Numerical Simulation on Cold-Flow Impact of the Soft Pulse Separation Device in Dual Pulse Solid Rocket Motor

Wanli Liu\textsuperscript{1,a}, Xiong Chen\textsuperscript{1,b}, Yingkun Li\textsuperscript{1,c}

\textsuperscript{1}School of Mechanical Engineering, NUST, Nanjing 210094, Jiangsu, China

E-mail: \textsuperscript{a}liuwanli203@163.com, \textsuperscript{b}chenxiong@njust.edu.cn, \textsuperscript{c}liyingkun@njust.edu.cn

Keywords: Dual Pulse Motor; Cold-Flow Impact; Soft Pulse Separation Device; Solid Rocket Motor.

Abstract. In this paper, the fluid solid coupling in the cold-flow impact process of the soft pulse separation device of dual pulse solid rocket motor is numerically simulated by using ANSYS Workbench software. This paper mainly analyzes the flow characteristics of the flow field near the interlayer, deformation and stress change of interlayer under the cold-flow impact, and obtain the time of the interlayer failure. Finally, the numerical simulation results are compared with the experimental results of cold-flow impact. Finally, at 8.41 ms, the stress at the prefabricated defect of the interlayer reaches the tensile strength, and the pressure at breaking is 0.745 MPa.

1. Introduction

As a new research achievement of solid rocket motor technology, dual pulse solid rocket motor can make up for the shortcomings of traditional solid rocket motor. The solid rocket motor combustor is divided into two separate combustion chambers by a pulse separation device with flame retardant and heat insulation function. The two combustion chambers share a common nozzle, and each combustion chamber is ignited successively to start, thus achieving thrust control. Reasonable distribution of engine thrust and pulse time interval can realize the optimal control of missile trajectory and the optimal use of energy, and greatly expand the application scope of solid rocket motor, thus comprehensively improving the range, accuracy, stealth performance and maneuverability of various missile weapon systems.

In 2004, S Schilling, P Trouillot et al.[1] carried out a series of tests on a 120 mm dual-pulse engine. The resulting debris is quickly melted by the gas and no injection is formed. Experiments show that the performance of the engine and its components is reliable.

In 2006, Stadler L et al.[2]designed a prototype of dual pulse motor for short-range air defense missile LFK-NG, and carried out ground static test and two flight tests, including individual test of each part.

In 2007, Chiyako Mihara, Tomohiro Kishida[3-5] used EPDM as interlayer to design an interlayer dual-pulse engine with axial and radial mixing, and applied for many patents for invention.

In 2008, a new generation of LFK-NG air defense missile was exhibited at the Berlin Aerospace Exhibition. The double pulse engine was used as its power unit and passed two experiments.[6]
In 2012, Javed et al. [7] used commercial software CFX to simulate the three-dimensional internal flow field of dual pulse motor. The metal diaphragm was used in the pulse separation device, and the diaphragm was assumed to be completely broken when the second pulse was working. The second pulse propellant fuel gas flows through the air guide holes on the support plate and enters the first pulse combustor, forming a large recirculation zone in the combustor. At the same time, the pressure, velocity and temperature on the engine axis are analyzed, and the pressure loss of the fuel gas flowing through the channel of the pulse isolation device is studied.

In 2015, Cho et al. [8] based on commercial software ANSYS FLUENT V14.5, numerically simulated the internal flow field of the dual-pulse engine, studied the internal flow field characteristics of the engine under cold and real gas flow in the combustion chamber, and analyzed the influence of the channel aperture of the isolation device on the pressure, temperature and Mach number in the combustion chamber.

2. Basic governing equation of cold-flow impact process

The expression of the compressible unsteady Navier-Stokes equation is as follows:

$$\frac{\partial (\rho \phi)}{\partial t} + \nabla (\rho \phi \mathbf{v}) = \nabla \cdot (\Gamma_\phi \nabla \phi) + S_\phi$$

(1)

In this formula, $\phi$ represents a variable, $\Gamma_\phi$ is diffusion coefficient, $S_\phi$ is source term, because this paper mainly studies the flow field motion of the dual pulse motor under the cold-flow impact, without considering the process of propellant charging and combustion, the source term in the equation is neglected.

The expression of the continuity equation is as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_k)}{\partial x_k} = S_{\rho}$$

(2)

The expression of the momentum equation is as follows:

$$\frac{\partial (\rho u_k)}{\partial t} + \frac{\partial (\rho u_k u_j)}{\partial x_j} + \frac{\partial P}{\partial x_k} = \frac{\partial (\rho \mu \frac{\partial u_k}{\partial x_j})}{\partial x_j} + S_{f_i}$$

(3)

The expression of the energy equation is as follows:

$$\frac{\partial (\rho E)}{\partial t} + \frac{\partial (\rho u_k H)}{\partial x_k} = -\frac{\partial}{\partial x_k} \left( u_j \frac{\partial E}{\partial x_j} \right) + \frac{\partial \varepsilon}{\partial x_k} + S_h$$

(4)

In these formulas, $\rho$, $u_k$, $p$, $E$, $H$ is the density, velocity component, pressure, total Energy, total enthalpy, $S_{\rho}$ is mass source term of gas generated by propellant burning, $S_h$ is energy source term of gas generated by propellant burning, $S_{f_i}$ is momentum source term of gas generated by propellant burning.

The expression of the turbulence model is as follows:

$$\frac{\partial (\rho_{en})}{\partial t} + \frac{\partial (\rho_{en} u_k)}{\partial x_k} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\kappa} \frac{\partial u_k}{\partial x_j} \right) + G_{en} + G_b - \rho \varepsilon - Y_{en} + S_{\mu}$$

(5)

$$\frac{\partial (\rho_{en})}{\partial t} + \frac{\partial (\rho_{en} u_k)}{\partial x_k} = \frac{\partial}{\partial x_j} \left( \mu + \frac{\mu_t}{\kappa} \frac{\partial u_k}{\partial x_j} \right) + C_{1e} \frac{\varepsilon}{\kappa} (G_{en} + G_b - C_{2e} \rho \varepsilon^2) + S_{\varepsilon}$$

(6)

$$G_{en} = \mu_t \frac{\partial u_k}{\partial x_j} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial u_j}{\partial x_j} \right)$$

(7)

$$G_b = \beta \frac{\mu_t}{\kappa} \frac{\partial \varepsilon}{\partial x_j}$$

(8)

$$\mu_t = \rho C_p \frac{\kappa^2}{\varepsilon}$$

(9)
In these formulas, $G_k$ represents the turbulent energy generation term due to the average velocity gradient, $G_b$ is a generation term for turbulent flow energy caused by buoyancy, $Y_{H}$ is dissipation caused by pulsating expansion in a compressible fluid, $\beta$ is expansion coefficient, $P_{Pr}$ is turbulence Prandt constant, $C_\mu$ is empirical constant.

3. Numerical simulation model
This paper mainly studies the working process of dual pulse motor interlayer under the cold-flow impact. Because only the working process of the second pulse under the cold-flow impact is studied, the situation and influence of the first pulse are not considered. Fig. 1 is the diagram of computational domain model.

![Figure 1. Computational domain model](image)

The deformation and displacement of the charge of the second pulse propellant in the cold-flow impact process are not considered. The fluid solid coupling process of the fluid field acting on the interlayer in the vicinity of the interlayer is studied. Because the simulation model is symmetrical, it is simplified to symmetrical model, Fig 2 is the schematic diagram of the simplified calculation model.

![Figure 2. Schematic diagram of the simplified calculation model](image)

The second pulse grain is a single-hole tubular grain, the inner diameter is 29.6mm, the distance from monitoring point A to the surface of the interlayer is 15 mm, the distance between the interlayer and the vertical end face of the second pulse grain is 3 mm. The material of interlayer is EPDM, the thickness is 3mm, and the diameter is 74.8mm. The outer surface of the interlayer is provided with a cross-shaped prefabricated defect along the center of the circle, with a depth of 1 mm and a width of 0.6 mm. The interlayer is swelled and deformed under the cold-flow impact. When the second pulse is working, the swelling and deformation of the interlayer under the cold-flow impact, and when the pressure reaches a certain level, the interlayer ruptures at the prefabricated defect. If there is no prefabricated defect, the rupture time of the interlayer will be prolonged and the rupture position can not be predicted, which may affect the normal operation of the dual pulse motor.

Boundary conditions and initial conditions
In this paper, the initial pressure and temperature of the simulation model are atmospheric pressure and normal temperature, in which the pressure is 101325Pa, the temperature is 300K and the initial velocity of gas in all directions is zero.

Set the fluid domain entrance as the mass flow rate boundary condition. The pressure curve of
monitoring point A of the second pulse combustor with time is obtained by the cold-flow impact test. The pressure change curve of point A in the second pulse combustor with time was measured by the cold-flow impact test, and the mass flow change curve with time was obtained by simulating the free volume method. Finally, UDF (user-defined function) is compiled by FLUENT software as the entry boundary condition of fluid domain.

In the calculation area of the flow field, external surface of the second pulse propellant charge, the second pulse combustion chamber wall and fluid solid coupling interface are the adiabatic solid wall boundary conditions. According to the boundary layer theory, the phase gradient of the fixed wall pressure method is equal to zero. According to the gas equation of state, the normal gradient of wall density on adiabatic solid wall is equal to zero. The fixed wall satisfies the condition of no slip, and the velocity in each direction is equal to zero.

In order to save calculation time, without affecting the calculation results, the computational domain model takes one half of the original model and sets the fluid computational domain and the symmetrical surface of the interlayer as symmetrical surface boundary conditions respectively.

The side wall of the disc with soft pulse separation device is fixed constraint end face, and the interlayer is EPDM with density of 1100 kg/m$^3$, the parameters of the constitutive model are shown in Table 1.

| Mooney-Rivlin 5 Parameter | Value/MPa |
|---------------------------|-----------|
| Material Constant $C_{10}$ | 27.58     |
| Material Constant $C_{01}$ | 11.61     |
| Material Constant $C_{20}$ | -8.53     |
| Material Constant $C_{11}$ | -31.37    |
| Material Constant $C_{02}$ | -7.27     |

The inner flow field and the contact surface of the interlayer are respectively set as the fluid solid coupling interface. The System Coupling module in ANSYS Workbench is used for data exchange, and the time step is set to $1e^{-5}$s.

4. Results and discussion

From Fig. 3, after the impact of gas on the wall of the fluid domain, the pressure acts on the interlayer through the fluid solid interaction. When the interlayer is subjected to force and swelled, it expands and the fluid domain, which reduces the pressure of the fluid domain and the deformation of the interlayer, and the deformation of the interlayer oscillates under the action of the fluid field. Finally, at 8.41 ms, the stress at the prefabricated defect of the interlayer reaches the tensile strength, and the interlayer is destroyed. At this time, the maximum pressure in the fluid domain is 0.745 MPa.

![Figure 3. Pressure in the region of the flow field at 8.41ms](image)
From zero to 8.49 ms, the maximum pressure is 0.734 MPa, as shown in Fig. 4 (a). The initial monitoring point pressure from high to low is monitoring point 0, monitoring point 3, monitoring point 1, monitoring point 2. The pressure of monitoring point 0 rises first, and the gas sprays into the center of the inner surface of the interlayer from the entrance, which causes the pressure of monitoring point 3 to rise. Then the gas diffuses around the inner surface of the interlayer, which makes the monitoring point 2 and 1 rise in turn. After 1.5 ms, due to the impact of gas and the inner surface of the interlayer, the pressure of the monitoring point 3 rises fastest, as shown in Fig. 4 (b).

From Fig. 5, flow domain monitoring point 3 and experimental monitoring point A have the same upward trend. When the flow domain monitoring point 3 reaches 8.41 ms, the interlayer is destroyed and the pressure is 0.734 MPa. When the experimental monitoring point A reaches 8.1 ms, the interlayer is destroyed and the pressure is 0.7987 MPa. The time difference between them is 0.31 ms, and the pressure of flow domain monitoring point 3 is 8.11% lower than that of experimental monitoring point A. It can be seen that the difference between the numerical simulation results and the experimental results is small, which meets the requirements of engineering calculation.

5. Summary
In this paper, the numerical simulation of fluid solid coupling in the cold-flow impact process of soft pulse separation device is carried out by using commercial software ANSYS workbench. The development of flow field and pressure change curve during the cold-flow impact process are analyzed. The failure time and the failure pressure of interlayer are obtained. The results show that the gas flows in from the fluid domain entrance and strikes the center of the inner surface of the interlayer, resulting in an increase in pressure at the center. The gas diffuses from the center to all sides and collides with the wall of the second pulse combustor to increase the gas pressure here. Then vortex is formed around the wall. The central velocity of the vortex rises and the pressure decreases, and the deformation of the interlayer oscillates under the interaction of fluid solid
coupling. Finally, the interlayer is destroyed after the stress of prefabricated defects reaches tensile strength. The convergence section of nozzle is subsonic flow and the exit of nozzle is supersonic flow.

6. References
[1] Schilling S, Trouillot P, Weigand A. On the Development and Testing of a 120 Mm Caliber Double Pulse Motor (Dpm). 2013
[2] L. Stadler, C. Trouillot Pand Rienäcker, et al. The Dual Pulse Motor for Lfk Ng[C]. Aiaa/asme/sae/asee Joint Propulsion Conference & Exhibit, 2006.
[3] Mihara C, Ieki K. Two-pulse rocket motor[P]. United States Patent: 8397486, 2013-3-19.
[4] Mihara C, Suzuki T, Ieki K. Multi-pulse rocket motor and pulse unit thereof[P]. United States Patent: 13/491,725, 2012-12-13.
[5] Kishida T, Tsunezumi Y, Mihara C, et al. Ignition device for two-pulse rocket motor with thermal barrier membrane[P]. United States Patent: 9,371,8012, 016-6-21.
[6] Xiaoyan Hou, Aiping Zhu. SysFla exhibits LFK NG missile [J]. Aerial missile, 2008 (11): 7-7.
[7] Javed A, Manna P, Chakraborty D. Numerical simulation of a dual pulse solid rocket motor flow field[J]. Defence Science Journal, 2012, 62(6): 369-374.
[8] Cho K, Park J, Kim E. Internal flow characteristics of simulated dual pulse rocket motor by using the hot gas and cold gas[J]. Journal of the Korean Society of Propulsion Engineers, 2015, 19(2): 1-8.