COMPUTATIONALLY EFFICIENT TWO-REGIME FLOW ORIFICE MODEL FOR REAL-TIME SIMULATION

Rafael Åman, Heikki Handroos and Tero Eskola

Lappeenranta University of Technology,
Institute of Mechatronics and Virtual Engineering,
P.O. Box 20, FIN-53581 Lappeenranta, Finland

rafael.aman@lut.fi (Rafael Åman)

Abstract

Mainly in fluid power system simulation the orifice flow is clearly in turbulent area. Only when a valve is closed or an actuator driven against end stopper the flow becomes laminar as pressure drop over an orifice is approaching zero. So, in terms of accuracy, the description of laminar flow is hardly necessary. Unfortunately, when purely turbulent description of the orifice is used, numerical problems occur when pressure drop becomes close to zero. They do because the first derivative of flow with respect of pressure drop approaches infinity when pressure drop approaches zero. Also the second derivative is discontinuous. This causes numerical noise and also infinite small integration step when variable step integrator is used. In this study a numerically efficient model for the orifice flow is proposed by using a cubic spline function for describing the flow in the laminar and transition areas. Parameters for the cubic spline are selected such that its first derivative is equal to first derivative of pure turbulent orifice flow model in the boundary condition. The superiority of this model comes from the fact that no geometrical data is needed in calculation of flow from the pressure drop. In real-time simulation of fluid power circuits there exists a trade-off between accuracy and calculation speed. This investigation is made for the two-regime flow orifice model. The effect of selection of transition pressure drop and integration time step on the accuracy and speed of solution is investigated.

Keywords: Real-time, simulation, two-regime, orifice model.

Presenting Author's biography

Rafael Åman. M.Sc student in Lappeenranta University of Technology. Research assistant in Institute of Mechatronics and Virtual Engineering since 2006. Simulation of mechatronics and motion base control as research interests.

Nomenclature

A_o = geometrical orifice area
B_e = effective bulk modulus
C_d = discharge coefficient of the orifice
d_h = hydraulic diameter of the orifice
d = diameter of the orifice
K = variable depending on the type of orifice
p_0 = initial pressure
p_1 = pressure before the orifice
p_2 = pressure after the orifice
Q = volume flow
δ = empirical coefficient depending on the orifice geometry
Re = Reynolds number
Re_{cr} = critical Reynolds number
Re_{tr} = transition Reynolds number
T = integration time step
t = time
V = volume
w = flow rate
Δp = pressure drop
Δp_0 = transition pressure drop
Δp_{0_theor} = theoretical transition pressure drop
$\dot{p}$ = first derivative of pressure
ρ = density of hydraulic fluid
ν = kinematic viscosity of hydraulic fluid
√ = square root

1. Introduction

In majority of cases in fluid power system simulation the orifice flow is clearly in the turbulent area. Only when a valve is closed or an actuator driven against end stopper the flow becomes laminar as pressure drop over an orifice is approaching zero. Usually, this situation happens only in a relatively short time period. So, in terms of accuracy the description of laminar flow is hardly necessary. Unfortunately, when using pure turbulent description of the orifice numerical problems occur when pressure drop becomes close to zero. They do because the first derivative of flow with respect to pressure drop approaches infinity when pressure drop approaches zero. Also the second derivative is discontinuous. This causes numerical noise and also infinite small integration step when variable step integrator is used.

Various orifice models describing both laminar and turbulent orifice flows are proposed [1,2,3]. The discharge coefficient is taken as a function of Reynolds number in [2,3]. Merritt uses a linear function describing the laminar area [3] and Wu empirical functions that give smooth transition of the discharge coefficient value between laminar and turbulent area [2]. In Merritt's approximation the first derivative of the function is discontinuous. Wu's and Ellman's models may provide some numerical efficiency but their major drawback is that they require geometrical parameter information which is not wanted when using semi-empirical approach.

The semi-empirical modeling method was developed early 1990's by Handroos and Vilenius [4,5]. It has lately been applied in many fluid power circuit simulation programs. The superiority of the method is that the components like pressure, direction and flow control valves have not to be dismantled to identify the parameter values of their models. Instead of this, measured characteristic curves mostly provided in manufacturers catalogues can be used. A drawback in using this method has been the lack of semi-empirical orifice model describing the laminar and turbulent flow area.

The present paper proposes a numerically efficient semi-empirical orifice flow model based on cubic spline approximation of the orifice flow in laminar and transition area (area between clearly laminar and turbulent flow) and thus solves the problem discussed above. Ellman proposes similar transition description, but by using geometric orifice dimensions [1]. The pressure drop boundaries of the model are calculated using critical Reynolds number and the turbulent area value of discharge coefficient that are typical for the orifice type to be modeled.

2. Theory

2.1 Conventional models for orifice

Fig.1 shows a sharp-edged orifice. The dependency of flow on the pressure drop can be approximated by the well-known equation

$$Q = C_d A \sqrt{\frac{2(p_1 - p_2)}{\rho}} \quad (1)$$

, in which C_d is the discharge coefficient and A the geometrical orifice area ($A=A_0$ in Fig.1). In this approximation C_d is approximated to be equal to the contraction coefficient, C_c that describes the ratio between smallest area of jet in *vena contracta* point and the geometrical area of the orifice.

To use Eq.(1) in describing flow in both directions an absolute value of pressure drop and a sign function from the pressure drop must be used as follows [1]

$$Q = C_d A \sqrt{\frac{2|\, p_1 - p_2 \,|}{\rho}}\, sign(p_1 - p_2) \quad (2)$$

In the models proposed by Merritt and Wu et al. the discharge coefficient is taken as a function of Reynolds number or its square root [2,3]. Reynolds number for a circular sharp-edged orifice can be described as

$$\mathrm{Re} = \frac{wd_h}{\nu} \tag{3}$$

, where ν is the kinematic viscosity and d_h the hydraulic diameter [3]. In circular orifices d_h=d. By combining Eq.(1 and 3) and

$$w = \frac{Q}{A} \tag{4}$$

, we obtain [1]

$$\mathrm{Re} = \frac{Qd}{A\nu} = \frac{C_d d}{\nu}\sqrt{\frac{2(p_1 - p_2)}{\rho}} \tag{5}$$

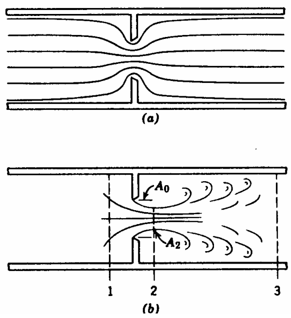

Fig. 1 Sharp-edged orifice [3]

The measured dependency of C_d on square root of R_e is shown in Fig.2. It can be concluded from the result that in the turbulent area C_d is constant and its value is close to 0.6.

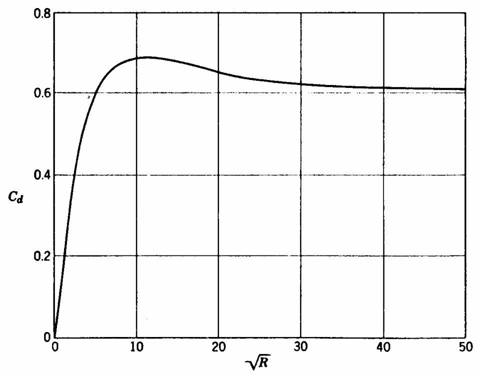

Fig. 2 Measured dependency of C_d on $\sqrt{R_e}$ [3]

Merritt proposed a simple linear description for C_d in laminar area

$$C_d = \delta\sqrt{\mathrm{Re}} \tag{6}$$

, where δ is an empirical coefficient depending on the orifice geometry [3]. Now a model may be built by using this expression when $\sqrt{\mathrm{Re}}$ is smaller than the transition value $\sqrt{\mathrm{Re}_{tr}}$. Ellman used cubic spline function for describing the flow as a function of pressure drop [1]. Wu et al. used a smooth empirical function giving a good fit with the data in Fig. 2 [2].

Drawback in Wu's expression is that iteration is required to calculate the volume flow, Q in each integration step. To solve this problem Wu et al. proposed a pre-calculated look-up table [2]. These two latter models may provide a reasonable numerical effectiveness because the first derivative of orifice model becomes finite and the second derivative becomes continuous.

2.2 Semi-empirical orifice flow model

It is shown in [4] and [5] that by assuming the density as a constant the volume flow in the fully turbulent area can be described in the semi-empirical form as follows

$$Q = K\sqrt{p_1 - p_2} \tag{7}$$

, where

$$K = C_d A\sqrt{\frac{2}{\rho}} \tag{8}$$

, where term K can be constant or variable depending on the type of orifice [4]. Its value or values can easily be determined from measured characteristic curves [4,5].

The first derivative of flow, Eq.(7), with respect to pressure drop (p_1-p_2) is

$$\frac{dQ}{d(p_1 - p_2)} = \frac{K}{2\sqrt{p_1 - p_2}} \tag{9}$$

, and the second derivative

$$\frac{d^2Q}{d^2(p_1 - p_2)} = -\frac{K}{4}(p_1 - p_2)^{-3/2} \tag{10}$$

By assuming similar behavior of the flow in positive and negative directions Eq. (7, 9 and 10) become

$$Q = K\sqrt{|\ p_1 - p_2\ |}\,sign(p_1 - p_2) \tag{11}$$

$$\frac{dQ}{d(p_1 - p_2)} = \frac{K}{2\sqrt{|\ p_1 - p_2\ |}}sign(p_1 - p_2) \tag{12}$$

$$\frac{d^2Q}{d^2(p_1 - p_2)} = -\frac{K}{4}|p_1 - p_2|^{-3/2}sign(p_1 - p_2) \tag{13}$$

Fig.(3, 4 and 5) show results calculated by Eq.(11,12 and 13) in small pressure drops with constant value of K (K=1.0*10^{-7} $m^3/s\sqrt{Pa}$).

Indeed it can be concluded from the Fig. 4 that the first derivative of Q, Eq.(12), is approaching infinity at zero pressure drop. Also the second derivative of Q, Eq.(13), is discontinuous, as shown in Fig.5. This makes purely turbulent orifice model numerically very inefficient.

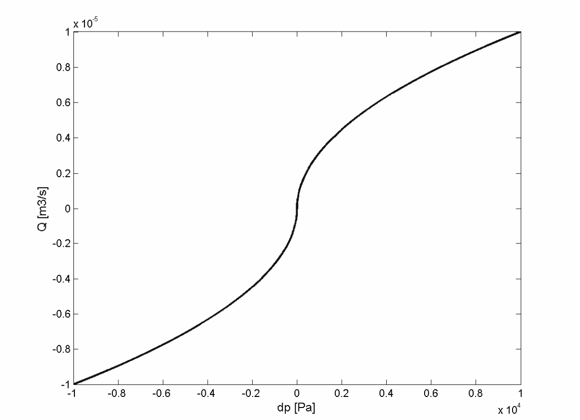

Fig.3 Q(Δp) calculated by turbulent orifice model

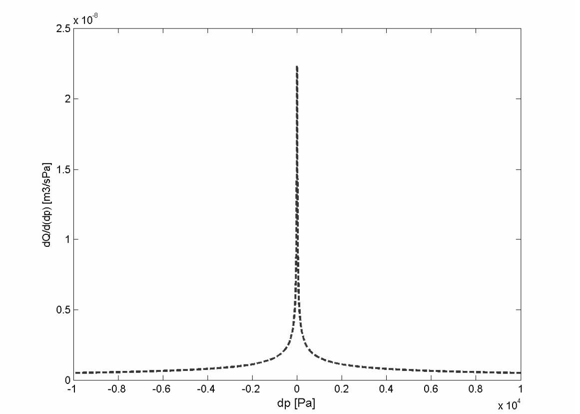

Fig. 4 dQ/d(Δp) calculated by turbulent orifice model

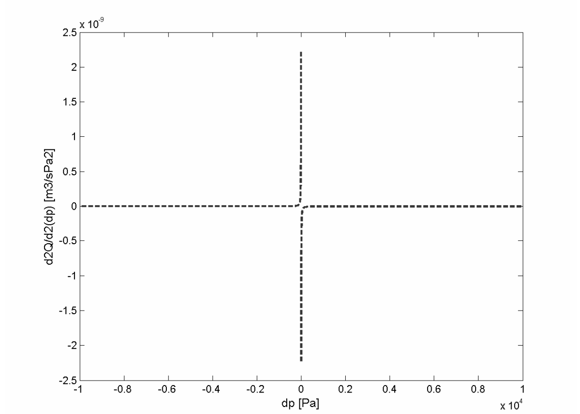

Fig. 5 $d^2Q/d^2(\Delta p)$ calculated by turbulent orifice model

2.3 Two-regime flow orifice model

In the following a numerically efficient model for the orifice flow is proposed by using a cubic spline function for describing the flow in the laminar and transition areas. Parameters for the cubic spline are selected such that its first derivative is equal to first derivative of pure turbulent orifice model, Eg.(12), in the boundary conditions. These boundary conditions are described by $\pm\Delta p_0$. They are calculated by using current value for the variable K, critical Reynolds number Re_{cr} and value of discharge coefficient in turbulent area $C_{d\infty}$ to obtain physically justified values. It must be noted that the superiority of this model comes from the fact that geometrical data is not needed in calculation of flow from the pressure drop.

Let us approximate the laminar and transition areas of the flow by the following cubic spline function

$$Q = a_0 + a_1\Delta p + a_2\Delta p^2 sign(\Delta p) + a_3\Delta p^3 \tag{14}$$

, where $a_0 \ldots a_3$ are constants.

The first derivative of Q, Eq.(14) with respect to Δp is

$$\frac{dQ}{d\Delta p} = a_1 + 2a_2\Delta p + 3a_3\Delta p^2 sign(\Delta p) \tag{15}$$

Now by using boundary conditions $\pm\Delta p_0$ and assuming same value of Q in Eq.(11 and 14) and derivative of Q in Eq.(12 and 15) the constants $a_0 \ldots a_3$ can be solved from the following equilibrium

$$\begin{aligned}
&a_0 + a_1\Delta p_0 + a_2\Delta p_0^{\,2} + a_3\Delta p_0^{\,3} = K\sqrt{\Delta p_0} \\
&a_1 + 2a_2\Delta p_0 + 3a_3\Delta p_0^{\,2} = \frac{K}{2\sqrt{\Delta p_0}} \\
&a_0 + a_1\Delta p_0 - a_2(-\Delta p_0)^2 + a_3\Delta p_0^{\,3} = -K\sqrt{|-\Delta p_0|} \\
&a_1 + 2a_2\Delta p_0 - 3a_3(-\Delta p_0)^2 = -\frac{K}{2\sqrt{|-\Delta p_0|}}
\end{aligned} \tag{16}$$

Eq.(16) can be expressed in matrix form as

$$CA = D \tag{17}$$

, where

$$A = \begin{bmatrix} a_0 & a_1 & a_2 & a_3 \end{bmatrix}^T$$

$$C = \begin{bmatrix} 1 & \Delta p_0 & \Delta p_0^{\,2} & \Delta p_0^{\,3} \\ 0 & 1 & 2\Delta p_0 & 3\Delta p_0^{\,2} \\ 1 & \Delta p_0 & -\Delta p_0^{\,2} & \Delta p_0^{\,3} \\ 0 & 1 & 2\Delta p_0 & -3\Delta p_0^{\,2} \end{bmatrix}$$

$$D = \begin{bmatrix} K\sqrt{\Delta p_0} & \frac{K}{2\sqrt{\Delta p_0}} & -K\sqrt{|\Delta p_0|} & -\frac{K}{2\sqrt{|\Delta p_0|}} \end{bmatrix}^T$$

By using matrix algebra the parameter vector A can be solved as

$$A = C^{-1}D \tag{18}$$

In practice, because the matrix solution is time-consuming, analytical equations for a0…a4 that can be obtained from Eq.(18) should be used in the final orifice model.

Now, by combining the laminar, transition and turbulent regions the final semi-empirical orifice model can be expressed as follows

$$Q = a_0 + a_1\Delta p + a_2\Delta p^2 sign(\Delta p) + a_3\Delta p^3$$
$$\text{, when } |\Delta p| < |\Delta p_0| \tag{19}$$
$$Q = K\sqrt{|\ p_1 - p_2\ |}sign(p_1 - p_2)$$
$$\text{, when } |\Delta p| \geq |\Delta p_0|$$

Now the only task that is left is to find physically adequate values for $\pm\Delta p_0$. This can be solved from analytical formula of Reynolds number if it is possible to use variables K and Δp in the formula instead of Q, w and d. This problem is solved in the following:

Analytical formula for Reynolds number for orifice type shown in Fig.1 is [1]

$$\mathrm{Re} = \frac{wd}{\nu} = \frac{Qd}{A\nu} \tag{20}$$

By substituting Eq.(1) for Q in Eq.(20) we get

$$\mathrm{Re} = \frac{C_d d\sqrt{\frac{2\Delta p}{\rho}}}{\nu} \tag{21}$$

By solving Eq.(21) for Δp we obtain

$$\Delta p = \frac{\rho \mathrm{Re}^2 \nu^2}{2C_d^2 d^2} \tag{22}$$

By using Eq.(8) and relation $A=\pi d^2/4$, term d can be written as a function of K as follows

$$d = \sqrt{\frac{4K}{C_d \pi\sqrt{\frac{2}{\rho}}}} \tag{23}$$

The substitution of Eq.(23) into Eq.(22) yields

$$\Delta p = \frac{\rho\ \mathrm{Re}^2 \nu^2 \pi\sqrt{\frac{2}{\rho}}}{8C_d\ \ K} = \frac{\mathrm{Re}^2 \nu^2 \pi\sqrt{\rho}}{5.657\ C_d K} \tag{24}$$

Now by using Eq.(24) the boundary conditions for the orifice model Eq.(19) can be found for any value of K if transition Reynolds number Re_{tr} and the turbulent region value of C_d ($C_d=C_{d\infty}$) are known. By using these values the boundary conditions $\pm\Delta p_0$ can be calculated as follows [1]

$$\Delta p_0 = \frac{\mathrm{Re}_{tr}^2 \nu^2 \pi\sqrt{\rho}}{5.657 C_{d\infty}\ K} \tag{25}$$

$$-\Delta p_0 = -\frac{\mathrm{Re}_{tr}^2 \nu^2 \pi\sqrt{\rho}}{5.657 C_{d\infty}\ K} \tag{26}$$

3. Example

In this study Matlab software with its extension Simulink is used to model a simple fluid circuit. Circuit studied is represented in Fig. 6.

Studying of the orifice was started by forming the Simulink model based on the differential equation of pressure drop, Eq. (27) and alternative equations of the volume flow, Eq.(19). Usage of these alternative equations depends on current pressure and the defined boundary condition. Boundary condition separates the turbulent area from laminar and transition areas.

3.1 Fluid circuit modeled

In Fig. 6 there is a fluid source with initial parameters, a sharp-edged orifice as shown in Fig.1 and a separate container for fluid recovery. These components are connected with pipes to form a compact fluid circuit.

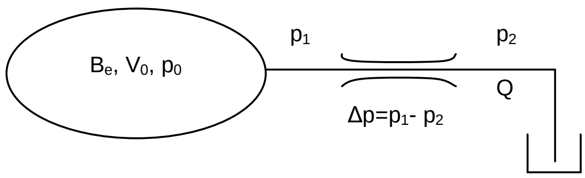

Fig. 6 A simple fluid circuit that is difficult to solve numerically near Δp = 0

In the initial state of the fluid circuit represented in Fig.6 in the fluid source there reigns defined volume, V, and initial pressure, p_0. For the differential equation of pressure drop, Eq.(27) the effective bulk modulus, B_e, is defined.

Value for the pressure p_1 is solved by integrating the differential equation of pressure after fluid begins to flow out of the container. Value for the pressure p_2 can be observed as zero because fluid flows freely to the recovery container. By this assumption the pressure drop over the orifice, Δp, is directly the value of the pressure p_1.

Volume flow through the orifice, Q, is solved from the pressure drop by Eq.(19) either in turbulent or in laminar area depending on the current pressure and the defined boundary condition.

3.2 Differential equilibrium of the pressure

$$\dot{p}(t) = -\frac{B_e}{V} Q(\Delta p) \qquad p(0) = p_0 \qquad (27)$$

, where Q is the volume flow out of orifice [1]

By integration of Eq.(27) value for the pressure p can be solved as noticed in chapter 3.1. In this case $p=p_1$. In initial state the volume flow, Q, is calculated by Eq.(19).

Due to assumption made in chapter 3.1 when $p_1(0) = p_0$ also pressure drop $\Delta p(0) = p_0$ in initial state. Hence the volume flow, Q, solved from the square root of pressure drop gets non-zero value.

In the first time step Q in Eq.(27) is substituted by its initial value mentioned above. Hence Eq.(27) gets value of non-zero which enables the integration and simulation to continue and correspond to real-life system.

3.3 Matlab/Simulink model created

In Fig.7 is shown the principle of working with the fluid power circuit in Matlab/Simulink. Initial values and all supportive calculation formulas are formed in M-File editor. This script controls the simulation run. During the simulation run all values are updated to Matlab Workspace where they are available for the Simulink model. Simulink model uses these initial values from Workspace and returns user-defined results to the Workspace. Results are also returned to the script which post-processes i.e. makes the comparison between different pressure vectors and plots graphs to illustrate the results.

The system in Fig. 7 prepares the way to test values of transition pressure and integration time step with minimum efforts. After vectors of range are defined the simulation can be started by the script which also controls all operations needed in fluid power simulation. User sees these operations only as one.

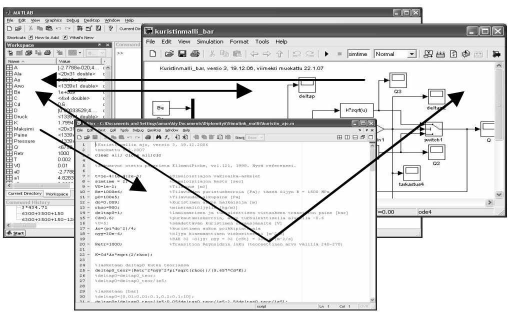

Fig. 7 Illustration of used tools and their interoperability

3.4 Influence of the cubic spline function

The cubic spline function produces error to the volume flow in comparison with conventional orifice flow model. This issue can not be avoided because the conventional model has the characteristics that are mentioned above. And on the other hand it is intended to get those characteristics out of its system. Hence the influence of cubic spline function is studied.

In Fig. 8 cubic spline and conventional formula separated by transition pressure is represented. Volume is calculated by conventional formula until current pressure meets the transition pressure threshold (spot in Fig. 8). When current pressure is lower than transition pressure the laminar and transition areas are calculated by cubic spline function.

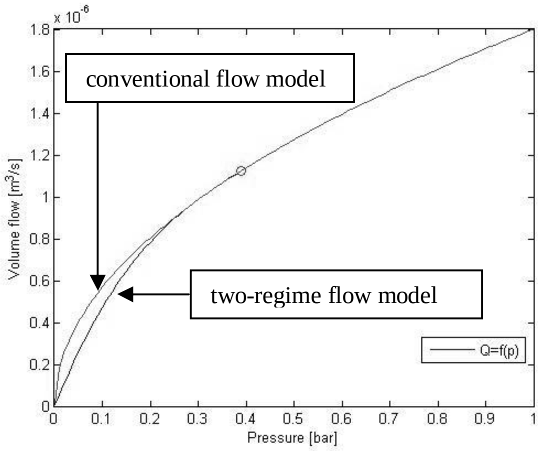

Fig. 8 Behavior of the two-regime orifice flow model compared to the conventional orifice flow model

Fig. 8 also illustrates the behavior of the two-regime orifice flow model compared to the conventional orifice flow model. The difference between models comes fully from the behavior in laminar and transition areas. Two-regime model differs advisedly from conventional in order to get volume flow continuous near zero pressures.

4. Post-processing

The post-processing perceives the management of overtaken results. Separate simulation runs are driven with various time step lengths and transition pressure threshold values.

Length of time step varies from 0.1ms up to 2ms at intervals of 0.1ms. Transition pressure threshold value, Δp_0, varies from theoretical transition pressure Δp_{0_theor}, see Eq.(25), up to 2.5 times Δp_{0_theor} at intervals of 0.1 times Δp_{0_theor}.

Example includes 20 different time steps and 31 different transition pressure threshold values. Together these forms 20 times 31 different pressure vectors. In terms of accuracy the theoretical transition pressure is taken as reference point because it is physically justified. As reference point the shortest (0.1ms) time step is selected. It can be assumed to be most accurate of varied time step because integrator naturally makes the most integration steps.

The reference pressure vector contains data overtaken by these reference point values and all following pressure vectors are compared to it. Because of dissimilar lengths of compared vectors the interpolation is made to get vectors comparable.

The difference between compared discrete pairs is integrated numerically by using the midpoint rule [6]. By means of this integration the area that describes error in comparison to reference graph is attained. After every discrete pair is compared and integrated the sum of attained values is stored into the separate array. Absolute value of difference is taken to avoid the possible negative areas from deducting the total area.

The array mentioned above is filled by results of different pairs corresponding varied time step lengths and transition pressure thresholds. Finally this array is plotted as shown in Fig. 12.

Similarly the maximum difference between compared vectors is treated. As from Fig. 9 and 10 can be seen the maximum difference between compared vectors lies in the cubic spline interval. At the end of simulation run both vectors compared aspires the zero. By this notice it is assumed that also the maximum difference tells something about the influence of tested variables. The less difference the near vector is to the reference.

The array is also formed of maximum difference values as described above. Plotted array is shown in Fig. 13.

In Fig. 9 lower curve illustrates the reference pressure graph and the upper illustrates the pressure graph reached by using the maximum time step length defined. Value for transition pressure drop has been kept constant.

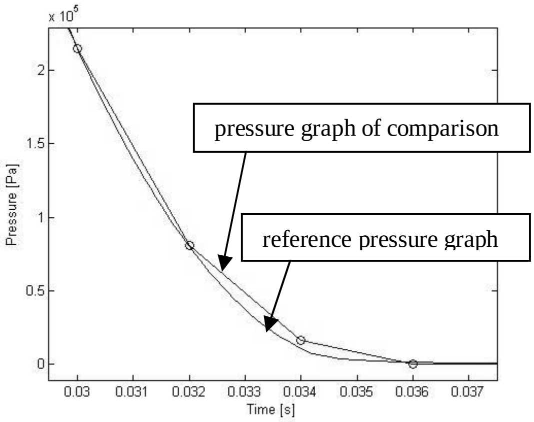

Fig. 9 Illustration of the effect of elongation of time step length (orifice diameter 9mm)

In Fig. 10 reference pressure graph illustrates the reference pressure graph and the pressure graph of comparison illustrates the pressure graph reached by using the maximum value for transition pressure defined. Time step length has been kept constant.

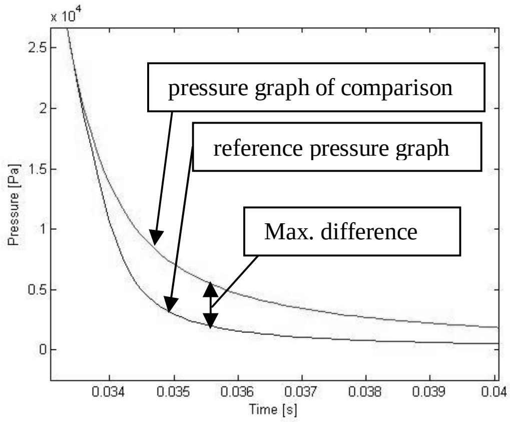

Fig. 10 Illustration of the effect of enlarging the value of transition pressure threshold (orifice diameter 9mm)

5. Results

In this study 10 different orifice diameters from 1 to 10 mm are studied. Each of them is treated as described in chapter 4.

First reference pressure vector is formed by simulation run whose simulation time is based on desired duration of laminar and transition areas i.e. time that cubic spline function is in use. Desired duration is defined by test runs whom show that end pressure settles low enough i.e. practically zero.

After duration of laminar and transition areas is defined the simulation time (run time) is transmitted to be constant in every simulation run concerning current diameter.

Tab. 1 illustrates difference of pressures at the end of simulation run between reference pressure vector and the pressure vector attained by longest time step. Value of transition pressure drop has been kept constant.

Theoretical transition pressure threshold, Δp_{0_theor} , is calculated by Eq.(25). Tab. 1 shows that end pressures are clearly under calculated threshold i.e. defined simulation time is sufficient.

The margin originates from the dissimilar amount of calculated points. The longer time step the less integration steps are made. For this reason the compared end data points may diverse.

Tab. 1 The influence of elongation of time step length

Ø	Run time	Δp_{0_theor} [Pa]	Reference end pressure [Pa]	End pressure [Pa]	Margin [Pa]
10	0.1275	11 249.7	18.0	0	-18
9	0.1338	13 888.5	30.6	0	-30.6
8	0.1426	17 577.7	55.3	55.2	-0.1
7	0.1553	22 958.6	108.2	108.3	0.1
6	0.1746	31 249.2	235.6	237.1	1.5
5	0.2062	44 998.8	592.9	594.2	1.3
4	0.2630	70 310.7	1 847.4	1 867.0	19.6
3	0.3809	124 996.8	8 103.1	8 180.9	76.8
2	0.6923	281 242.8	63 782.4	63 967.7	185.3
1	1.9921	1 124 971.1	898 283.8	898 494.8	211.0

Tab. 2 (consists of parts 2a and 2b) illustrates difference of pressures at the end of simulation run between reference pressure vector and the pressure vector attained by maximum transition pressure drop. Time step length has been kept constant.

Theoretical transition pressure drop, Δp_{0_theor} , is calculated by Eq.(25). Maximum value is selected to be 2.5 times theoretical transition pressure drop. For this reason the margin develops bigger as the orifice diameter gets smaller while also transition pressure drop gets bigger values.

The margin originates from variation of transition pressure drop. Volume flow is calculated by cubic spline function starting from different value of current pressure i.e. Fig. 10 illustrates. While simulation time (run time) is the same in both cases the varied curve does not attain the reference curve.

Tab. 2a The influence of enlarging the value of transition pressure drop

Ø [mm]	Run time	Δp_{0_theor} [Pa]	Reference end pressure
10	0.1275	11 249.7	18.0
9	0.1338	13 888.5	30.6
8	0.1426	17 577.7	55.3
7	0.1553	22 958.6	108.2
6	0.1746	31 249.2	235.6
5	0.2062	44 998.8	592.9
4	0.2630	70 310.7	1 847.4
3	0.3809	124 996.8	8 103.1
2	0.6923	281 242.8	63 782.4
1	1.9921	1 124 971.1	898 283.8

Tab. 2b The influence of enlarging the value of transition pressure drop

Ø [mm]	Max. Δp_0 [Pa]	End pressure [Pa]	Margin [Pa]
10	28 124.3	71.0	53.0
9	34 721.3	120.4	89.8
8	43 944.3	217.1	161.9
7	57 396.5	423.6	315.4
6	78 123.0	915.9	680.3
5	112 497.0	2 271.1	1 678.2
4	175 776.8	6 825.5	4 978.1
3	312 492.0	26 938.5	18 835.4
2	703 107.0	149 076.3	85 293.9
1	2 812 427.8	1 136 274.2	237 990.4

Examples shown in Fig. 11, 12 and 13 are attained by using the orifice diameter of 9 mm.

As from Fig. 11 can be seen behaviour of graphs is continuous as expected. Graphs draw on smoothly to zero without numerical noise i.e. they are not vibrating in zero pressure surrounds.

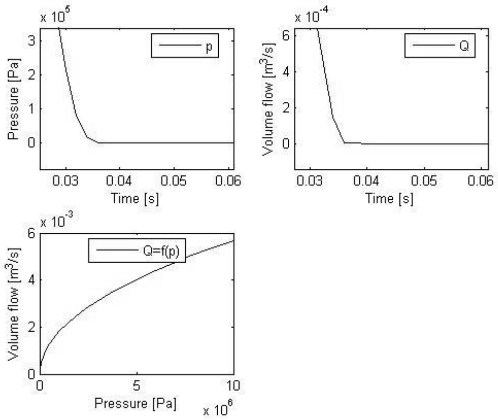

Fig. 11 Example of attained results

Fig. 12 shows integrated error between compared pairs in relation to time step length and transition pressure drop. In terms of accuracy and speed of calculation the optimum must be the best combination of time step length and transition pressure drop. In this case it seems to be located in the near of T=1 ms and Δp_0=0.2 bar.

Fig. 13 shows maximum difference of compared vectors in relation to time step length and transition pressure drop. This result supports the implication made in chapter 4.

ISBN 978-3-901608-32-2

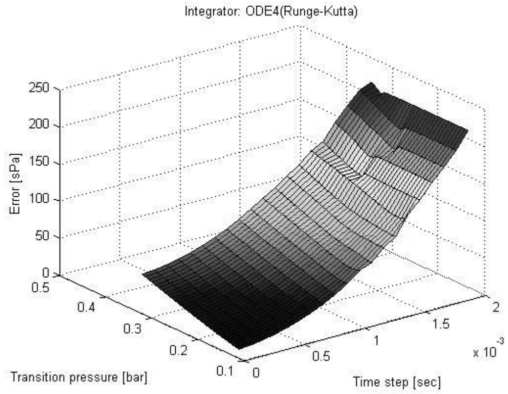

Fig. 12 Integrated error

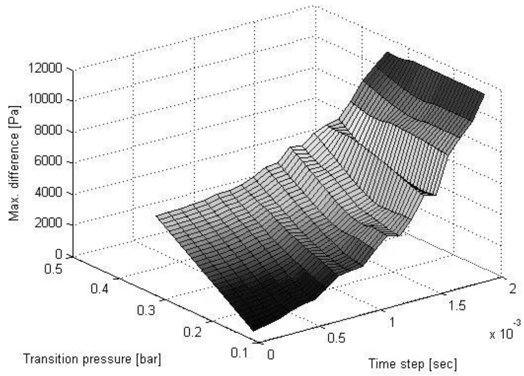

Fig. 13 Maximum difference

6. Conclusions

A numerically efficient, physically justified two-regime orifice model is proposed. The laminar and transition region in the orifice model are described by a cubic spline function that gives continuous second derivative of flow with respect to pressure drop. The traditional turbulent orifice model is used in describing the turbulent region.

The model can be used in semi-empirical modelling of fluid power components because it does not require any geometrical information of the orifice type. If it is desired to use physically adequate values for boundary pressure drops the equation for Reynolds number can be used in calculating their values. Then the transition value for Reynolds number and turbulent region value for the discharge coefficient should be given. It is shown that the model gives quite good approximation of discharge coefficient as a function of derivative of Reynolds number.

The dependency of calculation error on the integrator time step and selected boundary pressure drop is also investigated by simulating a simple hydraulic circuit.

The physically justified transition pressure is used as the reference response. Maximum errors and integrated errors are illustrated.

References

[1] A. Ellman and R. Piché. A Two Regime Orifice Flow Formula for Numerical Simulation. *ASME Journal of Dynamics Systems, Measurement, and Control*, Vol.121, pp.721-724, December 1999.

[2] D. Wu, R. Burton and G. Schoenau. An Empirical Discharge Coefficient Model for Orifice Flow. *International Journal of Fluid Power*, No.3, pp 13-18, December 2002.

[3] H. Merritt. Hydraulic Control Systems. John Wiley et Sons. 1967.

[4] H. Handroos and M. Vilenius. The Utilization of Experimental Data in Modelling Hydraulic Single Stage Pressure Control Valves. *ASME Journal of Dynamics Systems, Measurement, and Control*, Vol.112, Number 3, pp 482-488, September 1990.

[5] H. Handroos and M. Vilenius. Flexible Semi-empirical Models for Hydraulic Flow Control Valves. *ASME Journal of Mechanical Design*, Vol.113, Number 3, pp.232-238, September 1991.

[6] L. Råde and B. Westergren. Mathematics Handbook for Science and Engineering. Studentlitteratur. 2004.