Fusion Engineering and Design 129 (2018) 214–220

Contents lists available at ScienceDirect

Fusion Engineering and Design

journal homepage: www.elsevier.com/locate/fusengdes

# Numerical study of transient flow in the preliminary design of fusion power shutdown system

Xiaoli Ren[a,b,c], Jia Zhai[b,c,*], Ge Ren[a,b]

[a] Key Laboratory of Optical Engineering, Chinese Academy of Sciences, China
[b] Institute of Optics and Electronics, Chinese Academy of Sciences, China
[c] University of Chinese Academy of Sciences, China

ARTICLE INFO

*Keywords:*
Transient model
Dynamic characteristic
CFD
FPSS
ITER

ABSTRACT

The FPSS (fusion power shutdown system) provides a safety emergency shutdown system terminating the ITER (international thermonuclear experimental reactor). It has a reservoir of impurity gas and a dedicated tube for the impurity gas injection. In the preliminary design stage, the FPSS is studied as a sealed gas injection system, it is a supersonic and high pressure flow field. The transient flow of the system is analyzed with a commercial CFD code, ANSYS® FLUENT in this paper. To satisfy the functional requirement, the injected gas volume of the VV (vacuum vessel) is predicted from a series of tank pressures in 3s. The influence of initial tank pressures, vessel pressure and flow times of the FPSS is quantified. CFD data is compared with the experimental activities and the discrepancy is acceptable. All the results will be used for optimized design of the FPSS.

## 1. Introduction

The FPSS (fusion power shutdown system) is a safety system. It provides emergency fusion power shutdown immediately for the ITER (international thermonuclear experimental reactor) in any case of severe accident [1]. The FPSS has a reservoir of impurity gas and a dedicated tube for the impurity gas injection [2].

The earliest concept design for the FPSS was given by Yu Yang. The basic configuration of the FPSS was shown and simulations were made to check the feasibility of meeting the response time [3]. However, particular researches are needed to be executed with following reasons: device selection (valves and sensors), pipe structural changes (distribution, length and diameter), source pressure selection and configuration changes.

Computational Fluid Dynamics (CFD) is chosen as an indispensable tool for the design of the FPSS [4]. Significant existing efforts about gas injection system can be referred to study the flow field in this systems.

Sangeun Roh used transient numerical study to investigate a pressurized liquefied natural gas storage tank. It quantified the tank, include its pressure and size, and studied the pressurization procedure [5]. A stream of high pressurized gas through pipes was researched in numerical analyses by Margherita Cadorin. It emphasized the influence of the roughness of the pipe for the flow field [6]. Jacopo De Amicis and C.S. Oon analyzed the steady incompressible laminar flow in coiled pipes. The result predicted that the geometrical parameters of the pipe can influence the critical Reynolds number [7–11]. Fuel jet was one kind of the injection gas. Simulations about its turbulence can obtain the vital parameter like pressure, velocity, density or flow time for the flow state [12–14].

Except the simulation results mentioned above, numerous experimental activities were conducted by scientists and engineers. D.Lucas gathered lots of experimental results to compile a benchmark database for steady evolution flows in a vertical pipe [15]. Tetsuaki Takeda carried out numerical analysis and experiments to investigate helium gas injection of a reverse U-shaped channel [16]. Some contrastive studies with numerical analysis and experiments were performed to calibrate and validate mutually [17–20].

In this study, a gas injection system as a preliminary design for the FPSS is investigated. It is a simple gas dynamic device because the momentum is induced by the pressure difference between a gas storage tank and a VV (vacuum vessel). The principal advantages of this system are absence of any dynamical appliances, and simplicity of design and operation. A series of pressures in the storage tank are investigated for estimating the injected gas volume of the VV. Further, the response time was evaluated which is acquired by the ITER requirement. CFD models are used in transient calculation to solve the mass, momentum and energy conservation equations. A deep analysis of unsteady calculation is carried out to compare with the experimental results.

* Corresponding author at: Institute of Optics and Electronics, Chinese Academy of Sciences, China.
*E-mail address:* un31415926@163.com (J. Zhai).

https://doi.org/10.1016/j.fusengdes.2018.02.096
Received 17 April 2017; Received in revised form 20 November 2017; Accepted 26 February 2018

**Nomenclature**

| | |
|---|---|
| $\rho$ | Density |
| $\kappa$ | Turbulent kinetic energy |
| v | Velocity |
| u | Dynamic viscosity |
| $\varepsilon$ | Turbulent dissipate rate |
| $\tau$ | Viscous stress |
| g | Gravitational acceleration |
| P | Pressure |
| F | Other volume force |
| a | Speed of sound |
| t | Flow time |
| $\Delta t$ | Time step |
| $P_{tank}$ | Absolute tank pressure |
| $P_{vessel}$ | Absolute vessel pressure |
| $V_{tank}$ | Tank volume |
| $L_{pipe}$ | Pipe length |
| $T_{arrive}$ | Arrived time to V |
| $L_{min}$ | Minimal mesh length |

## 2. Numerical calculation

Fig. 1 shows the diagram of the FPSS studied in this paper. It is consisted of a gas storage tank, a long pipe, some monitor and control units. When the pneumatic valve is opened, the gas flows from the pressurized tank to the VV. The tank volume is $1 \times 10^{-3}$ m$^3$, which is the minimum volume with appropriate interfaces and pressure rating. The inner diameter of the pipe is $10.22 \times 10^{-3}$ m and its length is 23.4 m, only the diameter of the pneumatic valve is $7.6 \times 10^{-3}$ m. The structure of the pipe is irregular as it is limited by the practical installation condition. To complete a computational domain and boundary conditions, a VV with a volume of 4.5 m$^3$ and an initial pressure of 0.01 Pa is added in the simulation. The structure and dimensions are referred to the existing VV in the laboratory.

### 2.1. Governing equations

Bernoulli relation is a useful theoretical equation for computing the relationship between pressure, velocity, and altitude of a flow. For gas at standard conditions, a flow can be considered compressible if the dimensionless Mach number is more than 0.3.

Where

$$Ma = \frac{v}{a} \quad (1)$$

The primary parameter correlating the viscous behavior of all newtonian fluids is the dimensionless Reynolds number [21]:

$$Re = \frac{\rho v D}{u} \quad (2)$$

For axial flow through a circular tube, the Reynolds number for transition to turbulence is approximately 2300.

The used turbulence model is the standard k-ε turbulence model. The available turbulence models in the ANSYS® Fluent are k-ε, k-ω, k-kl-ω, SST, SAS, DES, LES, etc., but the two equations models are regarded as the most useful to solve engineering problems. Based on the k-ε turbulence model, the mass, energy, momentum, turbulent kinetic energy (k), and turbulent dissipation rate (ε) conservation equations are written as [22]

$$\frac{\partial(\rho\kappa)}{\partial t} + \frac{\partial(\rho \bar{u}_j \kappa)}{\partial x_j} = \frac{\partial}{\partial x_j}\left(u\frac{\partial \kappa}{\partial x_j}\right) + \frac{\partial}{\partial x_j}\left(\frac{u_t}{\sigma_\kappa}\frac{\partial \kappa}{\partial x_j}\right) + P_\kappa - u\overline{\frac{\partial u_i'}{\partial x_\kappa}\frac{\partial u_i'}{\partial x_\kappa}} \quad (3)$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = C_{\varepsilon 1} P_\kappa \frac{\varepsilon}{\kappa} - \rho C_{\varepsilon 2}\frac{\varepsilon^2}{\kappa} + \frac{\partial}{\partial x_j}\left(\frac{u_t}{\sigma_\varepsilon}\frac{\partial \varepsilon}{\partial x_j}\right) \quad (4)$$

where

$$P_\kappa = -\rho\overline{u_i' u_j'}\frac{\partial \bar{u}_i}{\partial x_j} \quad (5)$$

$$u_t = \rho C_u \frac{\kappa^2}{\varepsilon} \quad (6)$$

The default coefficients for the k-ε turbulence modeling are specified as

$$C_{\varepsilon 1} = 1.44 \quad C_{\varepsilon 2} = 1.92 \quad C_u = 0.09 \quad \sigma_\kappa = 1.0 \quad \sigma_\varepsilon = 1.3$$

### 2.2. CFD simulations

Fig. 2 shows the mesh model for the FPSS using the commercial software package ANSYS® ICEM CFD. Considering the irregular layout of the pipe, a three-dimensional model is developed. Hexahedral meshes are applied to the whole flow domain and the meshe in the pneumatic valve domain are well refined to capture the flow fields accurately.

Simulations are developed with density-based solver in the unsteady formulation. The implicit-time stepping method (dual-time formulation) is chosen for compressible and unsteady flows [22]. The ROD-FDS flux type is selected because it is a default setting and is recommended for transient cases. There is not any inlet or outlet flow and the momentum is generated by the pressure difference, which applies a decreasing pressure source of the upstream. The only boundary condition is the initial pressure in the gas storage tank and the VV. The glossy pipe is chosen to consider the time requirement, so the boundary at the wall is set to the 'no-slip boundary condition'.

The neon tank pressure is from 0.1 MPa to 3.5 MPa. The initial velocity of the neon is about 492.3 m/s with the minimal pressure difference, so compressibility effects must be accounted. The Reynolds number $Re = 1.3 \times 10^5$, thus a turbulent would appear in the flow. The flow domain is neon gas and its density are set to idea gas, temperature 300 K, specific heat Cp 1030 J/kg K and thermal conductivity 0.04794 W/m K. Unsteady assumption is chosen for all the cases until the simulation times satisfy 3s. For the transient calculation, the time step is $1 \times 10^{-5}$ s ($\Delta t = L_{min}/v$), which is sufficient small compared with the minimal mesh length($L_{min} = 9 \times 10^{-3}$ m) and the maximal viscosity.

## 3. Results and discussion

### 3.1. Transient calculation

Fig. 3 plots the turbulence ($p_{tank}$ = 3.5 MPa) at four times in the VV. The system requirement demands the FPSS reacts rapidly and timely, but the pneumatic valve and the irregular pipe limit the response time, so the arrival time for the vessel is observed. When t = 0.18s, there is not any streaming, as seen in Fig. 3(a). The flow has not arrived to the vessel which is still in a vacuum. But then at t = 0.19s, the fluid just reaches at the inlet of the vessel, so the turbulent kinetic energy starts to increase, as the red region seen in Fig. 3(b). The turbulent flow

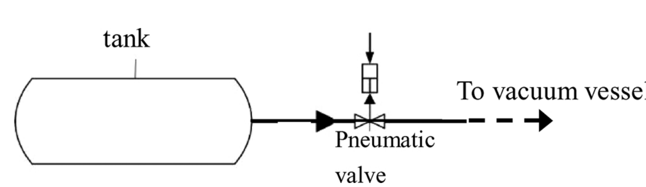


**Fig. 1.** Diagram of the FPSS.