Powder Technology 272 (2015) 132–141

Contents lists available at ScienceDirect

Powder Technology

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

CFD simulations to study parameters affecting dust explosion venting in silos

Alberto Tascón [a,*], Pedro J. Aguado [b]

[a] Departamento de Agricultura y Alimentación, Universidad de La Rioja, Av. Madre de Dios 51, 26006 Logroño, Spain
[b] Departamento de Ingeniería y Ciencias Agrarias, ESTI Agraria, Universidad de León, Av. Portugal 41, 24071 León, Spain

ARTICLE INFO

Article history:
Received 1 September 2014
Received in revised form 14 November 2014
Accepted 22 November 2014
Available online 2 December 2014

Keywords:
Dust explosion
Silo
Venting
CFD

ABSTRACT

Vented dust explosions in a 16.3 m^3 silo were simulated using a commercial CFD program. Simulations were carried out for vent panels without inertia and for a silo roof acting as a venting device, with inertia. For the latter, the influence of several parameters on the pressures generated was studied, including characteristics of the initial dust cloud, size and position of the dust cloud, and ignition location. In addition, different vent area sizes and activation pressures were studied. For large vent areas and low overpressures, the results showed that the negative pressures generated could be of the same magnitude as the overpressures. Several peaks in overpressure were identified along the pressure–time curves. The results showed the expected trends and agreed reasonably well with the standards on explosion venting. Although the standards seem to overestimate vent area sizes to some extent, pressures are very dependent on the initial conditions of the dust cloud, and more unfavourable scenarios than those considered in this study could easily arise, producing a stronger explosion and higher pressures. For the venting roof with inertia, the pressures and associated vent areas matched the NFPA 68 extremely well.

1. Introduction

Dust explosions are a serious hazard in many process industries and storage facilities. When the risk of explosion is not adequately addressed, the results can be catastrophic, involving loss of life and significant economic costs due to damage to equipment and buildings, loss of products, and suspension of activity [1].

Statistics show that a substantial percentage of dust explosions occur in silos, particularly those used in the agricultural and food sectors [2]. Bulk products inevitably form potentially explosive dust clouds inside a silo during filling and emptying processes. Although the elimination of potential ignition sources is a priority to prevent explosions, in many cases it can also be advisable to install protection systems to limit damage in the event of an explosion. When no protection system is in place, a dust explosion can generate pressures of up to 7–10 bar (700–1000 kPa), which may lead to fracture and collapse of the silo.

Venting is a widely used protection system due to its ease of installation and relatively low cost. The aim of venting is to limit the development of internal overpressure in the event of an explosion, maintaining it below a threshold value that the vessel is capable of resisting. Explosion venting devices are activated when a pre-determined pressure is reached, opening a vent area to the exterior that allows for expansion of the combustion gases produced in the interior of the vessel and thus relieving internal pressure. The larger the vent area, the lower the internal pressure reached.

Many different methods have been proposed for sizing vents [3]. At present, the two most widespread and internationally recognised methods are those described in the European standard EN 14491 [4] and the American standard NFPA 68 [5]. Both standards use the same parameters for sizing venting devices: the maximum pressure which must not be exceeded, the explosion characteristics of the stored product, the geometry of the vessel to be protected and the pressure which activates vent opening.

However, under certain circumstances the two aforementioned standards yield significantly different results, as has been shown in Tascón et al. [6]. Depending on the maximum reduced explosion overpressure[1] (P_{red}) and the length-to-diameter (L/D) ratio of the vessel, the vent area calculated according to the general formula in EN 14491 can be around twice that suggested by the NFPA 68.

The installation of silo venting systems is not always easy, and technical difficulties often arise. In many cases, the standards impose such large vent areas that the system is eventually rejected, especially in the case of the cylindrical metal silos commonly used to store agricultural and food products; the roof and upper walls of such silos typically have a very low resistance, obliging the installation of very large vent areas [7].

Despite the plethora of studies which have been conducted in recent decades, most of which are described in the comprehensive book by

* Corresponding author. Tel.: +34 941 299 733; fax: +34 941 299 721.
E-mail address: alberto.tascon@unirioja.es (A. Tascón).

[1] P_{red} is the maximum overpressure generated by an explosion in a vessel protected by either venting or suppression [4]. The silo should be able to resist this P_{red}.

http://dx.doi.org/10.1016/j.powtec.2014.11.031
0032-5910/

Eckhoff [2], our understanding of how the venting process works remains incomplete. Even today, controversy still exists over the validity of the standardised laboratory tests used to determine the explosion characteristics of stored materials, of the subsequent extrapolation of these results to the large volumes typical of industrial processes, and of the correspondence between the results obtained using 1 m^3 vessels and 20 litre spherical test vessels [8,9]; yet the formulas used in current standards for sizing vents are based on these explosion characteristics.

Meanwhile, numerical methods have become very useful tools in various fields of engineering and research. Simulations based on Computational Fluid Dynamics (CFD) are used to study complex fluid flow problems, including those involving heat transfer and chemical reactions.

The aim of the present research was to perform CFD simulations in order to study the influence of various factors on explosion venting in silos, including the characteristics of the initial dust cloud, venting device inertia, size and position of the dust cloud and point of ignition. The model used for these simulations was a real metal silo located at the Polytechnic University of Madrid, which has a volume of about 16.3 m^3. The ultimate aim of this research is to solve the technical difficulties that arise in silo design and protection against dust explosions.

2. Methodology

The simulations were performed using the commercial CFD tool FLACS-DustEx[2] (FLame ACceleration Simulator — Dust Explosions), a specific software for dust explosions marketed by the Norwegian company GexCon [10]. In the present study, version 1.0b3 was run under the Linux operating system Mandriva.

FLACS-DustEx is based on the CFD program for gas explosions known as FLACS, which was originally developed for simulating explosions in congested offshore geometries. The development of FLACS started at Chr. Michelsen Institute (CMI) in 1980 [11]. Since then, numerous efforts have been made to improve, extend, and validate the FLACS code [12,13]. The specific program for dust explosions was the result of a European project; a comprehensive review was reported by Skjold [10].

2.1. CFD model

The FLACS code solves the compressible form of the conservation equations for mass, momentum, enthalpy, and chemical species on a three-dimensional Cartesian grid using a finite volume method. FLACS models turbulence using a Reynolds-averaged Navier–Stokes (RANS) approach and the RANS equations are closed by means of the ideal gas equation of state and the standard k–ε model by Launder and Spalding [14]; however, some modifications to the standard k–ε model have been implemented in the code, including a model for turbulence generation behind subgrid objects [15].

FLACS uses a distributed porosity approach to model small subgrid obstacles [16]. Such approach enables the detailed representation of large industrial scenarios, such as process industries with complex arrangements of pipes and ducts, without leading to a very fine mesh and subsequent extremely long computing times. The distributed porosity approach also permits to model curved or angled surfaces using a Cartesian grid. Thus, a volume fraction or porosity β_v, which is defined as the fraction of volume that is available for fluid flow, is assigned to the individual mesh cells. Similarly, area porosities β_i in each of the three coordinate directions are defined. FLACS calculates the volume and area porosities on each of the mesh cells; porosities will have values from 0.0 (completely blocked, solid obstruction) to 1.0 (completely open, free space). This approach implies to incorporate the so-called porosity distributed resistance (PDR) formulation of the governing equations [16]. Also, the presence of subgrid objects contributes to flow resistance and turbulence generation.

Hjertager [16,17] described the basic equations of the model that is used in FLACS. The conservation equations for mass, momentum, enthalpy and mass fraction of chemical species, in tensor notation, are given below:

$$\frac{\partial}{\partial t}(\beta_v \rho) + \frac{\partial}{\partial x_i}(\beta_i \rho U_i) = 0 \tag{1}$$

$$\frac{\partial}{\partial t}(\beta_v \rho U_i) + \frac{\partial}{\partial x_j}\left(\beta_j \rho U_j U_i\right) = -\beta_v \frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\beta_j \sigma_{ij}\right) + \beta_v \rho g_i + R_i \tag{2}$$

$$\frac{\partial}{\partial t}(\beta_v \rho h) + \frac{\partial}{\partial x_j}\left(\beta_j \rho U_j h\right) = -\frac{\partial}{\partial x_j}\left(\beta_j J_{h,j}\right) + \beta_v \frac{Dp}{Dt} + \beta_v S_h \tag{3}$$

$$\frac{\partial}{\partial t}\left(\beta_v \rho m_j\right) + \frac{\partial}{\partial x_i}\left(\beta_i \rho U_i m_j\right) = -\frac{\partial}{\partial x_i}\left(\beta_i J_{j,i}\right) + R_j. \tag{4}$$

Here U_i is the velocity component in the x_i coordinate direction; p is the pressure; ρ is the density; σ_{ij} is the turbulent momentum flux at the cell surfaces; g_i is the gravitational acceleration in the x_i direction; R_i is the additional frictional resistance caused by subgrid obstacles; h is the enthalpy; $J_{h,j}$ is the enthalpy diffusive flux at the boundaries of the cell; S_h is the source term for enthalpy; m_j is the mass fraction of a chemical species j; $J_{j,i}$ is the mass diffusive flux of species m_j and R_j is the rate of production or consumption due to chemical reactions.

The fluxes in Eqs. (2)–(4) are modelled according to [16,17]:

$$J_{\Phi,i} = \frac{\mu_{eff} \partial \Phi}{\sigma_\Phi \partial x_i} \tag{5}$$

$$\sigma_{ij} = \mu_{eff}\left[\frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i}\right] - \frac{2}{3}\delta_{ij}\left[\rho k + \mu_{eff}\frac{\partial U_k}{\partial x_k}\right]. \tag{6}$$

Here Φ is a general scalar variable; μ_{eff} is the effective viscosity; σ_Φ is the effective Prandtl/Schmidt number and k is the kinetic energy of turbulence.

The conservation equations for the kinetic energy of turbulence (k) and its rate of dissipation (ε) read [16,17]:

$$\frac{\partial}{\partial t}(\beta_v \rho k) + \frac{\partial}{\partial x_j}\left(\beta_j \rho U_j k\right) = -\frac{\partial}{\partial x_j}\left(\beta_j \frac{\mu_{eff}}{\sigma_k}\frac{\partial k}{\partial x_j}\right) + G - \beta_v \rho \varepsilon \tag{7}$$

$$\frac{\partial}{\partial t}(\beta_v \rho \varepsilon) + \frac{\partial}{\partial x_j}\left(\beta_j \rho U_j \varepsilon\right) = -\frac{\partial}{\partial x_j}\left(\beta_j \frac{\mu_{eff}}{\sigma_\varepsilon}\frac{\partial k}{\partial x_j}\right) + C_1 \frac{\varepsilon}{k} G - C_2 \beta_v \rho \frac{\varepsilon^2}{k}. \tag{8}$$

C_1, C_2, σ_ε and σ_k are modelling constants [16] and G is the generation rate of turbulence.

FLACS uses a second-order central differencing scheme for diffusive fluxes, and a second-order "kappa" scheme, i.e. a hybrid scheme with weighting between second-order upwind and central difference schemes, for the convective fluxes. The time discretization scheme employed in the code version applied in this paper is a first-order implicit scheme (backward Euler), although second order time schemes have been implemented in more recent versions of the code. The SIMPLE[3] pressure correction algorithm [18] is applied, extended by Hjertager to compressible flows [11]. A comprehensive description of the FLACS code and a review of its characteristics can be found elsewhere [19,20].

FLACS-DustEx models particle-laden flows considering the dust cloud as an equilibrium mixture where dispersed particles are in dynamic and thermal equilibrium with the gaseous phase; this corresponds to an Eulerian approach in the limiting case when the Stokes

[2] FLACS-DustEx was previously marketed as DESC (Dust Explosion Simulation Code).

[3] SIMPLE stands for Semi Implicit Method for Pressure Linked Equations.