematic view of flow contraction and subsequent enlargement for on-line and integrated in-line emitters is shown in Figure 1 (Dutta, 2008):

Head loss due to the insertion of emitters can be calculated using the following equations (Giles et al., 1995; Keskin and Guner, 2007):

$$H_f = k \times \frac{v^2}{2g} \quad (3),$$

where; H_f is head loss due to insertion of emitter [m], k is the coefficient, v is the velocity of water [m s^{-1}], and g is acceleration due to gravity (9.81 [m s^{-2}]). Coefficient k can be expressed as:

$$k = 0.056 \times \left[\left(\frac{D_i}{D_g} \right)^{17.83} - 1 \right] \quad (4)$$

where; D_i *is* internal diameter of the pipe (m), and D_g is the internal diameter due to the emitter [m].

Total head loss at the ith emitter:

$$\Sigma H = (H_k + H_f) \quad (5)$$

Demir et al. (2004) developed a prediction model using dimensional analysis for friction losses in drip irrigation laterals Demir et al. (2004):

$$\Delta H_f = 0.007017\, Q^{1.728}\, D_i^{-1.224}\, \Delta L^{0.72}\, d_i^{-2.843}\, L_e^{0.027} \quad (6),$$

where: ΔH_f is the friction loss in emitter spacing [m], Q is the volumetric flow rate [m^3 s^{-1}], D_i is the pipe's internal di-

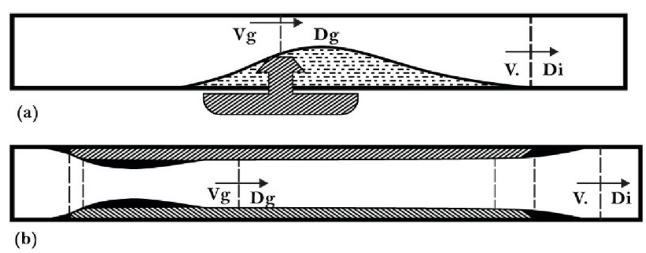

Fig. 1. Schematic view of flow for (a) on-line, (b) integrated in-line emitters

ameter [m], ΔL is the emitter spacing [m], d_i is the emitter's interior diameter [m], and L_e is the emitter length [m].

Standard flow volume rate equation is used to calculate flow velocity as below:

$$Q = A\,v \tag{7}$$

where: Q is volumetric flow rate [m^3 s^{-1}], A is cross-sectional area [m^2], v is velocity [m s^{-1}].

CAD modelling and CFD analysis

In the study, a solid model of the cylindrical type in-line emitter, which has a volumetric flow capacity of 4.0 [l/h], was created using SolidWorks Parametric Solid Modelling Design Software, and five of these emitters were placed in a drip irrigation lateral pipe. All dimensions of the models were reverse engineered from an original emitter and the assembly of the system was set up for CFD analysis. The general specifications of the emitter and pipe are shown in Figure 2.

SolidWorks Flow Simulation commercial CFD code was utilised to investigate the flow behaviour of the water in the modelled irrigation system. It was assumed that the system exists with five of the emitters placed in a straight irrigation pipe (without incline of the pipe). Four analyses were carried out for inlet pressures of 0.5, 1, 1.5 and 2 [bar] (gauge pressure) respectively. Identical boundary conditions were defined for all four analyses. Environment pressure - 101.325 [kPa] -Absolute Pressure - was defined for the emitter's outlet holes which have diameters of 2 [mm]. Fluid domain cell structure was defined by the default mesh function with the advanced channel refinement option in the CFD code. Total calculation domain is a rectangular volume of 17 x 17 x 1250 mm. Default automatic turbulence parameters were defined in the CFD code (Solidworks Product, 2010). In fact, although the simulations present very important data evaluation capabilities, it is quite difficult to get all real-life responses from the simulations due to the technological limitations and unpredictability of the material, fluid, dynamic environment conditions, etc. Therefore, boundary conditions for the system were also defined with some additional assumptions such as those given below:

- The water was assigned as a viscous and incompressible fluid at environment temperature of 20ºC in the simulation;
- Flow in the system has a steady boundary (the water head of a drip irrigation system usually maintains a steady value over the whole work period) (We et al., 2008);

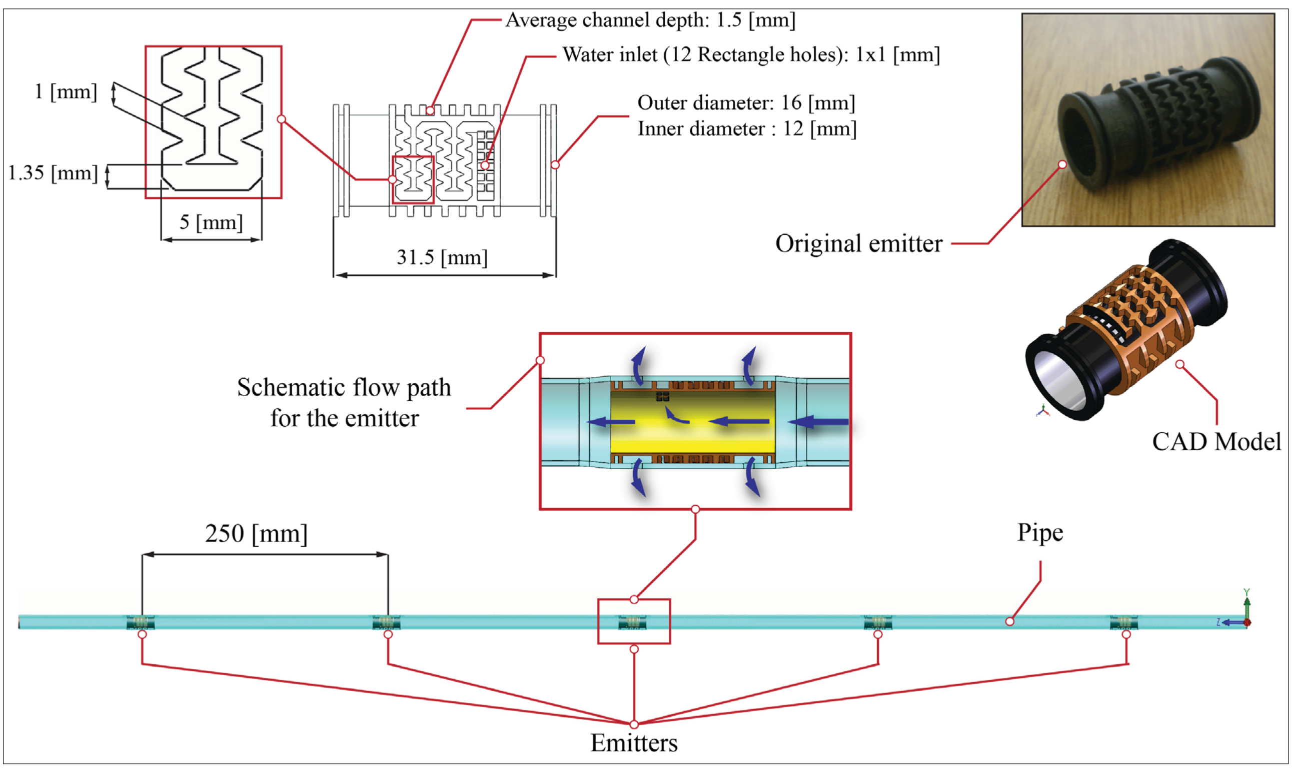

Fig. 2. General specifications of the emitter and pipe

- Standard Earth gravity (9.81 [m s^{-2}]) was considered;
- Surface roughness of the model walls was assigned as 0.01 [mm] (Palau-Salvador et al., 2004); All solid parts in the simulation are rigid (no physical deflection, no leakage of water).
- Boundary condition of the system is shown in Figure 3.

Results

After progressing the CFD simulation, visual and numerical outputs were obtained. The velocity and pressure magnitudes were calculated from 10 sections on the drip irrigation

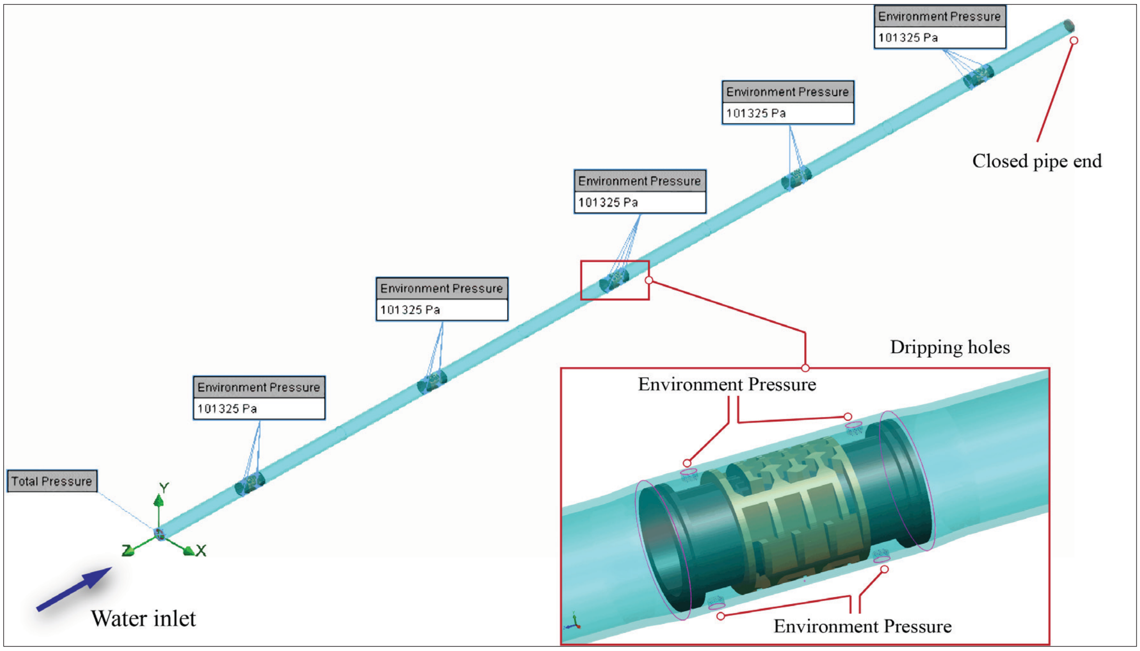

Fig. 3. Boundary conditions (Absolute Pressure)

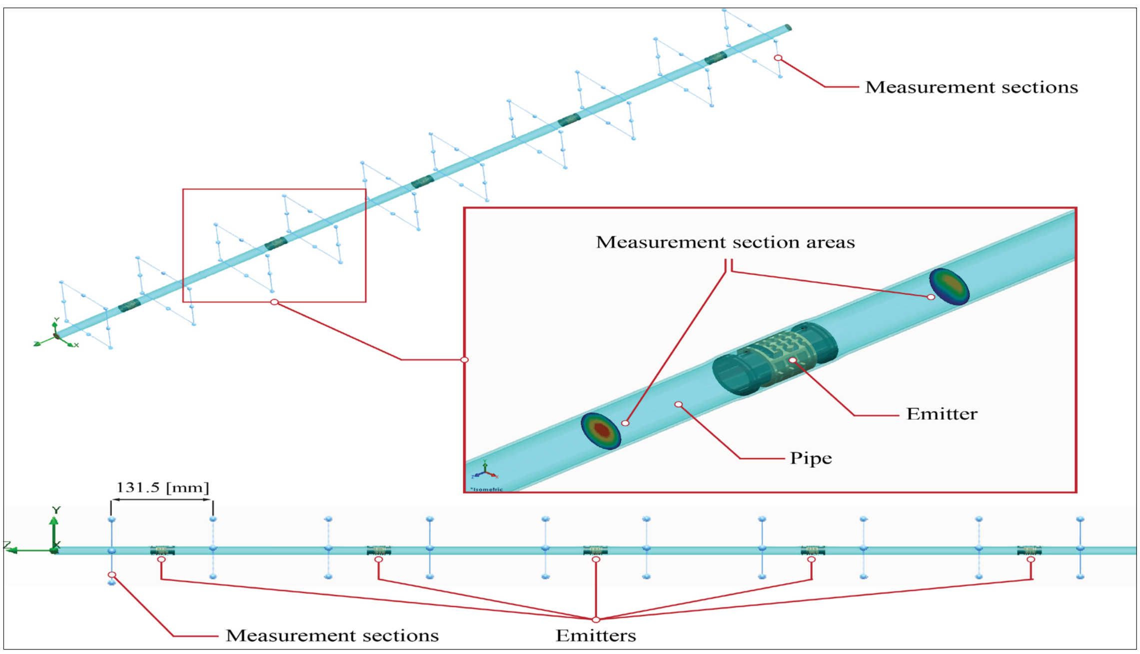

Fig. 4. Measurement section areas for velocity and pressure using CFD analysis

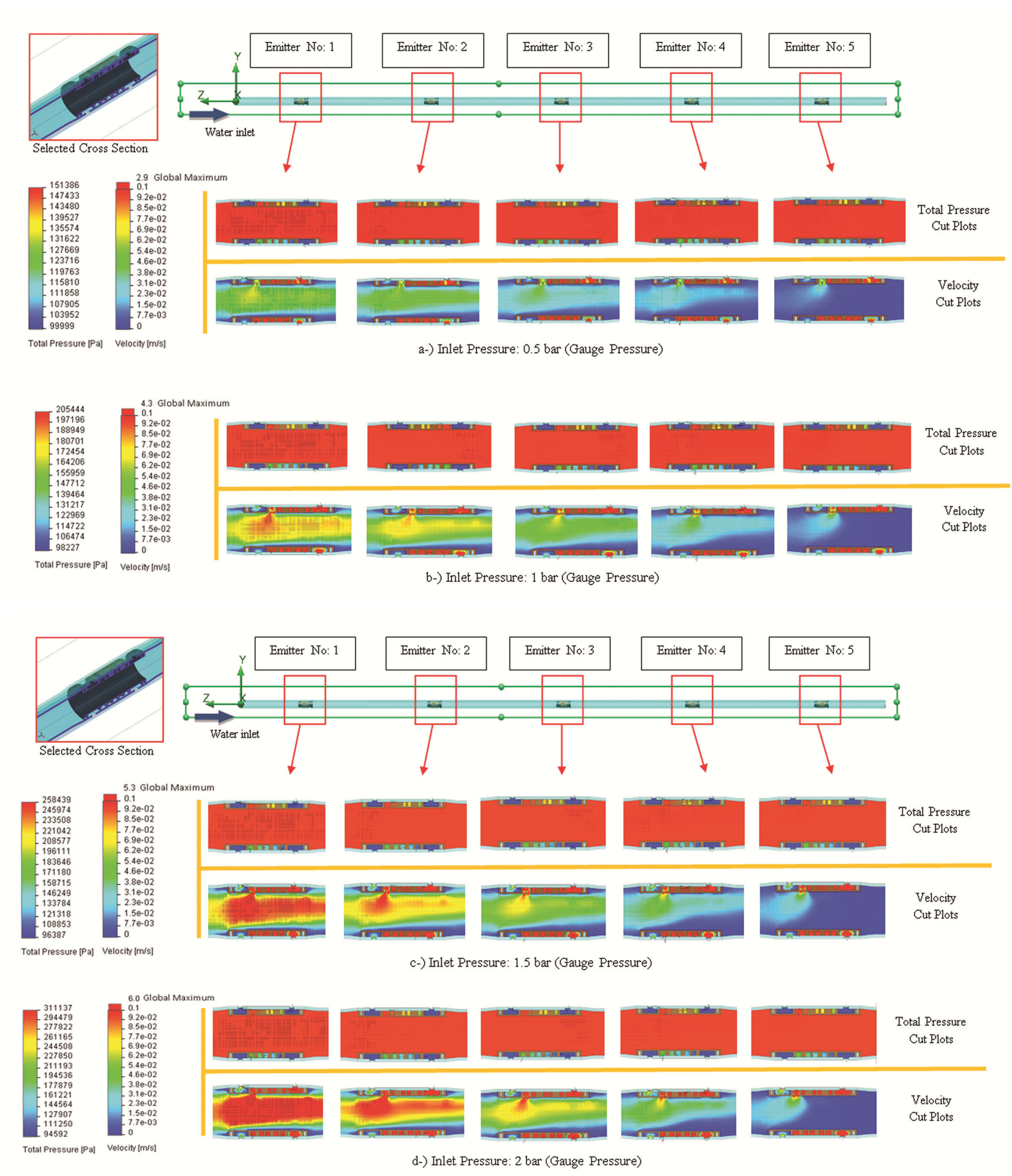

Fig. 5. Cross-sectional plots for total pressure and velocity

pipe (Figure 4). The cross sectional plots from the simulation for global total pressure and velocity are given in Figure 5 for each emitter, derived from different inlet pressures of water to evaluate the global fluid behaviour in the system. According to the CFD simulation results, total system global maximum velocities of 2.9, 4.3, 5.3 and 6 [m s^{-1}] for the inlet pres-

Table 1
Comparison of mean head loss due to friction and due to the insertion of the emitter

Inlet pressures, bar	Mean velocity in the Inlet Pipe, m s^{-1}	CFD			Empirical Equations				
		H_f, mm	H_k, mm	∑H, mm	Mean H_f, mm	Mean H_k, mm	Mean H_k, mm	∑H, mm	∑H, mm
					(Eq. 3)	(Demir et al.)	(Darcy-Weisbach)	(Eq. 3 + Demir et al.)	(Eq. 3 + Darcy-Weisbach)
0.50	0.020	0.079	0.105	0.105	0.085	0.028	0.028	0.113	0.113
1.00	0.030	0.158	0.048	0.206	0.163	0.049	0.051	0.212	0.214
1.50	0.035	0.254	0.056	0.310	0.269	0.068	0.071	0.337	0.340
2.00	0.040	0.305	0.079	0.384	0.334	0.081	0.082	0.415	0.416

Fig. 6. Comparison of the results obtained from empirical and CFD calculations

sures of 0.5, 1, 1.5 and 2 [bar] (gauge pressures) respectively were obtained.

Validation of the CFD analyses was conducted by making comparisons between the results from CFD analysis and empirical equations. Total head losses (friction + insertion of the emitters) were appointed as validation criteria, that are also aimed to be researched in this study, and they were calculated by considering results from CFD analysis and empirical equations.

The head losses due to friction and the insertion of emitters in laterals were determined using data taken from CFD analysis results and using the Bernoulli equation, given below:

$$\frac{v_1^2}{2g}+\frac{P_1}{\gamma}=\frac{v_2^2}{2g}+\frac{P_2}{\gamma}+H \quad , \qquad (8)$$

where: v is velocity [m s^{-1}], P is the pressure [Pa] and H is the head loss [m].

The head losses obtained from the CFD analysis were validated against the head losses calculated from equations derived by Demir et al. (2004) and Giles et al. (1995). In this process the numerical data are evaluated and mean of head losses of five emitters are presented in Table 1.

At the first sight, the absolute numerical values given in Table 1 looks like quite close each other, however, in the comparison process, simulation error in percent was considered to evaluate how to simulation approach converge to the empirical approach in percent. Therefore, the error rate between empirical data and CFD simulation was calculated according to the Equation 9 (Kurowski and Szabo, 1997). Calculated error rates are also illustrated in the charts given in Figure 6.

$$\text{Simulation Error} = \frac{\Sigma H_{Empirical} - \Sigma H_{CFD}}{\Sigma H_{Empirical}} x100 \ [\%] \qquad (9)$$

Calculated simulation error rates indicated that the errors show differences between the values of 2.830 % and 8.824 % at the pressure applications of 0.5, 1, 1.5 and 2 [bar] respectively. In some similar numerical simulation related literature, it is recommended that maximum differences between simulation and theoretical/experimental results should be less than 10% (Krutz et al., 1984; Sakakibara, 2008). Therefore it can be said that the correlation obtained in this study reflects this recommendation. This extraction lead us to say the CFD analysis which was set up in this study show a good accuracy in predicting pre-defined physical condition and this analysis approach can be used to predict accurately both the head loss due to friction and the head loss due to insertion of the emitter.

Conclusion

Engineering simulation technology and methods are now becoming more important in a very wide variety of engineering disciplines. It is obvious that the application of these engineering simulation techniques could be very useful within research in agricultural engineering applications. Therefore, as a part of advanced engineering simulations, a CFD approach was highlighted in this paper to determine the head losses in a sample drip irrigation system. CFD analysis simulation outputs which are constructed in the study presented a very good evaluation, which is very important to understand flow behaviour of the fluid in irrigation equipment. It is also observed that the numerical results taken from simulation and empirical calculations also have a good union with each other, which is very important for design validation applications of the irrigation equipment. In the case study, it is also observed that a good correlation between the CFD analysis and the empirical calculations exist. This correlation between simulation and empirical results can be interpreted that CFD analyses could be used to calculate the total head losses of drip irrigation pipes integrated with in-line emitters.

According to the study presented, some important points can be summarised as follows:

- CFD is complicated in regards to boundary and surface conditions of the wall such as surface roughness. Therefore, calculated results using CFD should definitely be verified. Therefore, the results of the CFD analysis verified by accepted empirical calculations available in the international literature. A good correlation was obtained between the CFD analysis and empirical calculations. This correlation can be interpreted that the CFD analysis could be used to determine both the head loss due to friction and insertion of emitters (local loss) with an acceptable accuracy.
- It can be recommended that CFD analyses could be used to determine head losses for the design of similar types of new emitters and irrigation equipment.
- CFD analyses may be used in the design studies for new types of emitters and the determination of optimum lengths of drip irrigation pipes.
- This research has improved the understanding of the hydraulic losses of the irrigation units and contributes to further research into the projection of drip irrigation systems enabled through the utilisation of advanced computer aided engineering tools.
- Three-dimensional CFD simulations for all kinds of irrigation systems can nowadays be performed without significant problems or excessive time investment. CFD is thus an excellent tool for undertaking detailed studies of complex flow situations. In the past, the CFD calculations were

limited to smaller systems due to the slower computational power available, but that problem is now gone. Larger systems involving hundreds of metres of pipe and different components can now be simulated within a reasonable amount of time.

Acknowledgements

This study was part-supported financially by the Scientific Research Fund of Akdeniz University.

References

Celik, H. K., D. Karayel , N. Cağlayan , A. E. W. Rennie and I. Akinci, 2010. Rapid Prototyping and Flow Simulation Applications in Design of Agricultural Irrigation Equipment: Case Study for a Sample Drip Emitter. 11th National Conference on Rapid Design, Prototyping & Manufacturing. ISBN 978-0-9566643-0-3, 12 June 2010, *Lancaster*, UK, pp. 181-191.

Demir, V., H. Yürdem and A. Değirmencioğlu, 2004. Development of a mathematical model using dimensional analysis for predicting the friction losses in drip irrigation laterals with cylindrical type in-line emitters. *Turk. J. Agric. For.*, **28** (1): 101-107.

Dutta, P. D., 2008. Characterization of Drip Emitters and Computing Distribution Uniformity in a Drip Irrigation System at Low Pressure under Uniform Land Slopes. MSc thesis, *Texas A&M University*, USA, 109 pp.

Giles, R. V., J. B. Evett and C. Liu, 1995. Fluid mechanics and hydraulics: Software and text. In: Schaum's Interactive Outlines, *McGraw-Hill Professional Publishing*, USA, 384 pp.

Keskin, R. and M. Guner, 2007. Su Cikartma Makinalari. *Ankara Universitesi*, Ziraat Fakultesi, Yayin, Ankara, Turkey, **1552**: 292.

Krutz, G, L. Thompson and P. Claar, 1984. Design of Agricultural Machinery. *John Wiley & Sons*, ISBN: 047108672, 473 pp.

Kurowski, P. and B. Szabo, 1997. How to find errors in finite element models. *Machine Design*, **9**: 93-98.

Norton, T., Da-W. Sun, J. Grant, R. Fallon and V. Dodd, 2007. Applications of Computational fluid dynamics (CFD) in the modelling and design of ventilation systems in the agricultural Industry, a review. *Bio-resource Technol.* **98**: 2386-2414.

Palau-Salvador, G., J. Arviza-Valverde and V. F. Bralts, 2004. Hydraulic flow behaviour through an in-line emitter labyrinth using CFD techniques. In: *ASAE/CSAE* Annual International Meeting, Ottawa, Ontario, Canada, paper No. 042252, pp 1-9.

Sakakibara, N., 2008. Finite Element in Fracture Mechanics. Lecture notes, *University of Texas*, Austin.

Solidworks Product, 2010. Flow Simulation Online User's Guide File.

Sun, Da-W, 2002. Computational Fluid Dynamics (CFD): A design and analysis tool for the agri-food Industry. *Computers and Electronics in Agriculture*, **34**: 1-3.

Wang, F. J. and W. E. Wang, 2006. Development and issues of CFD Analysis researches on flow path of emitters. *Transactions of the CSAE*, **22** (7): 188-192.

Wang, W., F. Wang and F. Zhao, 2006. Simulation of unsteady flow in labyrinth emitter of drip irrigation system. In: Computers in Agriculture and Natural Resources, 4th World Congress Conference Proceedings 24-26 July 2006, Florida, USA, pp. 630-635.

We, Q., G. Lu, L. Jie, Y. Shi, W. Dong and S. Huang, 2008. Evaluations of emitter clogging in drip irrigation by two phase flow simulations and laboratory experiments. *Computers and Electronics in Agriculture*, **63**: 294-303.

Wei, Q., Y. Shi, W. Dong and S. Huang, 2006. Advanced methods to develop drip emitters with new channel types, ASABE. *Appl. Eng. Agric.,* **22** (2): 243-49.

Wei, Q., Y. Shi, W. Dong, G. Lu and S. Huang, 2006. Study on hydraulic performance of drip emitters by computational fluid dynamics. *Agr. Water Manage*, **84**: 130-36.

Zhao, W., J. Zhang, Y. Tang, Z. Wei and B. Lu, 2009. Research on transitional flow characteristic of labyrinth channel emitter. Computer and Computing Technologies in Agriculture II, *International Federation for Information Processing* (IFIP), *Springer*, Boston, **294**: 881-890.

Received November, 3, 2014; accepted for printing February, 20, 2015.