American Journal of Engineering Research (AJER)
e-ISSN: 2320-0847 p-ISSN : 2320-0936
Volume-6, Issue-6, pp-67-79
www.ajer.org
Research Paper Open Access

A Validation Method of Computational Fluid Dynamics (CFD) Simulation against Experimental Data of Transient Flow In Pipes System

ThankGod E. Boye, [1] Thaddeus C. Nwaoha[2] Samuel D. Olusegun[1], Festus I. Ashiedu[1]

[1](Department of Mechanical Engineering, Federal University of Petroleum Resource, Effurun, Nigeria)
[2](Department of Marine Engineering, Federal University of Petroleum Resources, EffurunNigeria)

ABSTRACT: *Computational fluid dynamics (CFD) has become a significant engineering and technology design tool for modelling and analysis of incompressible and compressible flows in pipes. CFD has gained relevancy by providing cost-effective means of simulating real flows by the numerical solutions of the governing equations of fluid dynamics. CFD is largely depended on currently in taking decisions in fluids flows scenarios either closed or opened channels. Transients flow phenomena occurrence in oil, gas and water pipelines is inevitable due to some complex pipes network set-up, the sudden closure of control valves or unexpected failure of mechanical systems such as pumps or compressors as the case may be. Using CFD software to simulate transient's flow in pipeline could be demanding for effective and efficient predictions. This research paper is aimed at a method of CFD simulation results validation against published experimental transient data. CFD software such as applied flow technology (AFT) Impulse 4.0 was adopted as a case study for this research paper. The experimental data and boundary conditions were closely maintained to perform the transient's simulation on AFT Impulse 4.0 workspace. Results obtained from the CFD simulation of two transients' flows scenarios were validated with reasonable convergence with some published experimental data from the Hydraulic Engineering Research Institute (Institute cerceteriHidrotehnice –I.C.H.). It was a novelty that validation of published experimental data against CFD simulated results establishes cost-effectiveness and significant time reduction in testing and evaluation flow predictions in pipe systems at the design stage. Thus, this CFD software validation method against experimental results showcasesthe practicability, applicability and acceptability of estimating the credibility of fluid modelling commercial software.*

Keywords: *CFD, Validation Method, Transient Flow, AFT Impulse Software, Pipe, and Experimental published data*

I. INTRODUCTION

Increasing demands of oil and gas products for the global energy consumptions and its exploration in deep offshore and Arctic region have necessitated accurate prediction flow in a pipe system. Computational fluid dynamics (CFD), as a tool, can be used to improve safety in pipe system during design and reduce risk to human life, plant and environment. The transportation of fluid, for example, crude oil from recovery fields to storages or refinery in the most case depends on complex pipeline network systems which usually in hundreds of miles or kilometers of distance. They are equipped with sophisticated control and monitoring facilities such as control valves, pump etc. because of the likelihood of hydraulic transients (water-hammer) phenomena occurrence, which is practically inevitable in sure systems [1]. Hence, CFD is a useful tool to mitigate the aforementioned challenges.

Nomenclatures

C	Rapid closing valve
D	Pipe diameter (m)
f	Friction factor
g	Acceleration due to gravity (m/s^2)
H	Specific enthalpy of gas (kj/kg)
*H**	Supply reseviour
P	Pressure (atm)
*P**	Pump
q	Heat transfer rate through pipe wall into, per unit volume
q_o	Flow rate (l/s)
R	Receiving reseviour
u	flow velocity (m/s)
SPP-DA	Surge pressure protector (air device)
T	Induce Transient
t	Time (s)
VR	Receiving reseviour
x	Distance along the axis of the pipe (m)
y	Elevation from defined datum (Positive in the opposite direction of the gravity)

Greek Symbols

ρ	Density (kg m-3)

Abbreviations

AFT	Applied Flow Technology
CFD	Computational Fluid Dynamics
ICH	Institute cerceteriHidrotehnice ,
IUPAC	International Union of Pure and Applied Chemistry
MoC	Method of Characteristics

There are many important factors needed to be considered before a pipeline is designed and constructed there are: the terrain, fluid properties, and cost, among others. However, [1] argued that a flow requirement is a key factor. Hence, before a pipe diameter, pump or compressor unit size etc. can be chosen, a great deal of consideration must be given to the supply and demand flow builds up and their variations. In addition, the hydraulics of that pipe system must be given adequate consideration. In view of employing a robust modelling method such as transient's simulation (computer code), that is capable of taking into account of unsteady flow conditions, can be considered as the primary step of ensuring safety in any offshore or onshore pipeline systems.

A review of research work done indicated that insufficient published work in literature has been reported on the above subject matter. Hence the study is targeted to validate the experimental study undertaken at Hydraulic Engineering Research Institute (Institute cerceteriHidrotehnice – I.C.H.) laboratory pipes system [2]. The validation method reported therein would assist in handling specific flow problem in pipes and providing accurate prediction. Besides, it will remove bottleneck associated with a commercial computer code.

This research paper aims at and focuses on a method of CFD simulation results validation against a published experimental transient's data extracted from Hydraulic Engineering Research Institute (Institute cerceteriHidrotehnice – I.C.H.) laboratory pipes system [2]. This validating method employed in the research work proffers a practical approach to handling flows problems especially were the need of using a commercial computer software to study specific fluid flows problems is mandatory. Where by the utilisation of a selected fluid flows commercial computer software solutions are uncertain or could be judged, the need for validating the selected commercial computer software ingenuity arises. This CFD software validation method is considered bridging the gap for cost effective and timely approach in predicting the degree of accuracy of computer software results provided; the primary or secondary reference standards adopted are empirically formulated.

II. TOOL, PROCESS DESCRIPTION AND METHOD

The tool used for the research work includes Applied Flow Technology software (AFT Impulse 4.0), published transient's experimental test data and a high-speed processor computer. The tasks include both computer-based modelling and design supported by published experimental test data. The experimental test data represents a reference standard in order to validate the credibility of the computer software (AFT Impulse 4.0), as a case study. It is employed in the transients' studies of crude oil pipes system [3]. The AFT Impulse 4.0 software validation simulation outcomes became a significant evidence to decide its credibility. In order, facilitate generic design rules of hydraulic transients' control and mitigation of a crude oil transportation pipeline system.

Presented in figure 1 is a CFD transient simulation flow chart. The flow chart consists of fourteen major steps. Seven of the steps are major steps while the others are complementary to the others. The first major step adapt transient of experimental data. In the second step, a validation of CFD software to all experimental data is carried out. The third stage is the analysis of validation results. The fourth entails designing a model for crude oil transportation. In the fourth step, the computer model developed is used to predict steady and transient flow in a crude oil pipeline. The fifth step is control of transient flow predicted in the new model. While the sixth step is associated with the exploration of a new crude oil pipeline model on different transient events. In the last step, the analysis result is assigned to develop generic design rules in order to mitigate and control transient flow in a pipe system.

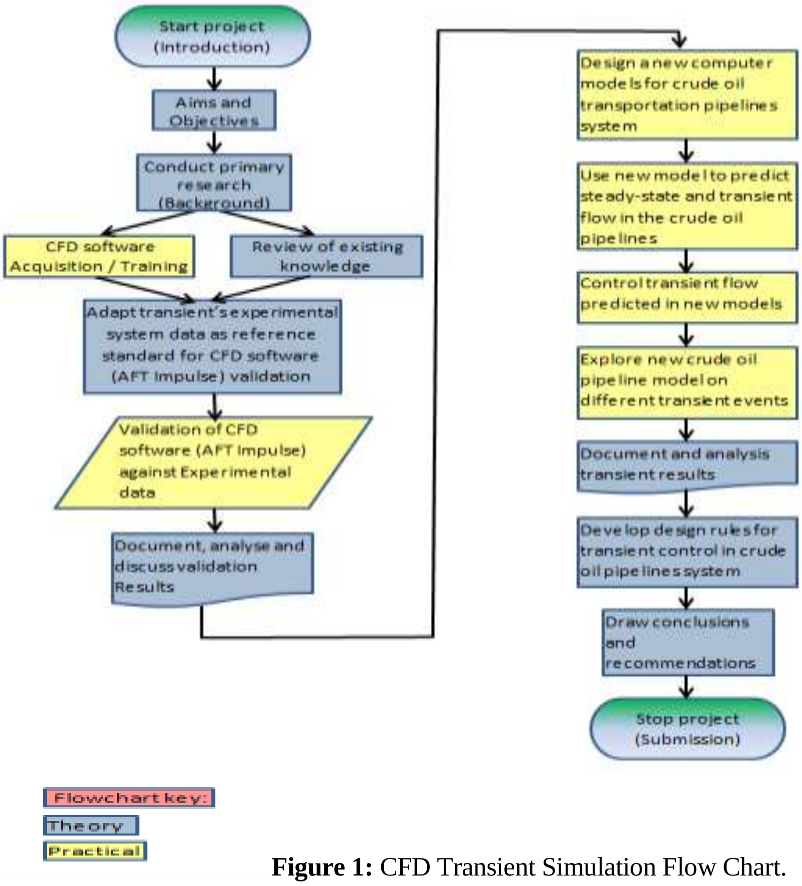

Figure 1: CFD Transient Simulation Flow Chart.

2.1. Theoretical Fundamentals of Computer Modelling of Hydraulic Transients

The following equations are generally utilized for computer-based hydraulic modelling for transient flow in a pipe. The time-dependent, one-dimensional flow of a fluid in an inclined conical conduit, as shown in figure 2 can be described by three equations representing conservation of mass, momentum, and energy [1]. The transient flow software solves these conservation laws of transport phenomena simultaneously with the discretized a computer.

Conservation of mass (Continuity):

$$\frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(\rho u) = 0 \qquad 1$$

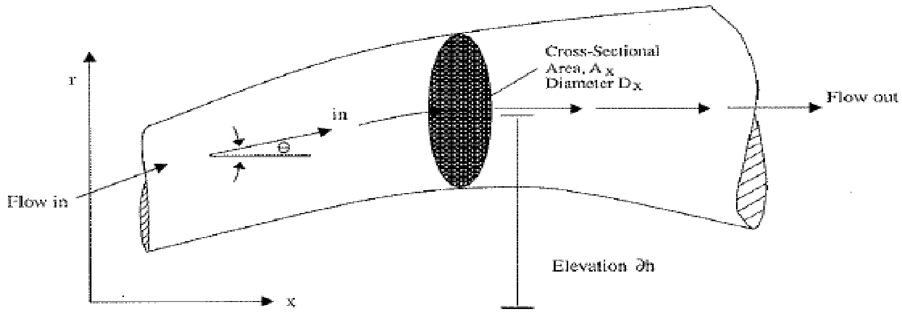

Figure 2: Fluid flow through non-uniform inclined cylindrical pipe segment [1].

Conservation of momentum:

$$\frac{\partial\rho}{\partial t}(\rho u)+\frac{\partial}{\partial x}(\rho u^2+P)+\rho g\frac{dy}{dx}+\frac{4fu|u|}{2D}=0 \qquad 2$$

Conservation of energy:

$$\frac{\partial\rho}{\partial t}\left[\rho\left(h-\frac{P}{\rho}+\frac{u^2}{2}\right)\right]+\frac{\partial}{\partial x}\left[\rho u\left(h+\frac{u^2}{2}+gy\right)\right]+q=0 \qquad 3$$

2.1.1. Solution Technique

For an operating pipeline in nonisothermal situations, Equation 1 and 2 are usually solved simultaneously. To consider heat transfer across the fluid and environment, separate heat transfer equations can be utilized to solve for temperature variation along the pipeline such as equation 3 [1].

2.2. CFD transient's validation

First, the CFD software AFT Impulse 4.0 is used and boundary conditions of an experimental pipe system are defined in its graphical user interface (GUI) platform. The laboratory experimental set-up of the transient experiment is built on CFD software GUI workbench. The outcomes of the simulation will determine if the CFD pipe flow simulation software could rely on as a credible design tool and it could be used to take engineering design judgements. AFT Impulse 4.0 is used as a case study to demonstrate this type of validation method of commercial CFD software, because it's considered to have the capabilities to handle this specific problem of transients flows in pipes system and its unique ability to accommodate complex physical boundary conditions of pipes system and various pipe sections on its graphical users interface platform.

2.2.1. Software Validation against Published Experimental Test Data

According to Applied Flow Technology AFT (2007) [4], AFT Impulse 4.0 Water-hammer user guide, the 'AFT Impulse is a graphical platform for modelling water-hammer and surge transients in pipe networks'. It's advanced Windows graphical interface simplifies the complex process of building water-hammer models. The transient solutions engine employs the proven and well understood Method of Characterises (MoC) to solve the fundamental equations of water-hammer. The MoC converts partial differential equations of motion and continuity transport phenomena into four first-order equations represented in finite differential form and solved simultaneously with a computer [5]. Aljanabi [6] also utilised the MoC to perform numerical modelling of transient flow in long oil pipeline system and find out that MoC methods are capable of accuracy solving for transient pressure and flow in oil pipe system, including the effect of pipe friction:

Engineering assumption in AFT Impulse 4.0

- Liquid flow and in One-dimension flow
- No chemical reactions
- Wave speed remains constant during transients
- Non-condensable gas release is negligible
- Bubbles that form during transient cavitation are not moving.

The above assumption is in conformity with possible assumption with governing equations of computational transport phenomena.

2.2.2. Rationale of theValidation Method

In order to validate the credibility of commercial transient’s solver, a published experimental test data is required for the validation purpose. One of the reasons for preferring this validation method is due to the uniqueness in each case of transient flow phenomena data. A slight change in physical components of pipes system such as valve position, pipe elevation etc. undermines any previous transient’s analysis done for that system [1, 6].

Therefore, software validation against published experimental results such as laboratory studies or field experimental results is considered to be a practical and acceptable method use to estimate the validity of most computer modelling transient's software[3]. Denton [7] reviewed methods of validation for CFD simulation of highly transients flow in pipeline system extensively; similarly [8] utilized numerical method of validation extensively for the validation of transient flow of two-phase flow models for simulation of slug flow in pipelines. However, if the validated software results converge to the experimental results, this will build reasonable confident on the simulation results generated by the validated software when used for any other hydraulic transients model design and studies as a credible innovative engineering and technology design tool.

2.3. Experimental Installation and Measuring Devices

This is the laboratory experimental set-up for the studying of unsteady flow of comparative large scales, designed and built at the Hydraulic Engineering Research Institute (InstitutulCerceteriHidrotehnice – I.C.H.) [2], shown in figure 2, this laboratory experimental set-up is replicated in AFT Impulse 4.0 GUI workbench.

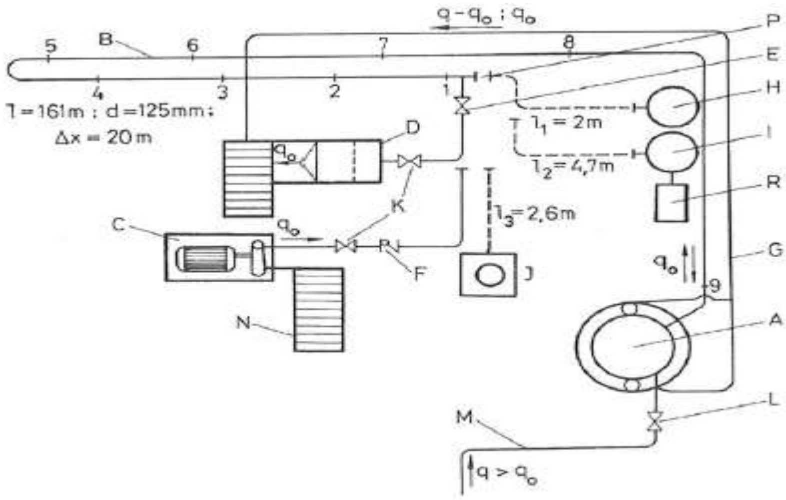

Figure3: General Scheme of the (ICH) laboratory experimental Installation for the study of non-stationary motion inside hydraulic surge systems [2].

2.3.1. Description of the Experimental Setup and Physical Boundary Conditions

The following describes the experimental set-up and operation conditions (Scheme II) in which the experiments were conducted [2] that formulated the published test data. Some of the relevance operational data, physical boundary conditions and components in the main devices on the installations are captured in *Appendix A.*

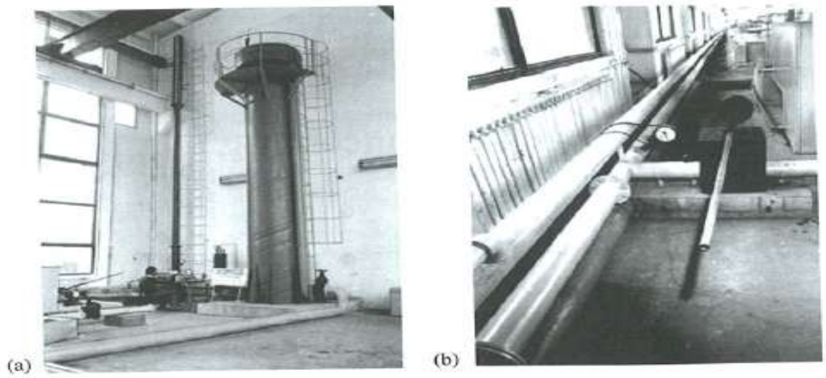

Figure 4:(ICH) Laboratory experimental installation for the study of unsteady motion inside hydraulic surge – overall view: (a) supply chamber, protection system; (b) surge pipe, valve, measure chamber [2].

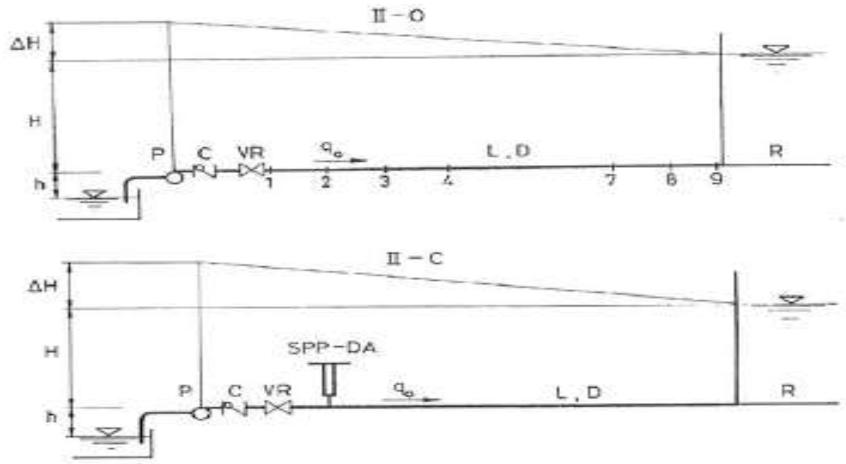

Figure 5: Hydraulic scheme II longitudinal section in the variants with and without protective devices respectively [2].

2.4. The Computer Model Version of the Laboratory Experimental Set-up

The computer model version built in AFT Impulse 4.0 workbench is simplified according to the longitudinal view of that experimental system see (figure 5). However, details of the installed experimental set-up are given in [2]. In order to get reasonable convergence of results, adequate assumptions were made such as the type of rapid closure valve that was used, the pipe wall thickness, pipe friction (internal surface of the pipe wall) and the wave speed. The following figures (6 - 7) show the computer models version of the experimental systems without any protection device and the other with protection device respectively.

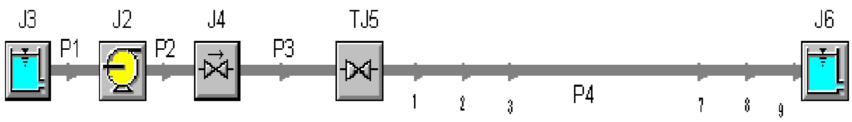

Figure 6:AFT Impulse model version of the experimental set-up (Scheme II) longitudinal section without any protection device.

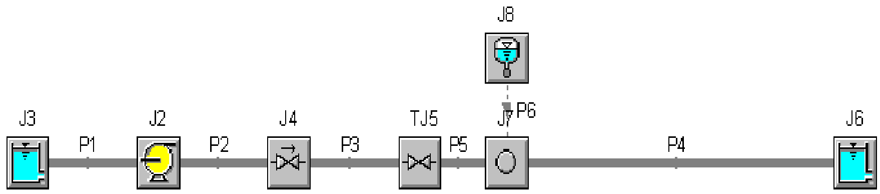

Figure 7:AFT Impulse model version of the experimental set-up (Scheme II) longitudinal section with protection device.

Table1. Relationship between AFT Impulse model version and Experimental Set-up

Junction	Experimental Set-up [2]	AFT Impulse Model
Pump	P*	J2
Supply reseviour	H	J3
Check valve (Non-return)	C	J4
Rapid closing valve	VR	J5
Receiving reseviour	R	J6
Branch (connector)		J7
Protective device	SPP-DA	J8

Both computer models have the some operating and boundary conditions in reference to the laboratory experimental installation. The computer models are made up of flow components such as junctions and pipesincluding reservoirs, pump, check valve, rapid closing valve and protective device (air device) and a connection branch. Table 1 showed the relationship between the AFT Impulse 4.0 models version in figure 6 and 7 and that of laboratory experimental set-up shown figure 5.

The transient *(T)* event of the computer model is time-based and it was induced by a rapid closing valve closure *(J5)* in less than *0.1 sec*. the type of rapid closing valve in the experimental installation was not specified as well as the exact time period of the valve closure. Hence, some commonly used valves such as ball and gate valves are assumed and used for the computer models. The valve closure time was stated to be less than 0.1seconds. But the worst case scenario is considered for the rapid closure valves of the transient simulation in the computer model.

Times of 0.095 secondsis used to close the valve to induce transient *(T)* in both of the computer models in order to investigate the convergence of the computer model results to that of the experimentally measured and published in [2]. The pipes are of the same diameter of 0.125m (5 Inch.), the pump supply the water to the system at the flow rate of 19.21 l/sec. The pipe elevation of the experimental installation was not stated; ball and gate valves were assumed on the computer model simulation.

The specify detail of the water properties used for the laboratory experimental study was not specified, for example, the temperature was not stated. However, standard properties of water at 1 atmosphere are assumed for the computer model simulation, however, the International Union of Pure and Applied Chemistry (IUPAC) is used as a reference point.

2.5. Published Experimental Study Results

Experiment results published can be reliable for the need for computational models validation. It should be stressed that computer-based models require validation using reliable experimental data before they can be put to good use. To this end, the following figures show hydraulic transient results of the laboratory experimental studies of Scheme II with protection and with protection device. The numeric data used in helping to validate this CFD transient software were extracted from the graphs below.

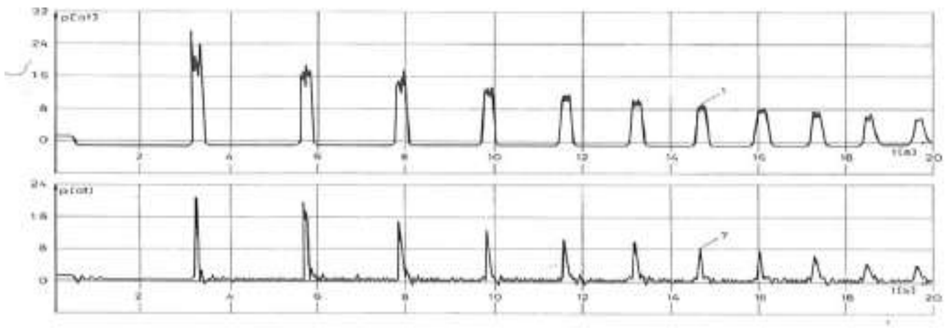

Figure 8:II-C Scheme of the of the hydraulic circuit: simultaneous Pressure measurements at point 1 and 7 on the pipe *(at q_o = 19.21 l/s)*[2].

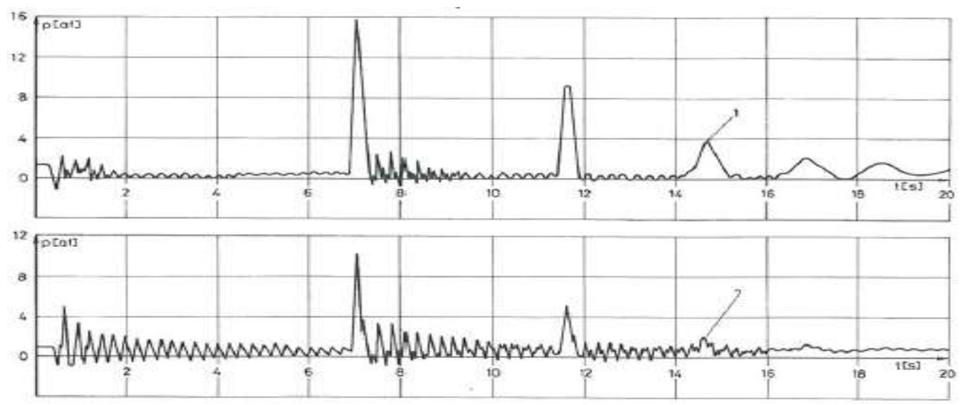

Figure 9: II-C Scheme of the hydraulic circuit equipped with air devices (DA): Pressure measurements point 1 and 7 on the pipe *(at q_o = 19.21 l/s)*[2].

III. RESULTS

The results showcased on graphs in this section are flow profiles in pressure against time generated from AFT Impulse 4.0 transient (water hammer) software used to simulate the transient flow in figure 7 and 8. All known physical and boundary condition of the laboratory experimental set-up shown in figure 4 and 5 are maintained in the computer version (figure 6 and 7) of the system. Thus, adequate assumptions are also made before the transient flow in the pipe simulation. Ball and Gate valve were the two type of valves adapted for the rapid closing valves.

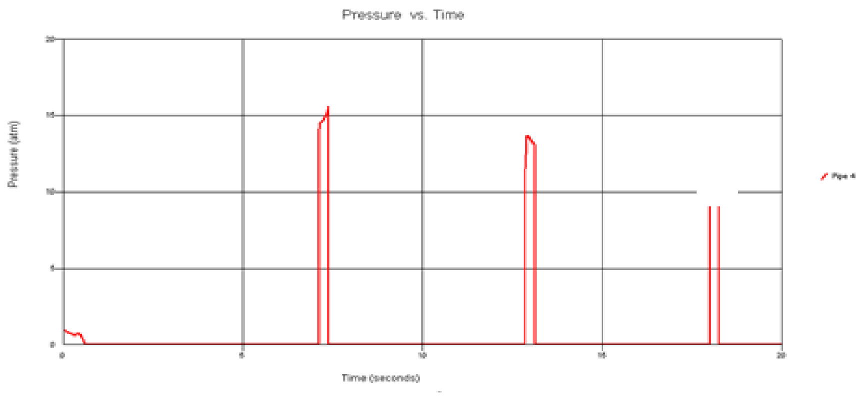

Figure 10: Computer model of the hydraulic circuit: simultaneous pressure measurements at point 1 on pipe 4 without protection in figure 6 *(at q_o = 19.21 l/s Ball Valve, Rapid closing valve time 0.095 sec.)*

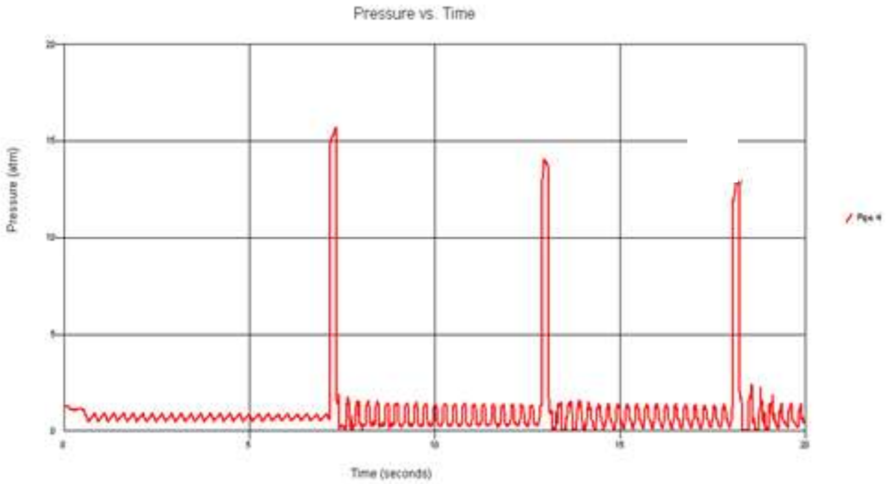

Figure 11: Computer model of the hydraulic circuit: simultaneous Pressure measurements at point 7 on pipe 4 without protection in figure 6 *(at q_o = 19.21 l/s Ball Valve, Rapid closing valve time 0.095 sec.)*

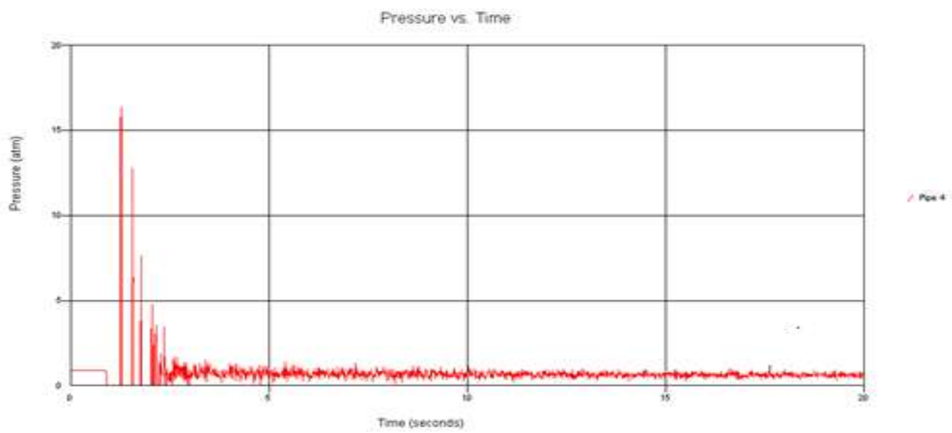

Figure 12: Computer model of the hydraulic circuit: equipped with air devices (DA): Pressure measurements at point 1 on pipe 4 in figure 7 *(at q_o = 19.21 l/s Ball Valve, Rapid closing valve time 0.095 sec.)*

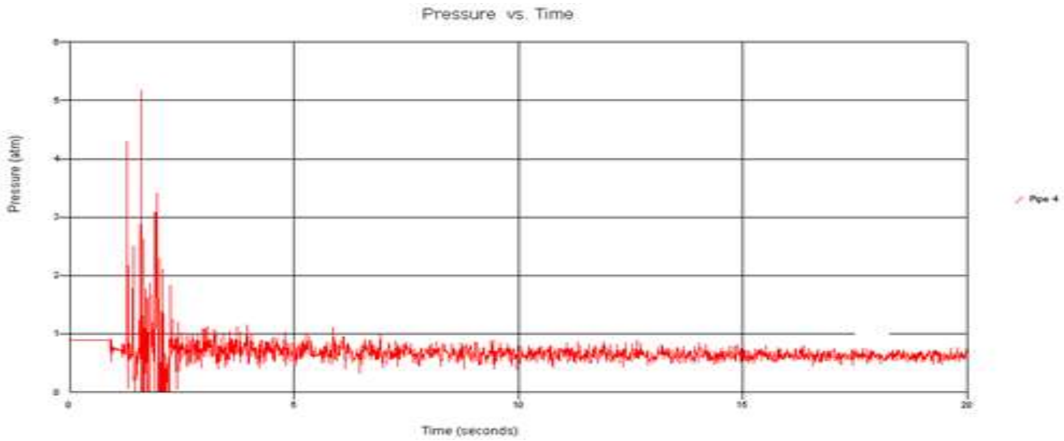

Figure 13: Computer model of the hydraulic circuit: equipped with air devices (DA): Pressure measurements at point 7 on the pipe 4 in figure 7 *(at q_o = 19.21 l/s Ball Valve, Rapid closing valve time 0.095 sec.)*

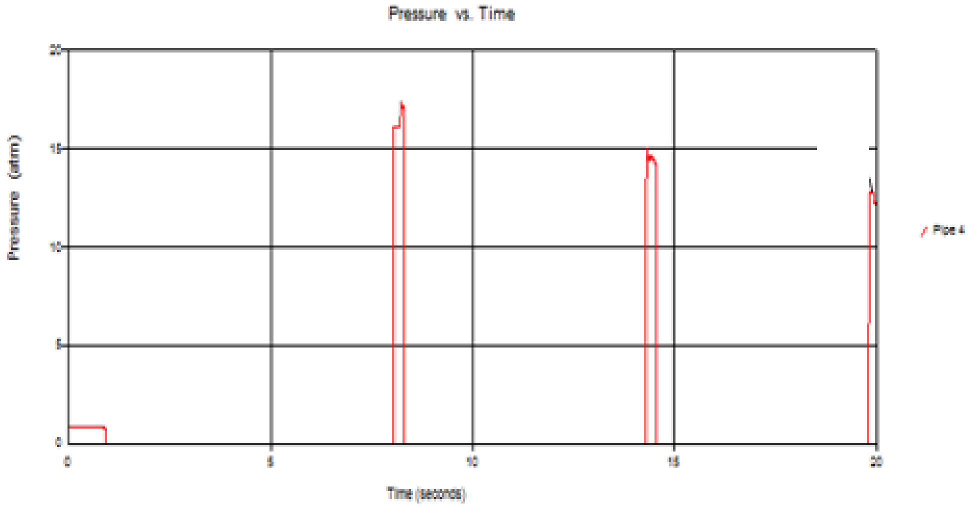

Figure 14: Computer model of the hydraulic circuit: simultaneous Pressure measurements at point 1 on pipe 4 in figure 6 *(at q_o = 19.21 l/sGate Valve, Rapid closing valve time 0.095 sec.)*

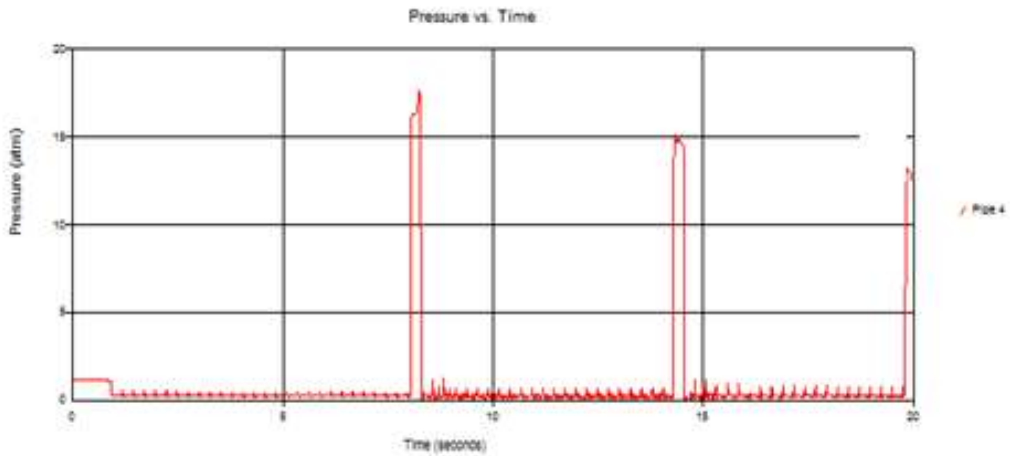

Figure 15: Computer model of the hydraulic circuit: simultaneous pressure measurements at point 7 on pipe 4 in figure 6 (*at q_o = 19.21 l/s,Gate Valve, Rapid closing valve time 0.095 sec.)*

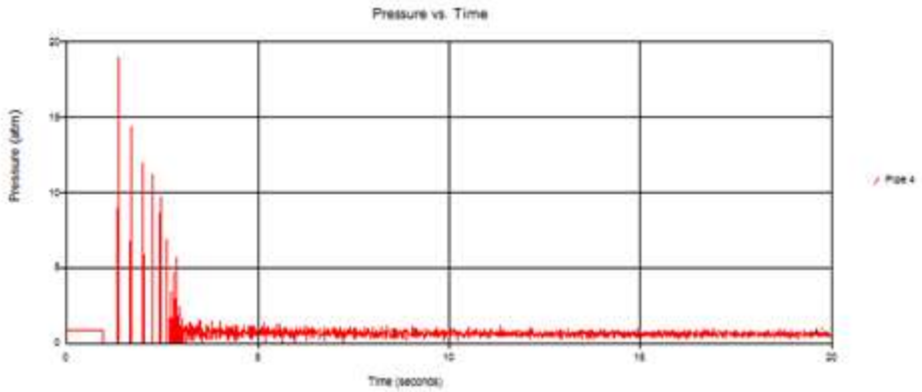

Figure 16: Computer model of the hydraulic circuit: equipped with air devices (DA): Pressure measurements at point 1 on pipe 4 in figure 7 *(at q_o = 19.21 l/s Gate Valve, Rapid closing valve time 0.095 sec.)*

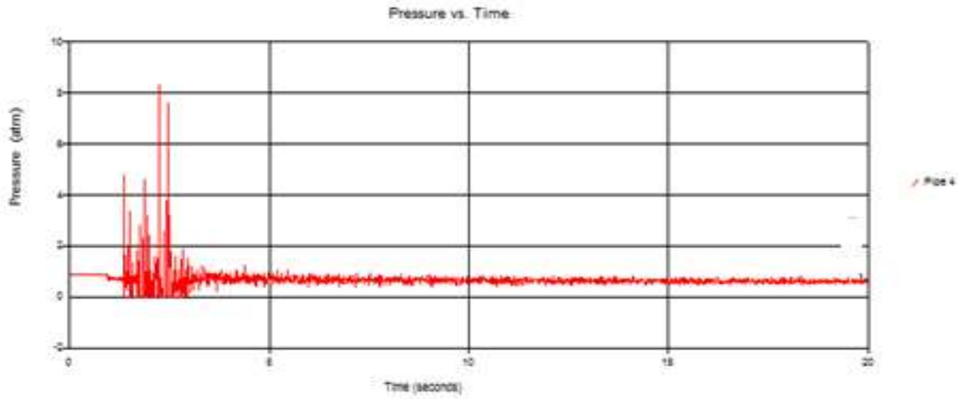

Figure 17: Computer model of the hydraulic circuit: equipped with air devices (DA): Pressure measurements at point 7 on pipe 4 in figure 6 (*at q_o = 19.21 l/s Gate Valve, Rapid closing valve time 0.095 sec.)*

3.1 Validation of Results

In attempt to give reasonable presentation of the comparison of the obtained results, the following figures are Excel graphs plotted to compare the published experimental test data against the AFT Impulse 4.0 computer models transients results generated by different rapid closure valves (Ball and Gate valves) on the pipelines system with valves closure time of 0.095 second, which is less than 0.1 second used for the original laboratory experimental test in [2]. The experimental data were extracted from figures 8 and 9 and is used to plot against the computer models results of both transients' events, induced by a rapid closure ball and gate valves.

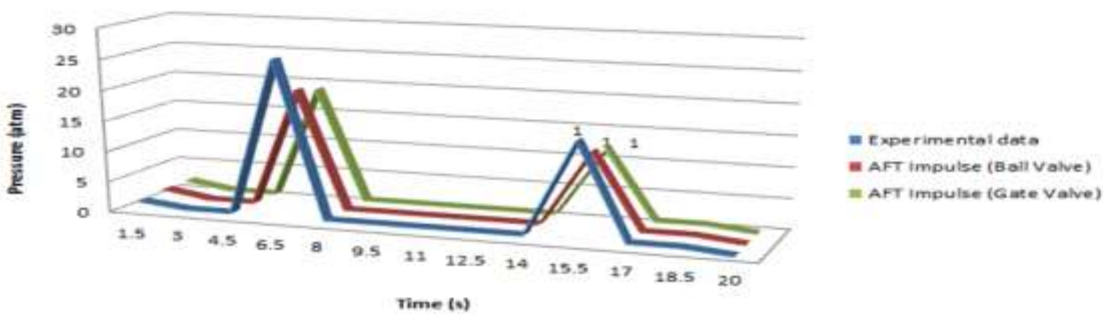

Figure 18: Pressure versus Time – Comparison of experimental test result against AFT Impulse Computer model with no protection device at point 1 on pipe 4 *(q_o = 19.21l/s, valve closing time 0.095 sec.)*

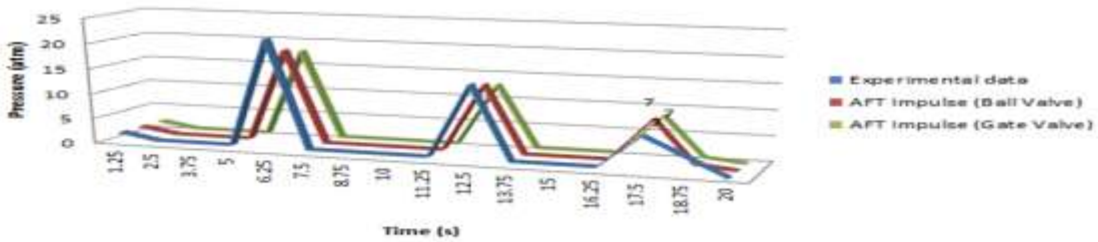

Figure 19: Pressure versus Time – Comparison of experimental test result against AFT Impulse Computer model results with any protection device at point 7 on pipe 4 *(q_o = 19.21l/s, valve closing time 0.095 sec.)*

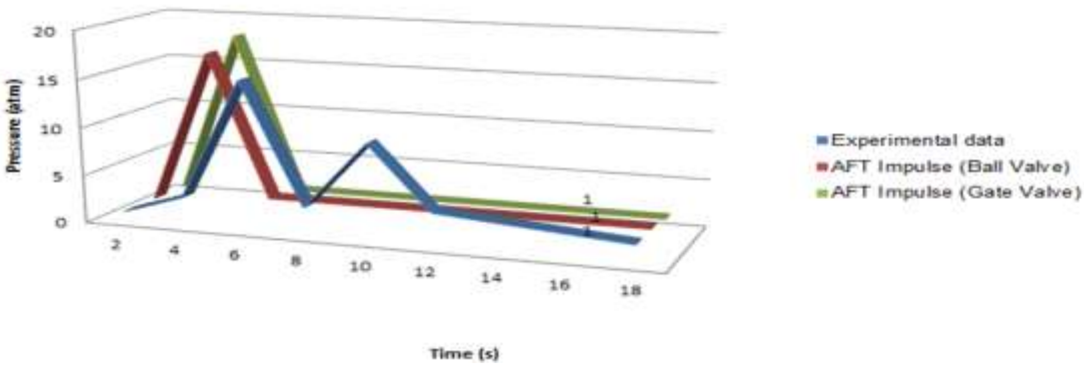

Figure 20: Pressure versus Time – Comparison of experimental test result against AFT Impulse Computer model with protection device at point 1 on pipe 4 *(q_o = 19.21l/s, valve closing time 0.095 sec.)*

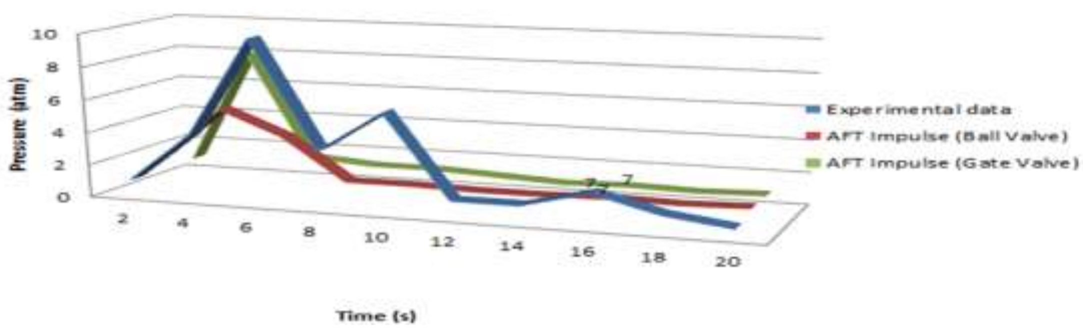

Figure: 21. Pressure versus Time – Comparison of experimental test result against AFT Impulse Computer model with protection device at point 7 on pipe 4 *(q_o = 19.21l/s, valve closing time 0.095 sec.)*

IV. DISCUSSION

The AFT Impulse computer model version without any protection and with protection device at point 1 and 7 on the 4 graphs in session 3.0 has been validated with a reasonable degree of success by comparison with data extracted from the Hydraulic Engineering Research Institute (Institute Cerceteri Hidrotehnice – I.C.H.) published experimental transient's tests data. Figures 18, 19, 20 and 21 above respectively showed the corresponding pressure profiles, wave's propagation along the pipe section 4 and pressure control with respect to time during the transient's flows regime of 0.095 seconds. It is observed that reasonable agreement is shown between the simulated computer base models version results and that of the published laboratory experimental test data. Although small inconsistency in pressure profiles with respect to time on the computer model results was detected.

It was deduced that the negligible discrepancies of pressure profiles and time observed during the transient's flows regime of the computer models could be as a result of the following reasons: The lack of sufficient details of the complex transient laboratory experimental system set-up and values that were assumed for the computer models version simulations. For example, the pipe frictions (internal surface roughness of the pipe wall), the wave propagation speed, the type of rapid closing valve used, temperature of the water, and the pump capacity, the density of the laboratory water used among others. Therefore,the AFT Impulse 4.0 for transients modelling software performance can be objectively acceptable compared with the published experimental results particularly in transients flow pipe system design and control cases, for example, as seen in the validated results of the computer models and the good response time of the computer model.In addition, the AFT Impulse could be used for prediction of hydraulic transients or water-hammer flows in pipes system with reasonable confident on the outcomes of the simulated system. Hence, the AFT Impulse 4.0 would be regarded as a useful engineering fluid flow design tool for design and analysis of transients flow in pipes system by judging on the outcomes of this study of validation method.

V. CONCLUSIONS

In conclusion, the validation of CFD software simulated results against of published experimental data have been predicted using two transient flows scenarios and two type of rapid closure valves for two pipeline models, one without any protection device and the other with protection device. Thus, for the purpose to validate the credibility of commercial CFD codes. The following conclusion can be drawn as follows:

- This validation method has further established that CFD simulation is cost-effective and significant time reduction for testing and evaluating of fluid flow predictions at the design stage for pipe network systems.
- The CFD software validation method has demonstrated ingenuity in proffering solutions for the credibility check of commercial CFD software of fluid flow in pipe systems. Therefore, software validation against experimental results such as laboratory studies or published experimental data. This has presented a practical and acceptable method that may be employed in estimating the reliability and credibility of any commercial CFD applications.
- Currently, computer modelling of transient flow in pipes system has become the major method of performing water-hammer analysis in pipe network system because of the capabilities of CFD software to solve unlimited boundaries conditions and creating numerous sections in the pipeline as well as complex pipe geometry. Hence, this practical approach of CFD software validation method can serve as a realistic sampling procedure in a population of commercial CFD software and cost effective method for utilisation of fluid flows design and analysis in pipes systems.

REFERENCES

[1]. Mohitpour, M.; Golshan, H. and Murray, A., Pipeline Design and Construction: A Practical Approach,ASME Press (American Society of Mechanical Engineers) 3rd Edition, 2007

[2]. Popescu, M.; Arsenie D. and Vlase P., Applied hydraulic transients for hydropower plant and pumping stations, A.A. BALKEMA PUBLISHERS Lisse/Abingdon/Exton (PA) Tokyo, 2003.

[3]. Boye, E.T., CFD Simulation: Design and modelling of transient flow of crude oil pipe system, M.S.c. thesis, Department of Automotive and Mechanical Engineering, Birmingham City University, United Kingdom, 2012.

[4]. Applied Flow Technology AFT, AFT Impulse 4.0 Waterhammer user guide,2955 Professional Pl, Colorado Springs, CO 80904, United States, 2012.

[5]. Micheal, B. A., An Introduction to the Method of Characteristics, New York, American Elsevier 1966.

[6]. Aljanabi, S.H., Numerical modelling of transient flow in long oil and pipeline system, Eng &Tech. Journal, Vol 28, No.16, 2010.

[7]. De Henau, V.; Raithby, G.D., A transient two-fluid model for the simulation of slug flow in pipelines--I. Theory International Journal of Multiphase Flow, Volume 21, Number 3, pp. 335-349(15), 1995.

[8]. Denton S.G., CFD simulation of highly transient flows, Ph.D. thesis, Department of Chemical Engineering, University College London, United Kingdom, 2009.

Acknowledgement

The authors acknowledge the immense help received from the scholars whose articles are cited and included in references of this manuscript. The authors are also grateful to authors/editors/publishers of all those articles, journals and books from where the research article has been studied and discussed. Special thanks go to the provider of the CFD software used (AFT Impulse 4.0) in the research work (Brian MC Manus - PISCES Engineering services U.K.).

Appendix A

Experimental set up of Hydraulic Engineering Research Institute (Institute cerceteri Hidrotehnice – I.C.H.)

(A) The water tank or reservoir: is metallic cylinder with height $H = 7.75$ m and diameter $D = 1$ m. in its upper part it has an overflow with a measuring weir and two effluent duct of 0.2 m diameter. The water supply of the tank is ensured from the water supply system of the hydraulic laboratory, where the installation is placed through a 0.2 m diameter duct fitted with valve. The hydraulic scheme II it serves as an outlet basin figure 2.

(B) The pressure pipe: made of steel, is 161 m long and 0.125m in diameter. To measure the pressure, eight section equally spaced along the pipe ($\Delta x = 20$ m) were provide, where sleeves are new fitted to allow pressure transducers to be connected. The measuring point are numbered from 1 to 8 figure (24 and 26).

(C) *The pump: of the centrifugal type, has rated power N = 13 Kn, rated speed n = 1500 rpm and steady characteristics H_o 20 m AND Q_o = 50 l/s. On the discharge pipe, a valve of 0.1 m diameter is fitted to allow the control of the pump's discharge by a non-return valve of 0.1 m diameter.*

(D) *The end basin: with a triangular measuring weir, allows measuring the steady flow when working in hydraulic scheme I.*

(E) *The closing valve (0.125 m in diameter), allows sudden closure of the conduit, with closing times t_i less than 0.1 sec.*

(F) *The non-return valve: (0.1 m in diameter) allows the isolation of the circuit pipe-outlet basin when the pump is stopped.*

(G) *The outlet pipe: (0.2 m in diameter), ensures the discharge of water overflow flow the reservior in both hydraulic schemes.*

(H) *The surge tank: H = 10.5 m in height and D = 0.2 m in diameter, can be connected to the installation in both scheme.*

(I) *The air chamber: with a volume of 140dm^3, can ensure a controlled air cushion of volume 10 -70dm^3. It can be fitted to the installation in both schemes.*

(J) *The protection device against water-hammer: the flowing device were tested.*

- *Air-devices (DA) 0.15m diameter.*
- *Air inflow/outflow device (DAD), 0.1 m diameter.*
- *Overpressure valve (SSP), 0.008 m diameter.*

(K) *The control valve: (0.1 m diameter) which allow control of the steady flow rate for the hydraulic scheme II.*

(L) *The control valve: (0.2 m diameter), which allow adjustment of the inflow to the constant-level reservoir for hydralic scheme I.*

(M) *The feed pipe: (0.2 m diameter), to the constant-level reservoir when hydraulic scheme I is operating (it works in an open circuit).*

(N) *The suction basin of the pump: to which the pipe coming from the overflow of the weir is connected. Thus, hydraulic scheme II operates in a closed circuit.*

(O) *The section for connecting to the installation in both operational variants: the experimental devices of protection including surge tank, air chamber, device against water-hammer.*

(P) *The air compressor. Which allow the necessary volume for the air cushion at a steady flow.*