

The Numerical Simulation of Hydraulic Manifold Internal flow field based on FLUENT

Zhengkun Wu

Mechanical Engineering College,
Yangzhou University
Yang Zhou, P.R. China
zhengkunwu@sina.com

Yihong Zhao*

Mechanical Engineering College,
Yangzhou University
Yang Zhou, P.R. China
yihongzhao99@sina.com

Jing Mei

Mechanical Engineering College,
Yangzhou University
Yang Zhou, P.R. China

Abstract—According to the hydraulic integrated piece in project, it use pre-treatment software Pro/E which establish its internal torrent three-dimension model, and it use ICEM software which establish its model boundary conditions and mesh. Last simulated structure pipeline using software provided by FLUENT turbulence model. In view of the analysis of pressure, velocity and streamlines of the pipeline, we found the reasons for the energy loss in the Manifold internal flow. By measures to increase the technical holes size, reducing the number of technical hole and right-angle turn structure can achieve the purpose of reducing energy loss in Manifold internal flow passage, and it provided theoretical basis for Manifold internal structure optimization.

Keywords- hydraulic manifold; FLUENT; flow field

I. INTRODUCTION

The unreasonable structural design of hydraulic integrated block caused a large proportion of total energy loss in hydraulic system. As a result of hydraulic integrated block takes that kind of distributed generation with integrated block connected inside flow passage, it is widely used in today's hydraulic system. Therefore, to improve the energy utilization rate of hydraulic integrated block has become one of the most important research directions. In order to improve the performance of the hydraulic integrated block, the author mainly analysis the major reason of pressure loss of hydraulic integrated block in the actual working conditions of hydraulic integrated block, so as to provide the basis for the optimization design [1].

II. MODE OF HYDRAULIC MANIFOLD PHYSICAL

In an engineering design, it uses pre-treatment software Pro/E which establishes its solid model as shown in Fig. 1. In this paper, A-B flow passage is taken as an example. The flow is made from the seven pipelines intersect (show on the left in Fig. 2).

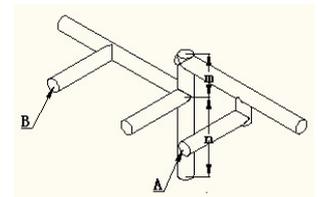
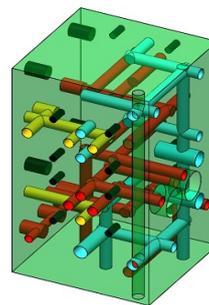


Fig 1. Hydraulic integrated block solid model Fig 2. Flow passage structure

III. HYDRAULIC MANIFOLD BLOCKS OF INTERNAL FLOW MATHEMATICAL MODEL

For hydraulic manifold blocks of tube 3D fluid flow, fluid for often flow state, hydraulic manifold blocks the flow characteristics inside liquid is usually on the turbulent flow [2]. Therefore, fluid flow by of conservation of mass and momentum conservation the domination of the vectors. Fluent in a right angle turn to pipe in most cases for turbulent flow, under the κ - ϵ turbulent model control.

1) Mass conservation equation:

$$\frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0 \quad (1)$$

Type ρ for density; u , v and w for vector in the speed x , y and z the direction of weight.

2) Momentum conservation equation (Navier-Stokes equation):

$$\frac{\partial(\rho u_i)}{\partial t} + \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial(p)}{\partial x_i} + \frac{\partial}{\partial x_j} (\mu \frac{\partial u_i}{\partial x_j} - \overline{\rho u'_i u'_j}) + S_i \quad (2)$$

Type ρ for fluid density; u_i for the average speed, $i = 1, 2, 3$; u_j for the average speed, $j = 1, 2, 3$; p for instantaneous pressure; μ or dynamic viscosity; $-\overline{\rho u'_i u'_j}$ or Reynolds stress items; S_i for the original item.

3) The standard $\kappa - \varepsilon$ equation:

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \quad (3)$$

Type μ_t for power in viscosity; C_μ for experienced constant; C_μ here take = 0.09; k for turbulent kinetic energy; ε for turbulent dissipation rate.

4) Turbulent kinetic energy transport equations:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho k u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + G_k - \rho \varepsilon \frac{\mu_t}{\sigma_k} \quad (4)$$

Type G_k for the average speed of gradient of a turbulent kinetic energy k produced; $G_k = \mu_t \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \frac{\partial u_i}{\partial x_j}$, $\sigma_k = 1.0$.

5) Dissipation rate ε transport equations:

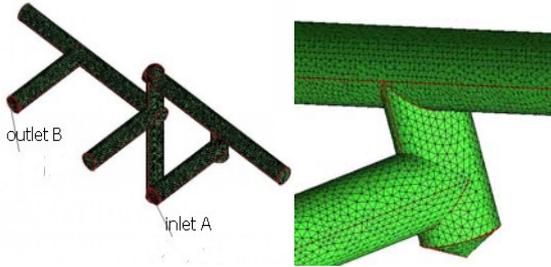
$$\frac{\partial(\rho \varepsilon)}{\partial t} + \frac{\partial(\rho \varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \frac{C_{1g}}{k} C_k - C_{2g} \rho \frac{\varepsilon^2}{k} \quad (5)$$

Type C_{1g} , C_{2g} , σ_ε for experience constant, $C_{1g} = 1.44$, $C_{2g} = 1.92$, $\sigma_\varepsilon = 1.3$.

IV. DOING GRID AND BOUNDARY CONDITIONS

FLUENT in before processing software Pro/E, to passage A-B modeling, again will be built good model by mesh division of import ICEM[3], the grid units used by Tet/Hybrid, grid type using TGird [4], the other to default. Produce grid unit for a total of 288385, generating grid model as shown in Fig. 3 below. Use output command liudao.msh files.

This paper mainly involves the following boundary conditions: pipeline inlet defined as mass-flow-inlet, its a value of 0.45 kg/s; Outlet pipeline defined as pressure-outlet, its a value of 1.2 e+07pascal, other face set to fixed Wall [5].



(a) The grid of whole flow passage (b) The grid of part flow passage
Fig 3. The structure of grid

V. FLOW FIELD PARAMETERS SETTINGS

In Fluent software, import liudao.msh documents, use check command check grid, the minimum grid volume greater than zero, meet the requirements. In the solver, choice of Segregate and Implicit, and other keep default state. Set the hydraulic oil density for 900 Kg/m², dynamic viscosity coefficient is 0.0405pa.s, flow for 32 L/min. Finite element method of volume of integrated internal flow numerical model,

and choose the standard turbulent model and SIMPLE algorithm simulation to control equation [6-9].

VI. THE SIMULATION RESULTS AND ANALYSIS

A. Integrated block flow field analysis

Fig. 4 for flow pressure contour image, liquid flow through a right angle in each turn, will produce approximately 0.1MPa pressure loss, inlet-outlet pressure differential of 0.723MPa, as is shown in table 1, the entire port there were five bends, therefore, right Angle corners structure is the integration of internal fluid flow pressure loss of main factors.

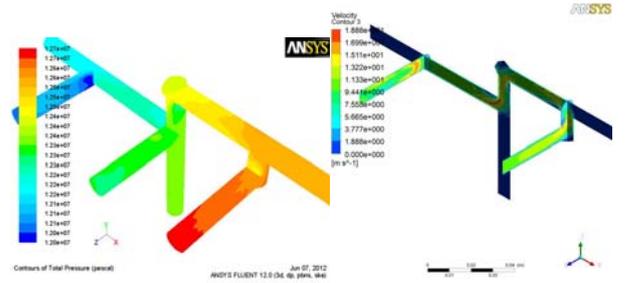


Fig 4. Pressure contour Fig 5. Velocity contour

Fig. 5 for the streamlines image, because the craft hole C chamber should n section is the non-mainstream passage, the internal fluid flow speed is low, in the cavity formed flow sluggish check area, in order to overcome the oil flow delay stop area viscous force produce energy loss.

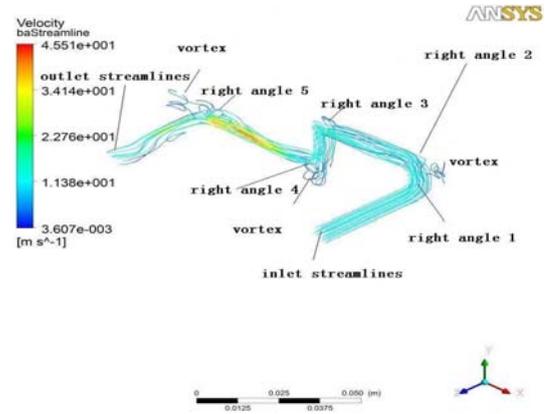


Fig 6. Streamlines

Fig. 6 for streamlines image, the oil into integrated piece of inlet passage, the velocity distribution basic did not change, streamlines for parallel lines, fluid flow at this time in a stable flow state. When the oil flowing through orthogonal corner 1 and 2 right angle corner, streamlines direction extreme changes happened, streamlines in turning suddenly contraction, in A right angle corners structure outside of the most densely populated, and fluid flow structure in the inside of the turn to produce separate A vortex area, as shown in Fig. 7 shows. In A separate area within the vortex, streamlines more sparse, the pressure is low, vortex center speed is near to zero, to form the

flowing sluggish check area, fluid flow to overcome delay in the area of the hydraulic oil viscosity force and generate energy loss function, at the same time vortex itself will also has A rotating the energy. Fig. 6 right corner by the two places of the flow chart can see that in the process of turning craft also has a swirl-producing, vortex zone within the fluid flow does irregular rotation, collision and return movement, the reverse flow of movement and mainstream movement mix together, further disturbance mainstream movement, and consumption s mainstream motion energy, thereby causing the fluid flow pressure loss [10].

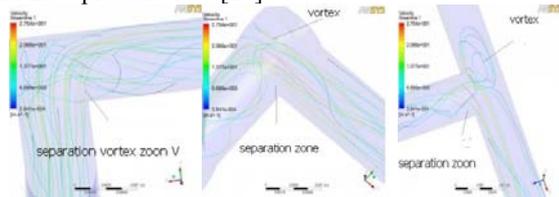
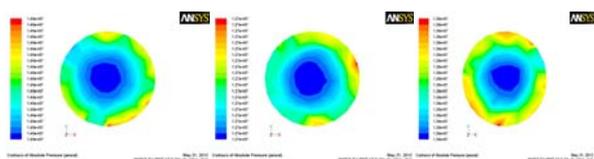


Fig 7. Separation vortex zone of 2,3,5 right-angle

A. Craft holes diameter of the pipes to the influence of flow field

Because the pressure loss of numerical integration device is a reflection of the pieces type design quality and level of the one of the important symbol, and therefore the inlet and outlet flow pressure loss from this Angle discusses different process hole diameter of flow of fluid flow characteristics, the influence of the internal pipeline of integrated blocks provide design basis.

Pipeline craft hole C as the research object, its diameter were set to 6 mm, 8 mm and 10 mm, length remain unchanged. Using the finite volume method (FVM) for integrated piece of internal passage flow field a numerical simulation ,craft hole C in different diameter under inlet pressure contour inlet Fig. 8 (a), (b) and (c) as shown.



(a) hole diameter 6mm (b) hole diameter 8mm (c) hole diameter 10mm

Fig 8. Pressure contour of inlet.

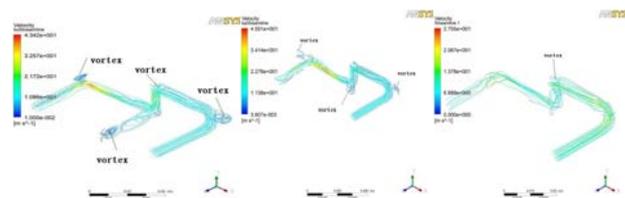
From the Fig. 8 (a), (b) and (c) can be found, inlet pressure face from central to circle pressure value is more and more big, the center area in a relatively area of low pressure, this is because the fluid flow in passage mediocre obsolete, the viscosity fluid flow and still attached to the wall of the solid surface contact, because the flow layer in the existence of the relative movement between the friction within, the greater the velocity gradient, by friction within the greater the, this is Newton's laws of internal friction[11-12]. In the inlet pressure value get face, for accurate quantitative shows the variation of the pressure loss relations in Fluent in the report are on the menu import the average pressure, combined with the outlet pressure, calculated in different craft hole diameter C of the inlet and outlet differential pressure, as is shown in table 1.

TABLE I. CRAFT HOLE IN THE DIFFERENT DIAMETER C IMPORT AND EXPORT THE DIFFERENTIAL PRESSURE

C hole diameter	Inlet pressure	Outlet pressure	Differential pressure
6mm	Area-Weighted Average Total Pressure (pascal) in 13827362	1.2e+07pascal	1027362pascal
8mm	Area-Weighted Average Total Pressure (pascal) in 12723789	1.2e+07pascal	72328pascal
10mm	Area-Weighted Average Total Pressure (pascal) in 12639242	1.2e+07pascal	639242pascal

From table 1 shows, craft hole diameter C for 6 mm respectively, 8 mm and 10 mm inlet and outlet pressure differential when were 1.027MPa, 0.723MPa, 0.639MPa, in general, present a kind of decline trend, the full explanation pipe diameter can effectively reduce the pressure increases the loss, and the increase of aperture can also improve the flow characteristics of fluid flow.

From the Fig. 9 (a), (b) and (c) can be found, the craft hole diameter of fluid flow characteristic also has certain influence, craft hole is too small, passage streamlines very concentrated, speed can not make full play, so as to form the eddy current.



(a)hole diameter 6mm (b)hole diameter8mm (c)hole diameter10mm

Fig 9. Streamlines

The Fig. 9 visible, craft hole diameter C for 6 mm, 8 mm and 10 mm within the eddy current flow when with the decrease of the aperture and tend to reduce or not obvious. That increase craft holes diameter can be effectively improve the flow of fluid flow stability and reduce the loss of the kinetic energy.

VII. CONCLUSION

This paper by using numerical analysis method, the application of hydraulic integration software FLUENT of the internal flow is analyzed. In the design of hydraulic integration process, had better use precision casting passage, this can avoid right angle turning, and eliminate the vortex formed after turning, reduce energy losses and fluid noise. In order to reduce the flow of liquid energy loss, in the design of integrated craft holes should decrease as far as possible to reduce the risk of vortex number. Because craft holes cavity is mainstream way, its holes cavity pressure oil low speeds, formed the flow sluggish area, the oil flow to overcome delay of the cohesive force, must want to consume some energy. In addition, the craft hole is too small, can lead to streamline very concentrated in port, speed can not make full play, cause tube of energy shock and loss, so that the design process hole, in conditions allow conditions appropriate amplification craft hole diameter.

REFERENCES

- [1] D. R. Gao, Y.Q. Wang. Hydraulic integrated block bending pipe flow field data calculation and analysis [J]. Machine Tool And Hydraulic Press, 2001, (6) :34-35.
- [2] W.F.Hughes,J.A.Brighton.Fluid dynamics [M]. Y. H. Xu, L. G. Xu, Y. L. Chen, al. Beijing: Science And Technology City Press, 2002.
- [3] G. Q. Hu, G. H. Zhang, C. G. Wu. LDA and κ - ϵ turbulent model research hydraulic manifold blocks the flow field [J]. Journal Of Chengdu University of Science And Technology, 1996.
- [4] L. H. Xie, X. Y. Zhao, J. M. Zhang. ANSYS CFX fluid analysis and simulation [M]. Beijing: Electronic Industry Press, 2012.1.
- [5] B. B.Ji , J. P. Chen.ANSYS ICEM CFD grid partition example explanation [J]. Water Conservancy And Hydropower Press, 2012.1.
- [6] F. Jiang, P. Huang.FLUENT in advanced application example analysis [M]: Tsinghua University Press, 2008.
- [7] Z. Wen, L. C. Shi, Y. R. Ren .FLUENT calculation tutorial application [M] : Tsinghua University Press, 2009.
- [8] Y. Yu, J.M. Zhang, L.T. Jiang. FLUENT with advanced tutorial introduction [M]. Beijing: Beijing University Press, 2008.9.
- [9] F. J. Wang. Computational fluid dynamics analysis-CFD software principle and application [M]. Beijing: Tsinghua University Press, 2004.
- [10] J. M. Du, B. R. Li. A system hydraulic manifold blocks the flow characteristic analysis of liquid flow [J]. Machine Tool And Hydraulic Press, 2010, 38 (13) : 131-133.
- [11] FLUENT6.3 TutorialGuide. FLUENTInc., 2006.
- [12] M. Z. Chen.Viscosity fluid dynamics [M]. Beijing: Higher Education Press, 1993.