ISSN 2229-5518

CFD Analysis and Evaluation of Flow Resistance Coefficient of a Quick Closing Non Return Valve(QCNRV)

Gokul Prasath B[1], Kalaiarasan R[2], Vivekanadan M[3], Joseph Manickam A[4]
PG Student, MAM College of Engineering and Technology, Trichy, Tamil Nadu, India

Abstract— Quick Closing Non-Return Valve (QCNRV) is basically a Non return valve, equipped with a power cylinder to close the valve quickly. This non return valve allows the fluid to flow in one direction only. These valves are used mainly in typical power plant application. The valve is installed in turbine extraction lines wherein the back flow of wet steam to turbine is not allowed. In case of boiler trips, steam to turbine will be disconnected, at same instant the QCNR Valve is actuated by power cylinder and it closes the flap in 0.5seconds, thus preventing the reverse flow of wet steam to turbine which is detrimental to the turbine casing and rotor. The pressure drop offered by the valve is important as the steam is taken for some other process. The pressure drop is based on the Flow resistance coefficient (ZETA) of the valve. To evaluate the value of zeta, Test Rigs are set up and tests are conducted according to International Test standards. Based on test results, Zeta is evaluated.But testing needs costly test set up, takes more time for design and erection, calibration of various instruments and needy of transportation of the valve to test center. All these makes the testing procedure a tedious job. So as an endeavor Computational Fluid Dynamics (CFD) technique is employed to evaluate zeta so that most of the practical difficulties in actual testing can be avoided. This paper also deals with study the flow pattern of the fluid inside the Valve .The control volume is modeled in Pro/E 2001 and the flow is analysed in CFX 5.5.1 software. The pressure drop across the valve, velocity data are taken from the CFX results and the flow resistance coefficient is calculated. The result is compared with the experimental result data obtained from the standard valve test rigs. Comparison with the field data showed that the CFD program (CFX5.5.1) could reproduce the real conditions quite well. The deviation from the field data was much smaller that makes the CFD as a efficient tool to simulate the real life situation.

Keywords - Quick closing Non-Return Valve, Uni direction Valve, Pressure drop, Flow resistance coefficient

INTRODUCTION

Quick closing non-return valve (QCNRV) is a non return valve, which allows the fluid to flow through in one direction only. The NRV flap opens as the fluid flow through it and closes due to its self weight when there is no flow. It is widely used in power plants on turbine extract lines. In case of any emergency turbine will be tripped off .So the fluid flow to the valve gets stopped and the fluid starts flowing reversely to the turbine due to difference in pressure .If the fluid is allowed to flow inside the turbine, it will damage the turbine blades. Hence a Non return valve is used in extraction lines. Due to prolonged usage, the flap may not close when the flow ceases. In order to ensure a positive closure the valve a power cylinder is employed externally to the disc ,arm ,spindle assembly to close the flap immediately in 0.5 seconds. In addition to this the reverse flowing dynamic pressure of the fluid also enables effective closing of the valve.The valve is designed to open upto 35 degrees. The valve internal contours are designed in such a way that for 17.5 degree opening of the flap full flow is achieved .The amount of flap opens is directly proportional to the mass flow rate. Lesser the mass flow rate lesser the flap opening. So for maximum opening of flap the pressure drop is minimum and vice versa.

FLOW RESISTANCE COEFFICIENT

The pressure drop across the valve is an important criterion for the end user. So in order to evaluate the pressure drop across the valve, the flow resistance coefficient is to be determined. The flow resistance coefficient is

ISSN 2229-5518

unique for particular size valve.Flow resistance coefficient (ζ) is a measure of the pressure loss across the valve and is calculated from the relation

$$\varsigma = \frac{\Delta P}{1/2\rho v^2}$$

Where,

Δp is the difference between upstream and downstream pressure P1– P2 in Pa

ρ is the density of fluid in kg/m3

v is the velocity of fluid at inlet in m/s

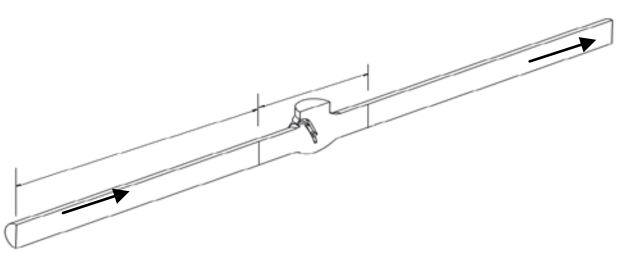
From the pressure drop, velocity and density at the specified temperature and pressure the zeta value can be found out.

OBJECTIVE

The objective of the paper is to simulate the test condition through CFD technique and to compute the flow resistance coefficient of a quick closing non-return valve using CFD technique. Hence the lengthy experimental procedure can be eliminated to large extent, also the variation of flow parameters at every point of domain can be spotted out which will be helpful in improving the design.

GEOMETRIC DETAILS

Simulating the Flow through the valve some smart assumptions are needed. But in other hand, those approximations must not affect the reality of the problem. In this project the main approximation done in the arm portion. The arm portion consists Small fillets, and hole to accommodate the stem of the disc. So in flow analysis the Arm is placed at the rear side of the disc hence does not obstruct the flow directly, also small fillets will act as sliver surfaces thus affecting the mesh qualiy.Also the nut at the rear side of the disc is removed.Also to ensure that the boundary conditions are applied at the right places where flow separation is not there and flow is streamlined, the end faces of the ... ie di-



EXPORT

The main aspect of exporting from Pro/E is ,CFX accepts geometry in surfaces form only, so the geometry is exported as surfaces in IGES format with an accuracy of 1e-4.

IMPORT

When the geometry is imported into CFX, the surface model should be made as B-Rep solid. The process of converting surface boundaries into solid involves checking of surface continuity up to an accuracy of 0.005mm .So the small sliver surfaces are removed and a new large surface was created so that the gap between surfaces are below the tolerance level. This prevents the creation of highly distorted elements during mesh generation.

DISCRETISATION

Discretisation is a process by which a closed-form mathematical expression ,such as a function or a differential or integral equation involving functions, all of which are viewed as having an infinite continuum of values throughout some domain, is approximated by analogous expressions which prescribe values at only a finite number of discrete points or volumes in the domain.The matter of grid generation is a significant consideration in CFD .The type of grid chosen for a problem has a direct effect on the solution to be obtained. CFX 5.5.1 uses unstructured grids to discretise the control volume.

UNSTRUCTURED MESHING OF CONTROL VOLUMES

Unstructured grids have the advantage of generality in that they can be made to conform to nearly any desired geometry. This generality, however, comes with a price. The grid generation process is not completely automatic and may require considerable user interaction to produce grids with acceptable degrees of local resolution while at the same time having a minimum of element distortion. Unstructured grids require more information to be stored and recovered than structured grids (e.g., the neighbor connectivity list), and changing element types and sizes can increase numerical approximation errors.

A popular type of unstructured grid consists of tetrahedral elements. These grids tend to be easier to generate than those composed of hexahedral elements, but they generally have poorer numerical accuracy. For example, it is difficult to construct approximations that maintain an accurate propagation of one-dimensional flow disturbances because tetrahedral grid elements have no parallel faces.

In summary, the best choice for a grid system is a compromise of several factors: convenience in generation, memory requirements, numerical accuracy, flexibility to conform to complex geometries and flexibility for localized regions of high or low resolution.

MESH GENERATION METHOD

The surface mesh technique used is Delaunay Surface Meshing and the technique used for volume meshing is Advancing Front And Inflation. Mesh controls such as point mesh controls are used where mesh finesse is required.In CFX the meshing is performed in two stages. Initially surface meshing is done, then it is converted into volume mesh while writing the definition file for solver.

SURFACE MESHING

Surface meshing works on Fluid Domains created in CFX-Build. The surface mesh of triangular elements (by default) is generated through the Mesh forms according to the control settings that are current for the surface mesher. The surface mesh is not saved in the database in order to reduce storage requirements. Two surface meshers, the Delaunay and the Advancing Front (AF), are available for use with the default AFI volume mesher.

Delaunay Surface Mesher

Delaunay surface meshing is characterized by its speed and its ability to mesh closed surfaces. In general it is recommended that Delaunay be used for surface some cases where surfaces are poorly, parameterised, improved mesh quality may be obtained by using the Advancing Front method.

ADVANCING FRONT (AF) SURFACE MESHER

AF surface meshing is slower than Delaunay, but for some geometries can be more robust and may produce a higher quality mesh. It is not possible to mesh closed surfaces using the AF surface mesher.

Volume Meshing

Advancing Front And Inflation (AFI)

The volume mesh of tetrahedral elements (together with prismatic and pyramidal elements if “inflation” is used) is generated when the Definition File is written.
The AFI volume mesher allows ‘Element Inflation’. This is used to ‘grow’ a series of prismatic volume elements from triangular elements at the surface. This produces a more computationally efficient mesh near the boundaries of the Domain, where velocity gradients are large normal to the surface but small parallel to it.

As the model consists of 10D length pipe before and after the valve, the pipe portion is meshed coarse and the valve portion is meshed relatively fine.
The methods available for setting mesh parameters are

1.Global Tetrahedral element mesh edge length applicable to all elements.
2.Mesh control types such as point, line, triangle, surface for localized refinement.
3.Prismatic element mesh for near wall treatment in which k-epsilon model will not be applied.

Mesh Controls

Mesh Controls are used to refine the surface and volume mesh in specific regions of the model. The mesh refining effect decays with increasing distance from the control, generating progressively coarser elements. Four types of Mesh Controls are available,

- Point
- Line
- Triangle
- Surface

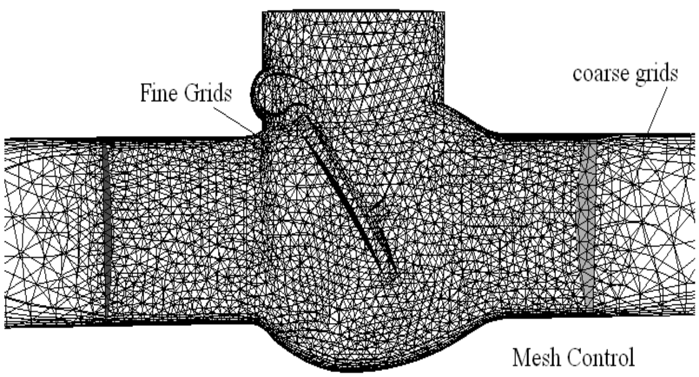
The first three are classed as volumetric controls and the last is a Surface Mesh Control.
Mesh Controls can be defined using any valid CFX-Build Point description. They can be located anywhere in the 3D space of your model: inside, outside or on the surface of the Fluid Domain.

Surface Mesh Controls

When creating a Surface Mesh Control, the Surface Mesh Spacing is controlled locally by using the **Constant**, **Relative Error** or **Angular Resolution**

ISSN 2229-5518

options. Surface mesh controls do not have a volumetric effect. An expansion factor can not be applied to affect neighbouring surfaces.



Parameters

Mesh Parameters control the background size of volume and surface
mesh elements.

Volume Mesh Spacing

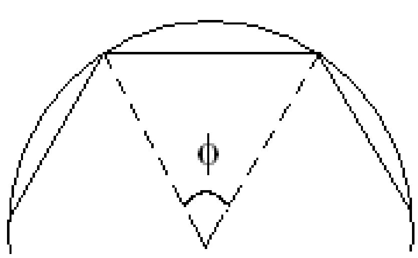
Generally, the **Maximum Edge Length** is used to provide a maximum background volume mesh scale. A default value is provided which is set to 5% of the maximum extent of the geometry.

Surface Mesh Spacing

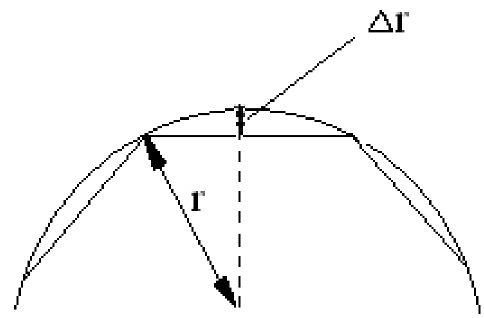
In addition to volume mesh control, you can control the size of the surface mesh length scale using a variety of techniques:

• Use volume spacing - this means that for the background length scale, the **Maximum Edge Length** is used.

• Angular resolution - this selection turns edge and surface curvature sensitivity on, where the sensitivity is specified using an angular parameter.



• relative error - this selection also turns edge and surface curvature sensitivity on, but the sensitivity is specified using a fractional parameter.



The fractional error in discretisation is the maximum deviation of the resulting mesh away from the geometry surface expressed as, $error = \frac{\Delta r}{r}$ where r is the local radius of curvature. The value entered should be in the range of approximately 0.001 to 0.13, which corresponds to 72 edges and 6 edges per circumference respectively.

• Constant - this selection allows you to set a fixed length scale for all surfaces. This setting overrides the volume length scale on all surfaces. The default value is 10% of the background volume mesh length scale.

Minimum and maximum surface length scales can also be set when using curvature sensitivity to prevent over-refinement in regions of high curvature, and over-coarsening on flat surfaces. By default, the minimum edge length is set to 1% of the background volume length scale, and the maximum is set to the same value as the background volume length scale.

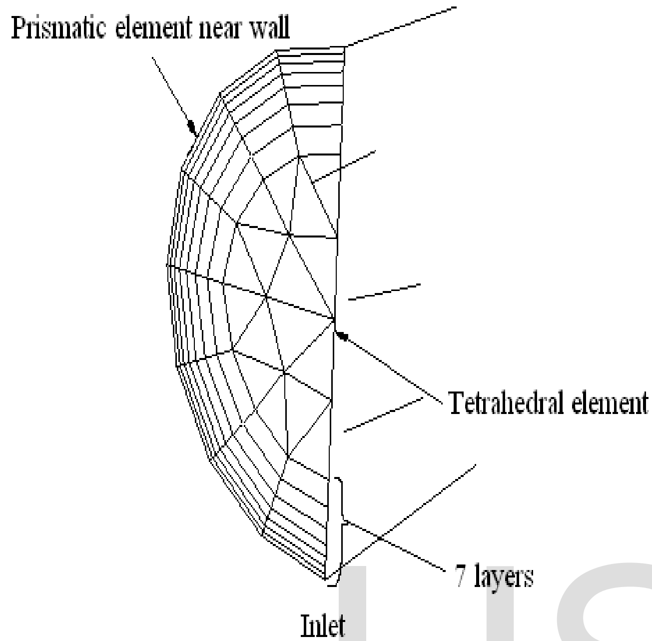
Element Inflation

In near wall regions, boundary layer effects give rise to velocity gradients, which are greatest normal to the surface. Computationally efficient meshes in these regions require that the elements have high aspect ratios. If tetrahedral are used, then a prohibitively fine surface mesh may be required to avoid generating highly distorted tetrahedral elements at the surface.

The AFI volume mesher overcomes this problem by using prisms to create a mesh that is finely resolved normal to the wall, but coarse parallel to it. This mesh arrangement is beneficial for cost effective CFD analysis.

ISSN 2229-5518

The AFI volume mesher can use the local surface element normals to 'inflate' 2D triangular surface elements into 3D 'prism' elements at selected walls or boundaries. You can control the creation of these elements using Inflation Parameters to determine their size and distribution in near-wall regions.



The thickness of the inflation is controlled by the number of layers, the maximum thickness specification, the local element edge length and the inflation thickness multiplier. If the element edge length changes in the region of the inflation layer, due to say a Mesh Control, then it is possible that the inflation thickness will not be constant over the inflated surface. In this project inflation of 7 layers is specified.

4.1 RESULTS AND DISCUSSION

The process of determining the degree to which a model is an accurate representation of the real world from the perspective of the intended uses of the model is validation. It is not possible to validate the entire CFD code. Applying the code to flows beyond the region of validity is termed prediction.

Validation examines if the conceptual models, computational models as implemented into the CFD code, and computational simulation agree with real world observations. The strategy is to identify and quantify error and uncertainty through comparison of simulation results with experimental data. The experiment data sets themselves will contain bias errors and random errors, which must be properly quantified and documented as part of the data set. The accuracy required in the validation activities is dependent on the application, and so, the validation should be flexible to allow various levels of accuracy.

The approach to Validation assessment is to perform a systematic comparison of CFD simulation results to experimental data from set increasingly complex cases.

Zeta calculation

The pressure drop across the valve has to be measured for the flow resistance coefficient calculation. This is accomplished by placing two planes in the fluid domain, which actually represents the inlet and the outlet of the valve. These two planes are created at either side of the valve and are separated by the valve length

For full open condition:

Pressure at inlet (plane1) = 887.695Pa
Pressure at outlet (plane2) = 149.12Pa

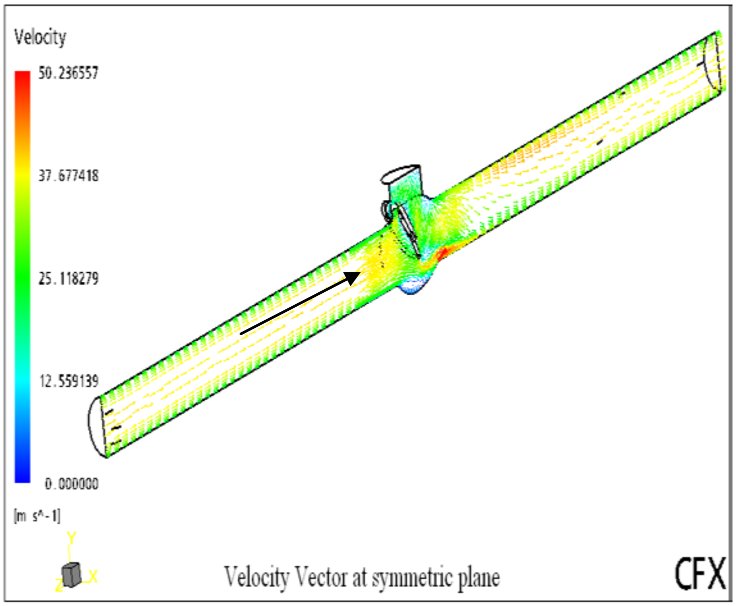
Pressure loss Δp = 887.695-149.12
= 738.575 Pa
Velocity at inlet = 32.12 m/s

Dynamic head = $1/2*\square v^2$
= $1/2*1.0641*32.12^2$
= 548.91 Pa
Flow resistance Coefficient = Pressure Loss/Flow dynamic head
= 738.575/548.91
= 1.345
Experimental Value of Zeta = 1.367

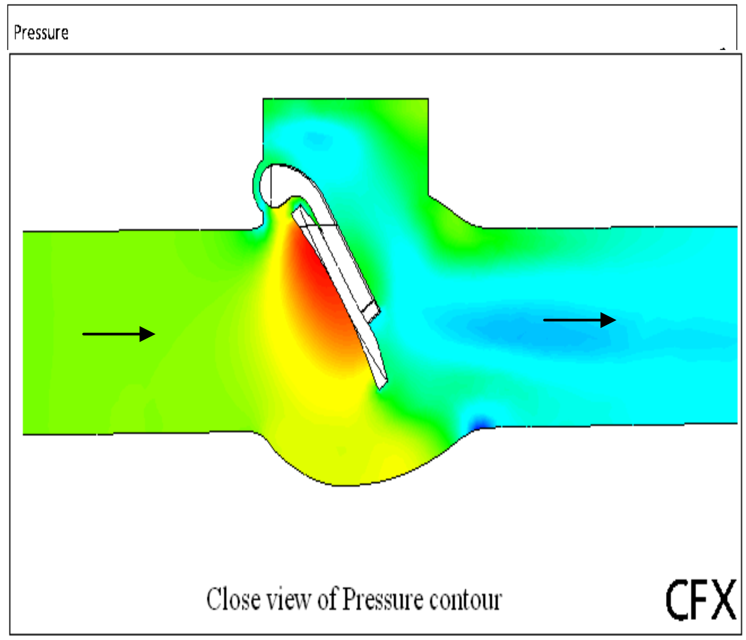
Percentage Error =abs(1.345-1.367)/1.367 *100 = 1.609 %

ISSN 2229-5518

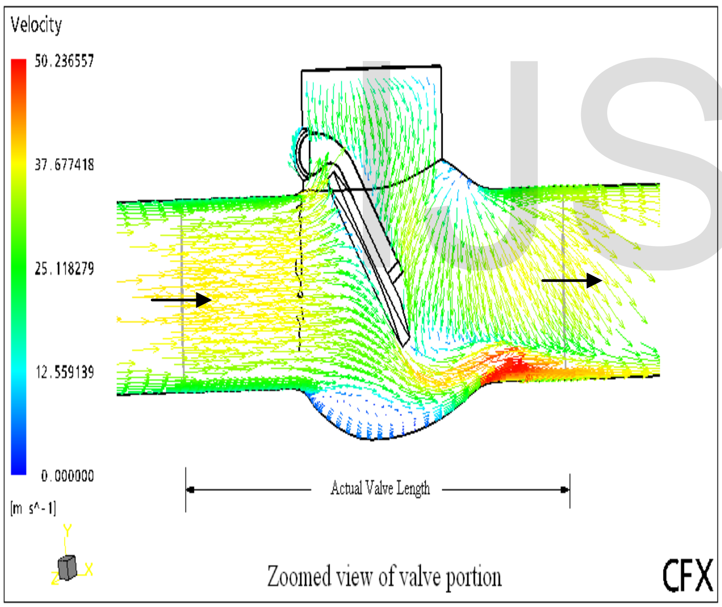
4.2 Result Plots



Velocity Vector at symmetric plane



Close view of Pressure contour



Zoomed view of valve portion

Plane 2

The Closer View Of The Visual Shows The Recirculation In The Bottom Portion Of The Valve,Due To The Flow.

ISSN 2229-5518

The above visual shows the variation pressure along the length of the valve,it also shows the distance up to which the variation of pressure is felt due to the flap.
The above plot shows the conversion of kinetic energy transformation into pressure energy due the resistance created by the flap against the flow.

Evaluation of Zeta for different flow rates

The value of Zeta is tested for various Mass flow rates through the valve at 35^0 degree opening of flap, the value remained constant, thus proving that Zeta is constant for a particular opening of the Flap also a geometry dependent factor not flow dependent.

Mass Flow rate(kg/s)	2.051	1.558	0.643
Zeta	1.341	1.347	1.345

Mesh	Grid points	Zeta
I	21668	1.352
II	24420	1.345
III	27960	1.341

Grid convergence is the term used to describe the improvement of results by using successively smaller cell sizes for the calculations. A calculation should approach the correct answer as the mesh becomes finer, hence the term grid convergence.

The normal CFD technique is to start with a coarse mesh and gradually refine it until the changes observed in the results are smaller than a pre-defined acceptable error. There are two problems with this approach. Firstly, it can be quite difficult with CFD software to obtain even a single coarse mesh result for some problems (particularly when time is pressing). Secondly refining a mesh by a factor of 2 can lead to a 8-fold increase in problem size so even more time is needed. Thus very high computational resources are needed as you proceed for small cell sizes. This is clearly unacceptable for software intended to be used as an engineering design tool operating to tight production deadlines. These and other issues have added greatly to the perception of CFD as an extremely difficult, time consuming and hence costly methodology. Yet a compromise between accuracy and computational resources can be achieved to get an accurate result

Mesh is made finer by decreasing the element edge length .so the number of nodes becomes increased. A point is reached when the increase in the size of the mesh does not yield proportional accuracy. At that point the mesh refining is stopped since further refining increases computational resources with no increase in accuracy.

Mesh Independency Test For full open condition

The Mesh independency study was conducted for 35 degree opening of flap.Three different meshes were taken and analysed for Zeta.The results obtained are tabulated above.From the results it was found that for mesh I and mesh II the value of zeta
varies by a significant amount.But at the same time the difference in value of zeta between the mesh II and mesh III doesnot vary much.Thus the dependency of mesh parameters is upto the II mesh only,afterwards the value of zeta remained almost constant
,thus the problem of mesh depndency stops at second mesh itself.

Conclusion

ISSN 2229-5518

The flow resistance coefficient of the 16”,150 pressure class valve for full opening(35^0 opening of flap) condition was found out using CFD analysis.The value is compared with experimental value and it was found that the deviation is less than 2%.The result is further more made accurate by conducting mesh independent study.Similarly the flow resistance coefficient is evaluated by varying the flap in five degree decrements i.e.,30^0,25^0,20^0.It is known fact that ,as the valve closes the pressure drop will be more and hence the value of zeta also gets increases.The results obtained also proved this clearly.

Thus the CFD which was considered to be solution to high end problems such as combustion also be used for many industrial problems such the coefficient evaluation .

References

[1] Rexroth, 4/3-, 4/2- Und 3/2-Wegeventil Typ WE 10/ Druckausgabe: 1.78’’ (011978).

[2] Rexroth Bosch Group Catalogue: 4/3, 4/2 and 3/2 directional valve 1/14 withwet-pin AC or DC solenoids, Type WE10, RE 23327/08.08., Replaces: 02.03(2010).

[3] Catalogue HYDAC: directional spool valve solenoid actuated, direct acting4WE10, E.5.220.0/04.11 (04 2011).

[4] ARGO–Hytos, Directional control valves solenoid operated, RPE4-10, HA 4039(07 2009).

[5] Catalogue Duplomatic Oleodinamica: DS5 Solenoid operated directionalcontrol valve series 12, 41 310/211 ED (08 2011).

[6] Parker, Hydraulic cartridge systems threaded cartridge valves and integratedhydraulic products, HY15-3502/US (07 2009).

[7] EATON, Vickers, Directional controls, wet armature solenoid operateddirectional control valves, Model DG4V-5, 20 series, 5069.00/EN/0497/A (052000).

[8] EATON, Vickers, Solenoid Operated Directional Valves Catalog, DG4V3-70, V VLDI-MC008-E (08 2008).

[9] PONAR Wadowice, Catalogue: directional spool valve type WE10 electrically
operated, WK 499 495 (04 2012).

[10] Valdés JR, Miana MJ, Núñez JL, Pütz T. Reduced order model for estimation offluid flow and flow forces in hydraulic proportional valves. Energy ConversManage2008;49(6):1517–29. http://dx.doi.org/10.1016/j.enconman.2007.12.010.

[11] Amirante R, Vescovo GD, Lippolis A. Flow forces analysis of an open centerhydraulic directional control valve sliding spool. Energy Convers Manage2006;47(1):114–31. http://dx.doi.org/10.1016/j.enconman.2005.03.010

[12] Amirante R, Moscatelli P, Catalano L. Evaluation of the flow forces on a direct(single stage) proportional valve by means of a computational fluid dynamicanalysis. Energy Convers Manage 2007;48(3):942–53. http://dx.doi.org/10.1016/j.enconman.2006.08.024.

[13] Amirante R, Vescovo GD, Lippolis A. Evaluation of the flow forces on an opencentre directional control valve by means of a computational fluid dynamic analysis. Energy Convers Manage 2006;47(13-14):1748–60. http://dx.doi.org/10.1016/j.enconman.2005.10.005.

[14] Domagala M, Lisowski E. Optimization hydraulic components using fluid–solidinteraction simulation. In: Proceedings of 4th FPNI-PhD symposium, Florida –Sarassota; 2006. p. 151–9.

[15] Domagala M. CFD analysis of a flow control valve. In: Proceedings of 5th FPNI-PhD Symposium, Kraków; 2008. p. 445–450.

[16] Vescovo GD, Lippolis A. Three-dimensional analysis of flow forces ondirectional control valves. Int J Fluid Power 2003;4(2):15–24.

[17] Vescovo GD, Lippolis A. Optimization of flow through a pneumatic controlvalve using CFD analysis and experimental validation. Fluid Power2007;1:31–41.

[18] ISO 4401:2005 Hydraulic fluid power – Four-port directional control valves –Mounting surfaces (2005).

[19] Bosch-Rexroth, Catalogue: 3/2, 4/2 and 4/3 directional valves, internally pilotoperated, externally pilot operated re 24751/08.08’’ (2010).

[20] ANSYS/Fluent: Users guide, 13th ed.; 2011.

[21] Launder BE, Spalding DB. The numerical computation of turbulent flows.Comput Methods Appl Mech Eng 1990:269–89.

[22] Pan X, Wang G, Lu Z. Flow field simulation and a flow model of servo-valve spool valve orifice. Energy Convers Manage 2011;52(10):3249–56. http://dx.doi.org/10.1016/j.enconman.2011.05.010.

[23] Launder BE, Spalding DB. Lectures in mathematical models ofturbulence. London, England: Academic Press; 1972.

[24] Lisowski E, Rajda J. CFD analysis of pressure

ISSN 2229-5518
loss during flow by hydraulicdirectional control valve constructed from logic valves. Energy ConversManage 2013;65(0):285–91.
http://dx.doi.org/10.1016/j.enconman.2012.08.015.

[25] Chattopadhyay H, Kundu A, Saha BK, Gangopadhyay T. Analysis of flowstructure inside a spool type pressure regulating valve. Energy ConversManage 2012;53(1):196–204.
http://dx.doi.org/10.1016/j.enconman.2011.08.021.

[26] Shih T-H, Liou WW, Shabbir A, Yang Z, Zhu J. A new k _e eddy viscosity modelfor high reynolds number turbulent flows. Comput Fluids 1995;24(3):227–38.http://dx.doi.org/10.1016/0045-7930(94)00032-T.

[27] ISO 6403:1988 Hydraulic fluid power – valves controlling flow and pressure –test methods.