Fire Safety Journal 69 (2014) 126–135

Contents lists available at ScienceDirect

Fire Safety Journal

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

Performance-based analysis of cylindrical steel containment Vessels exposed to fire

Julio Cesar G. Silva, Alexandre Landesmann*, Fernando Luiz B. Ribeiro

Civil Engineering Program, Federal University of Rio de Janeiro (COPPE/UFRJ), POB 68506, Rio de Janeiro 21945-970, Brazil

ARTICLE INFO

Article history:
Received 15 October 2013
Received in revised form
22 August 2014
Accepted 24 August 2014

Keywords:
FEM-analysis
CFD-models
Fire
Structural integrity
Steel containment vessel

ABSTRACT

This paper reports the results of a numerical investigation on the structural safety assessment of a nuclear cylindrical steel containment vessel exposed to an accidental external fire condition. A coupling procedure is proposed, linking the modeling of a nearly compressible flow with input energy given by a combustion model (CFD model) and a structural thermo-mechanical analysis (FEM model), allowing an accurate evaluation of a fluid-thermo-mechanical response for the entire duration of the simulated accident. The time-temperature evolution of the burned gases, resulting from the combustion of a hydraulic leakage pool, is performed by the CFD model. The FEM model is used to obtain (i) temperature variation and, (ii) thermo-mechanical behavior and ultimate strength of the vessel. The obtained results indicate that the suggested methodology can provide reliable fire-safety analyses, ensuring that the main safety (load-bearing and containment) functions of the installation are not impaired during accidental events.

1. Introduction

Fire safety is important throughout the lifetime of a plant, from design to construction and commissioning, during plant operation and in decommissioning [1]. Because fire hazard has been identified as a major contributor to a plant's operational safety risk [2], considerable developments have taken place in recent years in the design for fire safety in operating nuclear power plants, resulting in substantial improvements at many plants [1]. Indeed, the fire safety of Nuclear Power Plants (NPP) has been addressed in some IAEA safety standards publications covering design [3,4] and operation [5,1], as well as further guidance for fire safety assessments [6–9,2] and for fire safety reviews [10,11]. Moreover, at the end of February 1989, the IAEA held an International Symposium on Fire Protection and Fire Fighting in Nuclear Installations [12] intended to cover a large scope in the field and also to provide the opportunity for outlining several aspects of the technology at time, including research and development [13–15], standardization and licensing [16–18], firefighting practices [19,20] and operating experience [21]. However, although the international nuclear community (*e.g.*, regulators, operators and designers) has been studying and developing tools for defending against this hazard, it is reasonable to say that the research activity devoted to structural fire-safety analysis is still rather scarce, as attested by the relatively small number of available publications on the subject. Without claiming to be complete, such publications report essentially the work done by Jeon et al. [22], Suard et al. [23], Ferng and Liu [24], Ferng and Lin [25], Prétrel and Such [26], Berg [27] and, Levinson and Yeater [28].

Moreover, none of the above studies addresses failures of steel containment vessels associated with the occurrence of fire. Incidentally, in view of the importance of the containment integrity for nuclear power plants, this paper presents the results of a numerical investigation on the structural safety assessment of a nuclear cylindrical steel containment vessel exposed to an accidental external fire condition.

The paper begins by presenting a description of the proposed numerical coupling procedure, linking the modeling of a nearly compressible flow with input energy given by a combustion model (CFD model) and a structural thermo-mechanical analysis (FEM model), allowing an accurate evaluation of a fluid-thermo-mechanical response for the entire duration of the simulated accident. Then, the obtained numerical results concerning (i) the time–temperature evolution of the burned gases performed by the CFD model, resulting from the combustion of a hydraulic leakage pool, (ii) temperature variation and (iii) thermo-mechanical behavior and ultimate strength of the vessel are presented and discussed.

The obtained results indicate that the suggested methodology can provide reliable fire-safety analyses, ensuring that the main safety (load-bearing and containment) functions of the installation are not impaired during accidental events.

---

* Corresponding author. Tel.: +55 21 39388493; fax: +55 21 39388484.
E-mail addresses: jcsilva@coc.ufrj.br (J.C.G. Silva), alandes@coc.ufrj.br (A. Landesmann), fernando@coc.ufrj.br (F.L.B. Ribeiro).

http://dx.doi.org/10.1016/j.firesaf.2014.08.013
0379-7112/© 2014 Elsevier Ltd. All rights reserved.

Nomenclature

c_y	specific heat of steel (J/kg ºC)
E	elasticity modulus (GPa)
$\mathbf{f}_b$	external force vector (excluding gravity – N/m²)
$\mathbf{g}$	gravitational acceleration vector (m/s²)
h	heat transfer coefficient (W/°C m²)
h_s	sensible enthalpy (J/kg)
k	thermal conductivity (W/m °C)
$\dot{m}_b'''$	mass source term rate per unit volume (kg/m³ s)
p	pressure (N/m²)
$\overline{p}$	background pressure (N/m²)
$\dot{q}'''$	heat release rate per unit volume (W/m³)
$\dot{q}_b'''$	energy transferred to subgrid-scale droplets and particles (W/m³)
$\dot{\mathbf{q}}''_{rad}$	radiative flux vector (W/m²)
q''_{tot}	total heat flux (W/m²)
q''_{rad}	radiative heat flux (W/m²)
q''_{inc}	incident heat flux (W/m²)
q''_{conv}	convective heat flux (W/m²)
Q	internal generated heat per unit volume (W/m³)
R	ideal gas constant (8.314 J/mol.°C)
t	time (s)
T	temperature (°C)
T_{AST}	adiabatic surface temperature (°C)
T_s	surface temperature (°C)
T_g	gas temperature (°C)
$\mathbf{u}$	velocity vector (m/s)
x	Cartesian coordinate axis (m)
y	Cartesian coordinate axis (m)
Y	mass fraction
$\overline{W}$	molecular weight (kg/mol)
z	Cartesian coordinate axis (m)

Greek symbol

δ	displacement (cm)
ε	deformation (%)
ν	Poisson ratio
ξ	emissivity
ρ	density (kg/m³)
ς	Stefan–Boltzmann constant
σ	stress (MPa)
$\tau_{i,j}$	viscous stress tensor (N/m²)

Subscript

1	first principal direction
2	second principal direction
0	20 °C – ambient temperature
3	third principal direction
b	source term
el	elastic
pl	plastic
p	proportional
s	surface
T	temperature
vM	von Mises
y	steel

Superscript

$''$	per unit area
$'''$	per unit volume

2. Numerical modeling

The proposed procedure has two main steps, denoted as the CFD model and the FEM model. In the first step, the Fire Dynamics Simulator code (FDS [29]) is employed to simulate the fire propagation. Although the FDS is able to perform 1D heat transfer analysis throughout the solids obstructions in the flow domain, this resource is not sufficiently precise to simulate the temperature distribution in complex structures. Therefore, a complete 3D thermo-mechanical analysis model is needed, which is performed at the second analysis step by a FEM model. In the present work the Ansys package [30] is used to perform the FEM analyses. In order to accomplish this coupled field analysis, the structure thermal exposure must be transferred as boundary conditions from the CFD to the FEM models. The implicit differences between the solution method and the level of geometry discretization demanded by each type of analysis make the data exchange between both analyses a complex task. Next, each model will be presented separately.

2.1. CFD model

FDS [29] is one of the five fire models verified and validated by Nuclear Regulatory Commission (US NRC) in NUREG-1824 [31]. This fire-driven fluid flow model uses the conservation equations in an appropriate form for low-speed (low Mach number) applications. The hydrodynamic model is defined by the following equations:

Continuity equation

$$\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho\mathbf{u})=\dot{m}_b''', \tag{1}$$

Energy equation

$$\frac{\partial}{\partial t}(\rho h_s)+\nabla\cdot(\rho h_s\mathbf{u})=\frac{D\overline{p}}{Dt}+\dot{q}'''-\dot{q}_b'''-\nabla\cdot\dot{\mathbf{q}}''_{rad}+\nabla\cdot\left(\rho h_s D\nabla Y\right)+\nabla\cdot(k\nabla T), \tag{2}$$

Momentum equation

$$\frac{\partial(\rho\mathbf{u})}{\partial t}+(\mathbf{u}\cdot\nabla)\rho\mathbf{u}=-\nabla p+\rho\mathbf{g}+\mathbf{f}_b+\nabla\cdot\tau_{ij}, \tag{3}$$

Equation of State

$$\overline{p}=\frac{\rho TR}{\overline{W}}. \tag{4}$$

The low-speed flow induced by the fire is threated by pressure decomposition in background plus perturbation terms, as presented by Rehm and Baum [32], eliminating shock waves related to high speed flow. Then, temperature and density are considered as inversely proportional by Equation of State (as function of background pressure – Eq. 4), which leads to a reduction in the number of dependent variables. In applications presented in this paper, turbulence is treated by Large Eddy Simulation (LES, [33,34,35]) which is a technique to address the dissipative processes that occur at length scales smaller than those explicitly solved by the numerical grid. The spatial derivatives of the governing equations are approximated by a second-order accurate finite differences in a three-dimensional grid of uniform