Modelling of dust explosions/Modellierung von Staubexplosionen

K. van Wingerden, B.J. Arntzen, Bergen/N
P. Kosiński, Warschau/P

SUMMARY

A CFD-tool for simulation of dust explosions is presented. So far a pragmatic approach has been chosen where the dust particle properties have been ignored. Thermodynamic and reactivity data can be taken directly from 20-l sphere standard tests (K_{St}- and P_{max}-values). The tool allows for predicting dust explosion consequences for prevailing process conditions and any geometry aspects of installations (including coupled vessels) thereby opening up for optimising dust explosion protection technology.

ZUSAMMENFASSUNG

Ein CFD-Werkzeug für Simulation von Staubexplosionen wird presentiert. Eine pragmatische Angriffsweise ist benutzt worden wobei die Eigenschaften von den Staubteilchen vernachlässigt worden sind. Thermodynamische und Reaktivitätsdaten sind direkt von 20-l Kugelstandardversuche entnommen worden (K_{St}- und P_{max}-werte). Das Werkzeug macht es möglich die Folgen von Staubexplosionen hervor zu sagen für herschende Prozessumstände und für alle Geometrische Aspekten von Anlagen (inkl. verbundenen Behälter). Das Werkzeug gibt damit weitere Möglichkeiten die Staubexplosionsschutztechnologie zu optimalisieren.

1 INTRODUCTION

In order to control and mitigate the potential consequences of dust explosions several techniques have been developed such as dust explosion venting and dust explosion suppression often combined with necessary isolation techniques such as fast-acting sliding valves and extinguishing barriers. These techniques have been developed entirely and only on the basis of experiments. A standardised way of generating dust clouds [1] has been the initial basis for both the sizing of vent openings and the testing and design of suppression systems. This standardised technique (the dust is introduced into a single cubical explosion vessel by emptying a container containing a certain amount of dust pressurised with air. Ignition is effected after a fixed time) aims at assuring a homogeneous dust cloud and a fixed degree of turbulence at the moment of ignition. The same technique is used to determine certain dust explosion properties such as the K_{St}-value used to describe the reactivity of the dust and an important parameter again when designing dust explosion venting and suppression systems in practice. The technique has no direct connection with any process in industry but the level of initial turbulence appears to be relatively high compared to that in most industrial application. An exception may be a mill [2] where the level of turbulence sometimes may be higher than generated by the standardised technique. The standardised technique therefore normally assures conservative estimates of vent size openings in practice and a sufficiently well designed suppression system.

The same accounts for active isolation systems: the design is based on experiments where normally standardised dust cloud generation systems are used assuring that these systems are located in such a way that there is sufficient time to be activated and to be fully closed (in the case of a fast-acting valve) or the be able to act as 100 % active barrier (extinguishing barrier) when the explosion flame arrives.

In practice conditions may be very different from the standardised test conditions. The installation in which dust is being handled may only partly be filled with a dust cloud or the dust cloud may be non-homogeneous. Also the level of turbulence may be very different (often much lower). Using standard techniques to calculate vent openings may lead to very conservative vent opening sizes, i.e. very big compared to what really is necessary. The same accounts for suppression systems a conservative design may very well be possible. Isolation techniques are sometimes not applied, as due to geometry limitations there would be no time for the system to respond and close in time.

This has also been recognised and several experimental campaigns have been ongoing such as those reported in [3] and [4] regarding venting of pneumatic charged silos and cubical vessels, [5] regarding venting of a bag filter and [6] regarding protection of small mills. In general these studies confirm that non-homogeneous dust clouds partly filling an installation combined with low levels of turbulence lead to lower combustion rates, implying the possibility of applying smaller vent openings or less suppressant and still being able to safeguard industrial installations.

On the other hand one should consider geometrical aspects such as elongated vessels (e.g. silos), presence of obstructions (e.g. in transport systems), coupled vessels, inertia of venting systems etc. which normally cause explosions to generate higher pressures. This is also recognised in the various standards giving guidance for these kinds of aspects or ruling out application of this kind of protection technique for a certain application.

It would be impossible to perform experiments for all situations, which can arise in practice and therefore it is natural to look for a more universal model than the simple calculation techniques, which are offered in standards nowadays [e.g. 7]. The most promising technique would be models based on CFD-technology.

In this paper a pragmatic approach is presented for such a technique using a tool already widely in use to predict the consequences of gas explosions in petroleum product processing installations [8]. This tool is known as FLACS, which is an acronym for "FLame ACcelleration Simulator".

2 DESCRIPTION OF THE CFD-TOOL FLACS

CFD is acronym for Computational Fluid Dynamics. In CFD-tools, reality is discretised with a certain time and space resolution. Space is divided into a number of volumes, in which fluid properties (velocities, density, pressure, ...)

are assumed constant in each discrete time interval. FLACS is a special - CFD-code developed specially for description of ventilation, gas dispersion and explosion processes in complex geometries such as offshore modules [8]. The effects of turbulence and chemical reactions are included in the gas dynamic partial differential equations solved for each control volume. In FLACS, a finite-volume technique and a weighted upwind/central differencing scheme for the convection terms are applied to discretise these equations. using. Velocities are calculated on staggered grids. The effect of turbulence is included through the eddy viscosity concept by solving equations for turbulent kinetic energy (k) and its rate of decay (ε). The tool has been validated extensively against gas explosion experiments the majority of those being experiments carried out in partially confined geometries such as geometries representing confinement conditions prevailing in offshore modules.

For the communication between the user and FLACS advanced user interfaces were developed (based on CAD and computer graphics technology) and given the names CASD (Computer-Aided Scenario Design) and Flowvis. CASD is used to generate the geometry (i.e. geometrical layout of the plant, compartment, vessel) in which the dispersion or explosion is to be investigated. If available, a geometry can be directly imported from a CAD-system. CASD is also used to define the scenario of the incident, i.e. gas cloud composition, cloud location, ignition point location and to specify the output parameters (where and what to measure).
Flowvis is used to presented the output which can be time-series plots like pressure-, impulse-, and drag-time plots as well as coloured shaded-image contour representations of velocities, flame location, pressure etc.

Gas explosions are in some aspects easier to represent than dust explosions.

The combustion of dust varies considerably depending on the dust type. For organic dust types pyrolisis followed by a reaction in the gas phase plays an important role. For metal dusts high reaction temperatures are reached and heat radiation probably plays an important role. For all dusts applies that the effects of particle size distribution and specific surface area and presence of inert material in the dust such as moisture and oxidised material should be taken into account.

Additional challenges are the effect of gravity on the behaviour of airborne particles, forces (such as electrostatic forces) acting relatively in between airborne particles causing agglomeration and the process of dust lifting, i.e. dust becoming airborne due to expansion generated flow ahead of the explosion.

An academic approach would aim at describing all processes involved in the combustion (combustion in each particle, heat transfer from particle to particle, effect of turbulence on combustion, effect of presence of particle on turbulence, etc.) implying the necessity of knowledge of several dust specific properties such as heat transfer properties, diffusivity of oxygen into dust particles, combustion energy, etc. This is from a practical point of view, however, a too exacting and comprehensive way forward.

Therefore a more pragmatic course has been chosen in the present study.

To describe combustion of dust the following approximations are made:

- Particles are assumed to be infinitely small (not affected by gravity, no agglomeration),
- There is no dust present in the form of dust layers and that can be lifted,
- Dust combustion is described by a general theory based on parameters such as the laminar burning velocity, turbulence intensity and turbulence length scale.

In doing so the dust explosion problem is approached similar to the way gas explosions are modelled in CFD-codes such as FLACS.

3 DUST COMBUSTION MODEL

The combustion rate depends largely on the turbulence field ahead of the flame and the reactivity of the fuel. The modelling of the turbulence field in the FLACS code is based on a k-ε turbulence model adapted to make it independent of grid and initial conditions. The combustion model in FLACS consists of a flame model and a burning velocity model. The flame model gives a representation of the flame including localisation and flame area. The flame model moves the reaction zone with a specified burning velocity into the unburned reactants. This specified burning velocity is described by a burning velocity model describing the propagation velocity depending on parameters such as the reactivity of the fuel (expressed by the laminar burning velocity) and turbulence parameters.

The expansion of combustion products depends through thermodynamics on the product composition, which is controlled by equilibrium chemistry.

The approach chosen for developing a dust burning velocity model is based on the approach proposed by [9] developed for gaseous combustion and shown by the same institution to be applicable for dust combustion [10]. Figure 1 shows the relationship between the turbulent burning velocity of two maize starch-air mixtures as a function of the turbulence intensity measured in a 0.95 m^3 bomb (from [11]). Both the turbulent burning velocity and turbulence intensity has been normalised by the laminar burning velocity. The Figure shows a linear increase of the burning velocity with the turbulence intensity. In addition to the turbulence intensity also the turbulence length scale has an influence on the combustion rate. With increasing length scale the turbulent burning velocity increases. This could not been seen in the experiments of Figure 1 because they were performed at 1 scale only.

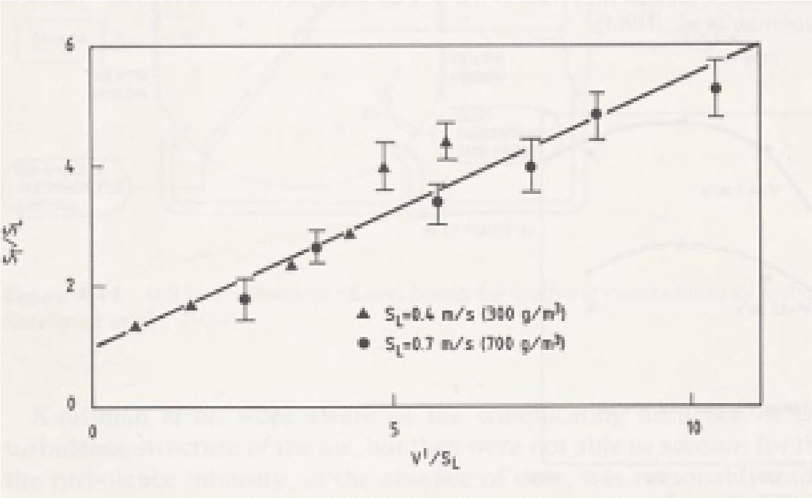

Figure 1 Variation of turbulent velocity S_T with turbulence intensity u' for maize starch-air mixtures. Both the turbulent intensity and turbulent burning velocity have been normalised by the laminar burning velocity S_L (taken from [11]).

Figure 2 shows how the turbulent burning velocity of gases varies with turbulent intensity and length scale (from [10]). The Figure also contains results for corn starch dust.

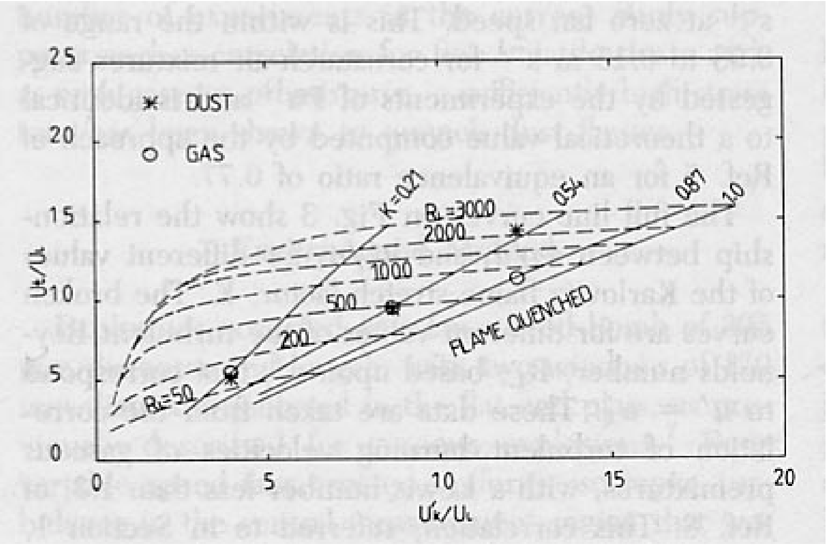

Figure 2 Variation of turbulent velocity S_T with turbulence intensity u' at different strain rates for gas-air and corn starch-air mixtures. Both the turbulent intensity and turbulent burning velocity have been normalised by the laminar burning velocity S_L (taken from [10]).

The data of Figure 2 are often referred to as flamelet library and can be expressed by a single relationship (valid for $u' > 0.3$ m/s):

$$S_T = 6.8\, S_L^{3/5} l_t^{3/20} u'^{3/20}$$

With S_T = turbulent burning velocity (m/s)

S_L = laminar burning velocity (m/s)
l_t = turbulent length scale (m)
u' = turbulence intensity (m/s)

4 MODEL APPLICATIONS

4.1 Tests in a 10 m^3 cubical closed vessel

The model has first of all been applied to tests performed in a 10 m^3 cubical closed vessel. These tests have been used to verify the model presented above. The tests were performed with maize starch (concentration 750 g/m^3). The maize starch was introduced pneumatically using the standard method described in [1]. Two delay times between opening of the valve of the pneumatic system and ignition were used (600 ms and 900 ms).

During the tests the dust concentration and turbulence intensity were measured at several locations in the vessel. On the basis of the turbulence intensity measurements the turbulence length scale could be determined (in contradiction to an earlier attempt where the effect of length scale was neglected [12]. The dust was pneumatically introduced into the vessel. Turbulence is produced near the nozzle during the outflow from the dust container. Neglecting this turbulence production will not lead to big deviations when modelling the turbulence field since the production occurs locally only. This simplification allows for estimation of a turbulence length scale, which can be used in the burning velocity model presented above. When there is only decay and no production of turbulence the k-ε turbulence model is reduced to $\partial k/\partial t=\varepsilon$ where $k = 1.5u'^2$ and $\varepsilon = u'^3/6l_t$. The relationship between u' and l_t is then
$\partial u'/\partial t = u'^3/18l_t$. A solution for this partial differential equation is:

$l_t = u'_2 u'_1 (t_2-t_1)/18(u'_1-u'_2)$ resulting in a length scale of l_t=0.1 m when applying the measured turbulence intensity.

Applying these results in the combustion model presented above results in pressure-time histories deviating from those measured. The deviation assumes to be related to the thickness of the reaction zone compared to what is common for gas explosions in empty enclosures. Laminar flames in gas-air mixtures have a typical flame thickness of 1 mm. A thicker flame will result in a reduction of the effective burning velocity. Introduction of a flame thickness model depending on the flame radius gave satisfactory results. Figures 3 and 4 present several measured pressure-time histories compared to a calculated one for ignition delay times of 600 ms and 900 ms respectively.

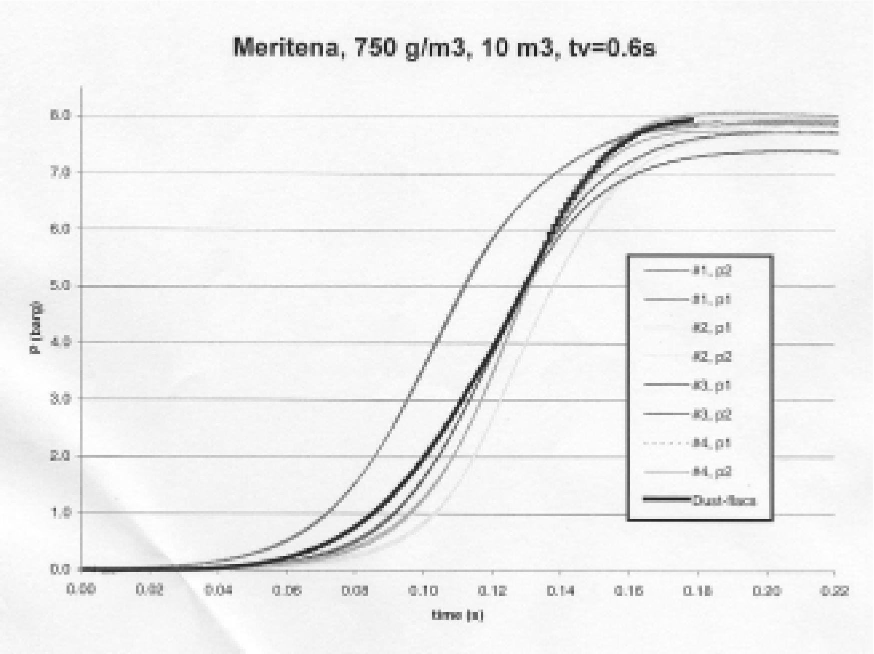

Figure 3 Measured and predicted pressure-time histories of explosions in 750 g/m^3 maize starch-air mixtures ignited 600 ms after start of the pneumatic injection of the dust in a 10 m^3 closed vessel.

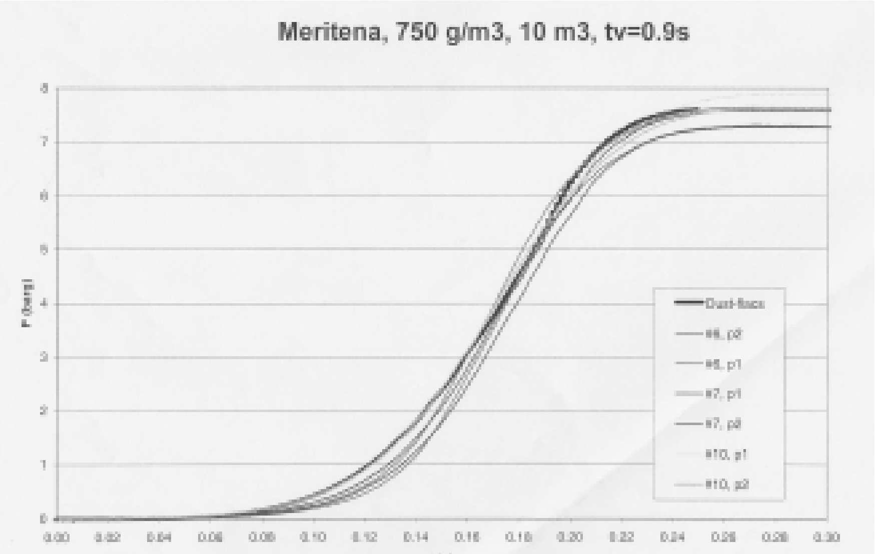

Figure 4 Measured and predicted pressure-time histories of explosions in 750 g/m^3 maize starch-air mixtures ignited 900 ms after start of the pneumatic injection of the dust in a 10 m^3 closed vessel.

4.2 Comparison of predictions for vented dust explosions with VDI 3673

Using the CFD-model adapted as described above simulations were done to see whether model predictions reproduce the vent sizing requirements as prescribed by VDI3673 [7]. The predictions were carried out for "homogeneous dust clouds". Results (considering a volume of 1 m^3 and K_{St}-values over a range of 131-510 bar.m/s) are presented in Figures 5 and 6 clearly indicating that the model reproduces the predictions of the VDI-guideline reasonably well. The turbulence parameters necessary for these calculations were obtained from measurements published in [13].

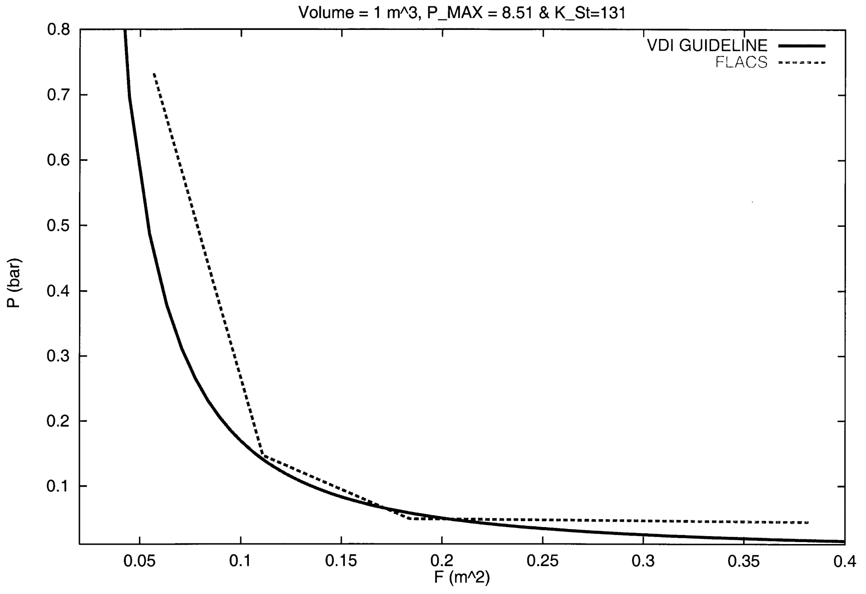

Figure 5 Comparison of maximum overpressures according to the VDI-guideline 3673 and the described CFD-model for various vent opening sizes in a 1 m^3 vessel (K_{St}-value = 131 bar.m/s).

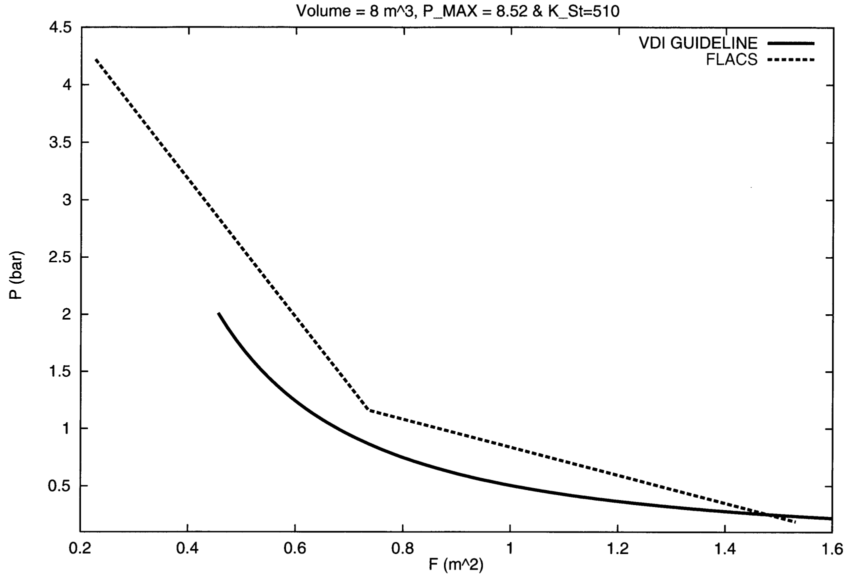

Figure 6 Comparison of maximum overpressures according to the VDI-guideline 3673 and the described CFD-model for various vent opening sizes in a 1 m^3 vessel (K_{St}-value = 510 bar.m/s).

4.3 Comparison with experiments performed in a pneumatically filled 12 m^3 silo

In [14] results are presented of experiments performed in a 12 m^3 silo which was filled pneumatically from the top. During these experiments measurements were performed of both the turbulence intensity and the dust concentration during the filling of the silo. The experiments were performed for different filling rates of maize starch. Explosion experiments involved ignition near the bottom in the middle and near the top of the silo. Two different vent areas were applied: 0.3 m^2 and 0.5 m^2.

These experiments represent an ideal test case for the model and therefore several of these tests were simulated. Results are presented in Table 1 comparing both the maximum reduced overpressure and the maximum rate of pressure rise in the silo. The simulations were performed without simulating the feeding process. The initial conditions were taken directly from the turbulence and concentration measurements. The necessary thermodynamical and reactivity data for maize starch were obtained directly from 20-l sphere tests (i.e. K_{St}- and P_{max}-values).

Table 1 Comparison of simulation results and experimental data (from [14]) for different ignition source locations and venting areas.

Experimental conditions		**p_{red} [bar]**		**dp/dt [bar/s]**	
Venting area	**Ignition source position**	**Numerical simulation**	**Experiment**	**Numerical simulation**	**Experiment**
A = 0.3 m^2	**bottom**	0.25	0.2 – 0.8	3.9	0.6 –10.2
	middle	0.43	~ 0.4	6.9	1.0 – 4.0
	top	0.13	0.1 - 0.4	2.5	2.0 – 3.0
A = 0.5 m^2	**middle**	0.24	~ 0.2	4.4	1.0 – 3.0

5 CONCLUSIONS

A CFD-tool is being developed aiming at being able to simulate dust explosion propagation in complex industrial installations. The thermodynamic and reactivity data necessary for being able to perform such calculations are derived from standard 20-l vessel tests (K_{St}- and P_{max}-values). The results of a comparison of predictions made with this tool and guideline predictions and results from tests performed in a pneumatically filled silo indicate that application of such a tool will be fully possible in the future. The availability of these kinds of tools will allow for

- predicting the course of explosions in complex plants consisting of several coupled items
- predicting the consequences of explosions in installations with flow and concentration conditions considerably deviating from the standard ISO-conditions
- improved plant design including optimised application of control and mitigating measures

Further developments are necessary including description of quenching of dust flames by turbulence, dust lifting for prediction of consequences of secondary explosions or explosions in plants where dust generally is present as dust layers instead of dust clouds (e.g. spray dryers), improvement of the dust combustion model, description of dust as particles and a more thorough validation.

6 ACKNOWLEDGEMENT

The authors wish to thank Mr. Georg Suter of Schweizerisches Institut zur Förderung der Sicherheit and Dr. Frank Hauert of BG Nahrungsmittel und Gaststätten for making available experimental data.

7 REFERENCES

[1] International Standardisation Organisation (1985) *Explosion Protection systems. Part 1: Determination of Explosion Indices of Combustible Dusts in air*, ISO 6184/1, Geneva

[2] Eckhoff, R.K. (1991) *Dust Explosions in the Process industries*, Butterworth-Heinemann, Oxford, UK

[3] Bartknecht, W. (1988), *Massnahmen gegen gefährliche Auswirkungen von Staubexplosionen in Silos und Behälter: Explosionsdruckentlastung beim Befüllen durch Flugförderung*, Abschlussbericht Project no. 01 HK 664 AO, Bundesministerium für Forschung und Technologie

[4] Siwek, R. (1989), *Druckentlastung von Staubexplosionen beim pneumatischen Befüllen von Behältern*, VDI-Berichte, vol. 701, pp. 529-567

[5] Eckhoff, R.K., Alfert, F. aund Fuhre, K. (1989), *Venting of dust explosions in a 5.8 m^3 bag filter under realistic conditions of dust cloud generation*, VDI-Berichte, vol. 701, pp. 695-722

[6] Barth, U., Siwek, R., Kubainsky, C., and Suter, G., (1996) *Staubexplosionsschutz für kleine Mahlanlagen*, VDI-Berichte Vol. 1272, pp. 493-510

[7] Verein Deutscher Ingenieure, VDI-Richtlinie 3673 "*Druckentlastung von Staubexplosionen*", 1995

[8] Arntzen, B.J. (1998) *Modelling of turbulence and combustion for simulation of gas explosions in complex geometries*, PhD Thesis Norwegian University of Science and Technology, Trondheim

[9] Bradley, D., Lau, A.K.C. and Lawes, M. (1992), *Flame stretch rate as a determinant of turbulent burning velocity*, Phil. Trans. R. Soc. London A (1992) 338, pp. 359-387

[10] Bradley, D., Chen, Z. and Swithenbank, J.R. (1988), *Burning rates in turbulent fine dust/air explosions*, Twenty-second Symposium (International) on Combustion, pp. 1767-1775, The Combustion Institute, Pittsburgh

[11] Tezok, F.I., Kauffman, C.W., Sichel, M. and Nichols, J.A. (1986), *Turbulent burning velocity measurements for dust-air mixtures in a constant volume spherical bomb*, Progress in Astronautics and Aeronautics, vol. 105, part 2, AIAA, New York

[12] Van Wingerden, K., Hansen, O.R. and Stavseng, L. (1996) *The use of a CFD-model for predicting blast and fire ball effects due to vented dust explosions*, VDI-Berichte 1272, pp. 565-577

[13] Hauert, F., Vogl, A. and Radandt, S. (1994) *Measurement of turbulence and dust cocentration in silos and vessels*, Proceedings of the Sixth Int. Coll. on Dust Explosions, Shenyang, P.R.C., pp. 71-80

[14] Hauert, F., Fogt, H., Vogl, A., Wennerberg, D. and Radandt, S. (1996) *Staubverteilung und Geschwindigkeitsparameter in einem pneumatisch befüllten Silo mit numerischen Simulationen*, VDI- Berichte Vol. 1271, pp. 309-326.