Modelling of gas flows in the goaf of retreating faces

Christian Tauziède, Yvon Mouilleau, Rémy Bouet

MODELLING OF GAS FLOWS IN THE GOAF OF RETREATING FACES

Christian TAUZIEDE, Yvon MOUILLEAU & Rémy BOUET

INERIS,
Verneuil en Halatte
France

SUMMARY

In retreating faces, dilution of firedamp flowing out of the goaf area at the junction of face and return roadway is problematic. Despite important flowrates and high CH_4 drainage efficiencies, it has been observed that 2/3 of firedamp evacuated by the air stream concentrate in this particular place.

So as to progress in preventing such difficulties, it is necessary to better understand the repartition of gas fluxes in the goaf. This has obviously a benefit on firedamp drainage efficiency, but also on the choice of nitrogen injection points and flowrates in case of spontaneous combustion in the goaf.

The use of a CFD code, PHOENICS, has allowed simulation of circulation of three gases : air, CH_4, and N_2.

Influence of the face dip on the repartition of CH_4 patterns in the goaf area has been qualitatively demonstrated.

INTRODUCTION

Caved-in longwall mining poses the particular problem of dilution of firedamp at the junction of the longwall and the return airway. A large number of measuring runs carried out in the Lorraine Coalfield have shown that 50 to 70 % of the firedamp which enters the ventilation network does so in this zone, despite high air flow rates (30 to 40 m^3/s) and a high drainage efficiency (as much as 70 % in certain cases). Various methods can be used to control this phenomenon and either prevent the formation of zones where the firedamp concentration in the air is dangerous, or push them back towards the goaf. These methods include the exhausting ventilation system developed in France (Lorraine Basin Collieries, 1990) and the Back Return System used in Great Britain (British Coal, 1993). However, these methods are only effective under certain favourable conditions and can therefore only provide a limited, partial solution to the problem.

For this reason, it seemed essential to improve our knowledge of gas circulation in the goaf. Various experimental studies have been successfully carried out in the past, aimed in particular at providing details about firedamp and air exchanges between the face and the goaf. However, relatively little is known about gas flows in the goaf itself, both as concerns circuits and flow rates.

In the Lorraine Collieries, there is also the risk of spontaneous combustion in the goaf. The method used to control this is injecting nitrogen into the goaf. However, this practice remains relatively empirical in terms of the choice of injection points and the flow rates used.

A better understanding of these flows would therefore allow the following :

- identification, for example, of the geometrical mining configurations

25ème Conférence Internationale des Instituts de Recherches sur la Sécurité dans les Mines, Prétoria, 13-17 septembre 1993, vol. 5, p. 41-50.

most likely to prevent firedamp heterogeneities at the face/return gate junction,

- improved specifications of firedamp drainage means,

- finally, the use of more efficient and perhaps more economical, nitrogen injections to control spontaneous combustion.

Carrying out experiments in the goaf is both very difficult and costly because of the inaccessibility of this area. Thus, the modelling of gas flows (air, methane and nitrogen) is an investigation method which is fairly well adapted to the situation. This approach is made possible by the development of computers and the existence on the market of general codes to solve fluid mechanics equations (Computational Fluid Dynamics or CFD). Work is being carried out in Great Britain on the basis of these considerations (British Coal, 1993). In France, we have used a set of programmes called PHOENICS [(1)] to produce a gas flow model for the entire geometry of a working : the face and the goaf area. The second chapter describes this model, while the third describes the qualitative results of the first simulations carried out.

DESCRIPTION OF THE FLOW MODEL

The approach used to produce the flow model consists in writing all the equations needed to describe the phenomena involved, digitize the system and then solve it using conventional numerical methods such as finite differences or finite elements techniques. The most popular computational algorithms have been extensively described (particularly Patenkar, 1979).
In the case we are interested in, the equations to be solved are those concerning mass conservation, linear momentum, and the transport of gases for which we need to know the breakdown in the geometry being considered i.e. methane and nitrogen if we wish to simulate the injection of this last gas.

Initially, for reasons of simplification and reduction of the calculation time, heat transfers will not be taken into account. The energy conservation equation is therefore not included in the system.

Once again with the aim of reducing the calculation time, the "gates/face/goaf" set has been reduced to a two-dimensional geometry.

Finally, because of the air speeds, which are less than ten meters per second, the fluids are considered to be Newtonian and incompressible.

The flows which need to be modelled occur within two media of an entirely different nature - the goaf can be considered to be a porous medium while, in the face and roadways, the flows are "free". Consequently, the forces which are exerted on the fluids are different from one medium to the other. Likewise, the diffusion of one or several gases occurs differently from one case to the other.

Thus, to model all the phenomena, two "sets" of formulae must be used, one to determine the free flow (in the roadways and face) and the other to determine the flow in a porous medium. These two formulae are described in the following paragraphs.

Modelling free flow

Conventionally, using two-dimensional Cartesian co-ordinates, the continuity equation for an incompressible fluid is written as follows :

(1) Registered Trademark, CHAM Ltd

$$\frac{\delta \rho}{\delta t} + \rho \frac{\delta \bar{u}}{\delta x} + \rho \frac{\delta \bar{v}}{\delta y} = 0 \qquad (1)$$

where :

- is the density
- $\bar{u}, \bar{v}$ are the average speeds of the fluid in the x and y directions.

Based on the steady state hypothesis, equation (1) is reduced to :

$$\rho \frac{\delta \bar{u}}{\delta x} + \rho \frac{\delta \bar{v}}{\delta y} = 0 \qquad (2)$$

Likewise, conservation of the linear momentum along the x-axis is written :

$$\rho \bar{u} \frac{\delta \bar{u}}{\delta x} + \rho \bar{v} \frac{\delta \bar{v}}{\delta y} = -\frac{\delta P}{\delta x} + \mu \left[\frac{\delta^2 u}{\delta x^2} + \frac{\delta^2 v}{\delta y^2} \right]$$

$$+ \rho g_x - \rho \frac{\delta}{\delta y} \left[\overline{u'v'} + u'^2 \right] \qquad (3)$$

where :

- P is the pressure (Pa)
- μ is the dynamic viscosity (kg/m.s)
- g_x is the acceleration x-component due to gravity (m/s^2)
- u' and v' are the speed fluctuations along the x- and y-axes.

The term (3a) represents the forces of inertia, (3b) expresses the effect of pressure, (3c) represents the effect of viscous stresses, (3d) that of gravity, and (3e) expresses the effect of fluctuations due to turbulence.

Conservation of linear momentum along the y-axis is written in the same way.

Introduction of the term (3e), and thus the new variables u' and v', make it impossible to resolve the system formed by equations (2) and (3). This is the well known problem of "closing the system".

Most CFD codes use the Boussinesq hypothesis which consists in considering turbulent agitation in a similar way to viscosity. The influence of turbulence is expressed using an equivalent "turbulent viscosity". To determine this new value, a so-called turbulence model must be developed. Such models and the theories they are based on are described fairly completely (Hinze, 1975; Launder and Spalding, 1972).

In the present case, the flows are extremely turbulent. They result in very high pressure drops throughout the ventilation circuit. The effect of turbulence must therefore be expressed in order for the flow dynamics to be represented correctly.

When using a turbulence model, experience shows that, among other things, the quality of the calculations is related to the quality of the geometric description of the medium being considered. This means that the grid must be close-meshed enough to describe the shape and size of the roadways. In practice, this invariably leads to a number of meshes which makes the calculation time prohibitive. This type of approach is therefore not practicable.

A large number of measuring runs carried out in the past (Simode, 1976) have enabled the values of typical linear head losses in roadways to be realised. Rather than setting up a turbulence model to ultimately obtain head losses which correspond to actual experience, it is far simpler in the long run and requires much less calculation time to introduce a flow resisting force into equations (2) and (3) which leads directly to the required head losses. Head losses can be expressed by equivalent heights, for example.

The equation for transport of a gas in the steady state is written as follows :

$$\rho \bar{u}\frac{\delta Y}{\delta x}+\rho \bar{v}\frac{\delta Y}{\delta y}=\rho D_t \left[\frac{\delta^2 Y}{\delta x^2}+\frac{\delta^2 Y}{\delta y^2}\right] \quad (4)$$

where :

- Y is the mass fraction of the gas considered,
- D_t is the turbulent diffusion coefficient.

Conventionally, the turbulent diffusion coefficient is obtained from a turbulence model. In the light of the previous remarks, it is obvious that this coefficient cannot be calculated. It must be remembered however, that the main aim with regards to free flow is to obtain the flow dynamics i.e. to calculate the pressure and average speed. The aim is not so much to follow a gas through the atmosphere of the working, but to ascertain its route in the goaf. A value between 10 and 100 times the order of magnitude of Fick diffusion coefficient D used when the flow is laminar was therefore taken for D_t.

Modelling the flow in a porous medium

Once again, we are faced with the question of solving a system based on the laws of conservation. In the present case, however, they must be modified in order to express the twofold effect of the porous medium: first, only a fraction of the total volume is available for the flow and second, an additional resisting force is exerted on the fluid.
Thus, the system of equations (2) and (3) is modified to give the following :

$$\rho\frac{\delta \bar{u}}{\delta x} + \rho\frac{\delta \bar{v}}{\delta y} = 0 \quad (5)$$

$$\rho \bar{u}\frac{\delta \bar{u}}{\delta x}+\rho \bar{v}\frac{\delta \bar{v}}{\delta y}=-\frac{\delta P}{\delta x}+\mu\left[\frac{\delta^2 u}{\delta x^2}+\frac{\delta^2 u}{\delta y^2}\right] + \rho g_x + R_i \quad (6)$$

For R_i, the following expression has been chosen :

$$Ri = -\frac{\mu}{k_p}\bar{u} - \beta \bar{u}^2 \quad (7)$$

where :

- k_p is the permeability of the porous medium
- ß is a function of the permeability and Reynolds number.

This approach, already proposed by a large number of authors (Vafai and Kim, 1990; Nield, 1991; Kim and Russel, 1985; Bradley et al., 1989), consists in expressing the forces exerted by the porous medium using a term resulting from the Darcy law (7a) and a term expressing the effect of the forces of inertia (7b).

One difficulty lies in determining the permeability k_p. Many formulations are to be found in the literature which connect the permeability to the geometrical characteristics of the medium, such as the porosity ϕ, the size of the grains in the medium, and the diameter of the pores. Dullien has given a relatively complete list of these formulations (Dullien, 1975).

The most widely used formulae in practice are no doubt those of Ergun and Kozeny-Carman (Carman, 1961). For example, the Kozeny-Carman law gives the following formula :

$$k_p = \frac{\phi^3}{(1-\phi)^2\, k.S_o^{\,2}} \quad \text{for } \phi < 0{,}8 \quad (8)$$

where So is the specific surface of the grains in the medium,
k is the Kozeny-Carman constant, between 5 and 6.

For the transport equation of a gas, the influence of the porous medium on the diffusion coefficient needs to be determined. To do this, correlations were used in order to express an "equivalent" diffusion coefficient as a function of the

Reynolds number, which is characteristic of the porous medium (Sinclair and Potter, 1965; Cussler, 1984).

Finally, the model built from the system of equations (2) to (8) describes the flows in the roadways and in the goaf.

In the case of goaf, the spatial distribution for the following must be defined :

- the permeability,
- the methane inlet flows towards the working.

Spatial distribution of the permeability

Several authors (Brunner, 1985; Stokes, 1985; Ediz and Edwards, 1991) relate the permeability to the mechanical stress exerted on the medium.

It can be imagined that when the stresses exerted on the solid rock are high, the blocks of rock are relatively compact, which results in low permeability. On the other hand, when the ground pressure is lower, there is less consolidation and greater permeability. The areas with the greatest permeability therefore correspond to the areas with the lowest stress level.

Given what we know about the mechanics of the ground around a caved-in longwall, it can be said that :

- the permeability will be higher in the area immediately behind the face,
- further back, the caved-in area is compressed again and the permeability gradually decreases and no doubt becomes stable after a certain distance.

It should also be noted that the old intake and return roadways and the face start plays a special role. With the arches left standing and the adjacent coal pillar playing a supporting role, recompaction of the caved-in ground does not occur in the same way as it does in the general area of the goaf. A strip of higher permeability therefore exists on the three sides of the rectangle formed by the goaf. Figure 1 illustrates the spatial distribution with regard to the permeability.

Distribution of methane flows towards the working

Figure 2 shows the distribution of methane surfacic flows coming into the medium which forms the goaf.

This distribution is based on fairly extensive knowledge of firedamp emission mechanisms. In particular :

- on a line perpendicular to the face, the firedamp flow increases from 0 to a maximum value located at a certain distance from the longwall and then decreases to a value in the vicinity of 0 next to the departure of the longwall (Tauziède and Pokryszka, 1993).
- on a line parallel to the face, the firedamp flow is higher near the edges of the panel than in its centre. This is due to the existence of cracks, over and under these particular zones, resulting from the movement of strata which occurs there after caving-in and relaxation. These cracks are ideal paths for firedamp flowing out from distant beds towards the working. In the middle zone, the beds in the roof, except for the close beds, subside more regularly. The firedamp has a greater tendency to flow in the direction of the beds and particularly the coal seams - which are relaxed - towards these cracks (Jeger, 1988).

In the following simulations, firedamp from mined coal has not been taken into account since, in the Lorraine Coalfield, it represents a very small proportion of the whole (5 to 10% in most cases).

In any event, this simplification has little or no influence on gas flow in the goaf area.

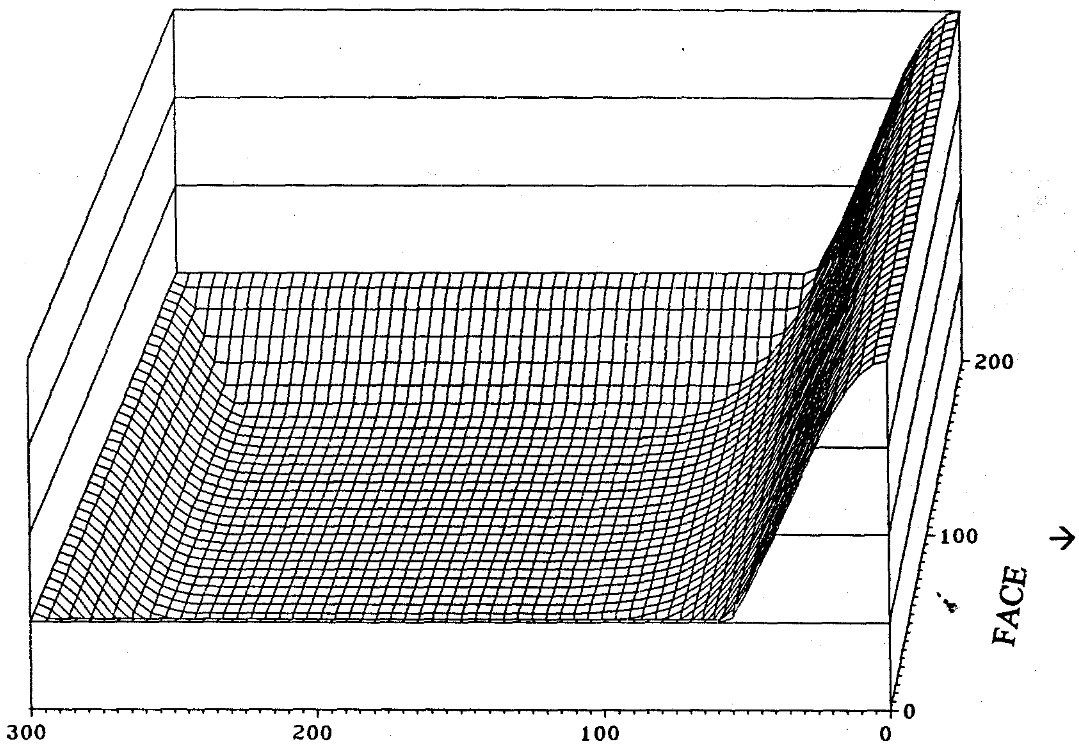


Figure 1 : Distribution of permeabilities in the goaf

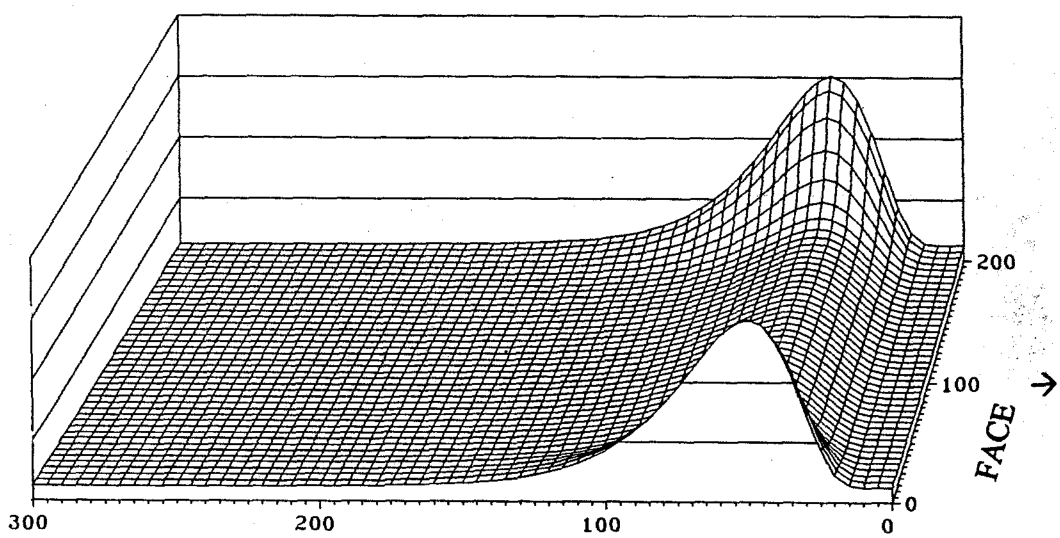


Figure 2 : Distribution of CH_4 flows in the goaf

RESULTS OF SIMULATIONS

The first simulations carried out concerned the simplified case of a longwall which is relatively representative of real conditions. It has the following characteristics :

- length of face : 200 m
- distance from start line : 100 m,
- face dip : variable,
- panel dip in the retreat direction of the face : 0°
- total ventilation flow rate : 33 m^3/s,
- total CH_4 flow rate : 0.5 m^3/s (corresponding to a CH_4 concentration of 1.5 % in the return gate).

The grid representing the face and goaf and the air intake and return roadways sections, consists of about 14 000 meshes.

The aim of the simulations was to check the capacity of the model to reproduce effects such as that of the face dip on the methane concentration distribution in the goaf, due to differences in density between methane and air. For this reason, different calculations were made, using the following dips : 0°, +10°, +20° (upcast ventilation) and -10° (downcast ventilation).

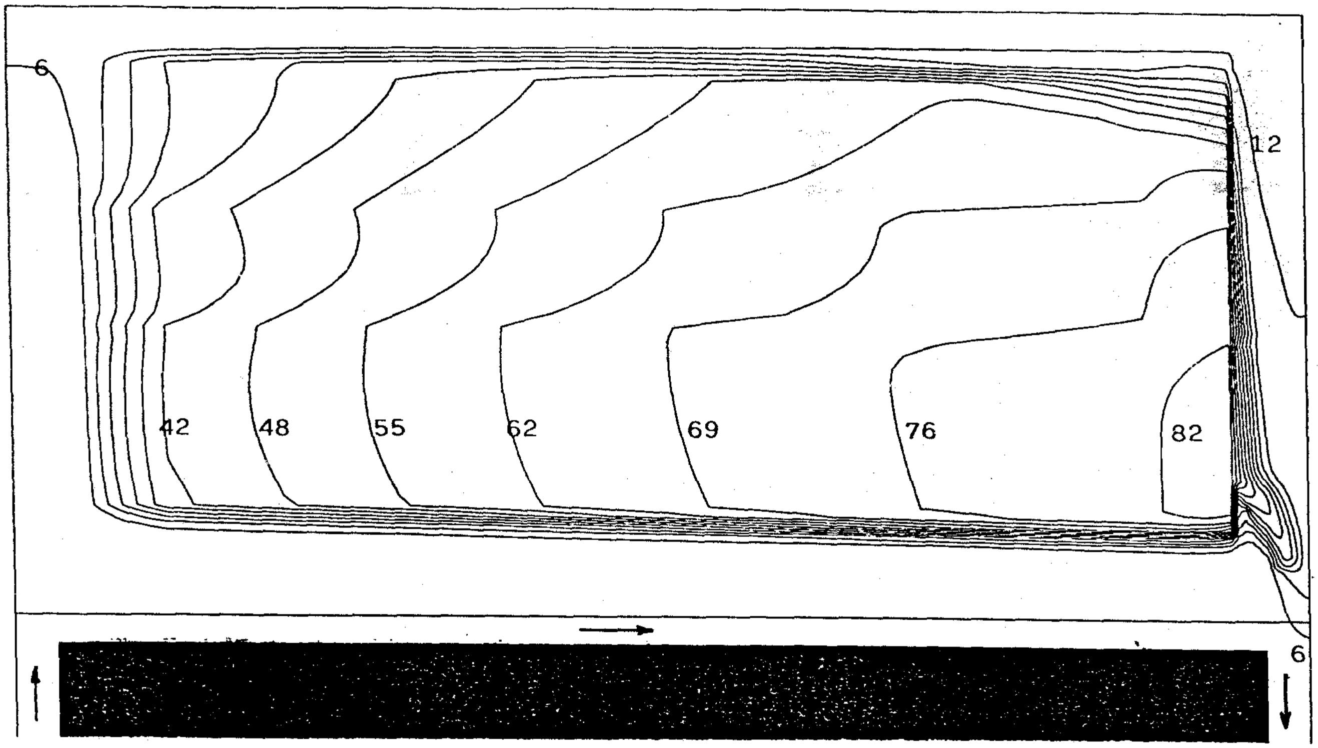
Looking at the plots of the CH_4 concentrations in Figure 3, pressures and speeds obtained, the following points can be noted :

- in all four cases: the calculation indicates a high methane concentration area (with respect to the statutory limit concentrations or the lower explosiveness limit) in the "blind angle" at the junction between the face and return gate. From a qualitative point of view, this is entirely in accordance with observations made in the workings (Cwiklinski and Josien, 1984). Flows in the goaf are more intense in the old start roadway and gates than in the rest of the goaf area. This, of course, is a result of the greater value given to permeability in these zones. The observations made in the workings also confirm this point.

- in the three cases of a 0°dip or upcast ventilation: flows observed in the goaf are always parallel to that of the face. With regard to methane concentrations, the zones with the highest concentration are situated in proximity to the return gate. This corresponds to the density effect. It is observed that the greater the slope, the less extensive the zones with a high CH_4 concentration, that is, the lower the average concentration in the goaf.

- in the case of downcast ventilation, flows are observed in the goaf in the opposite direction to that of the face. Firedamp in the distant zones in goaf flows upwards. Very high CH_4 concentrations therefore exist in virtually all of the goaf. The fringe of higher concentrations (for example > 5% CH_4) is found in the goaf at a much smaller distance from the face in this last case than in the case of upcast ventilation, and this is confirmed by the measurements.

UPCAST VENTILATION + 10°



UPCAST VENTILATION +20°

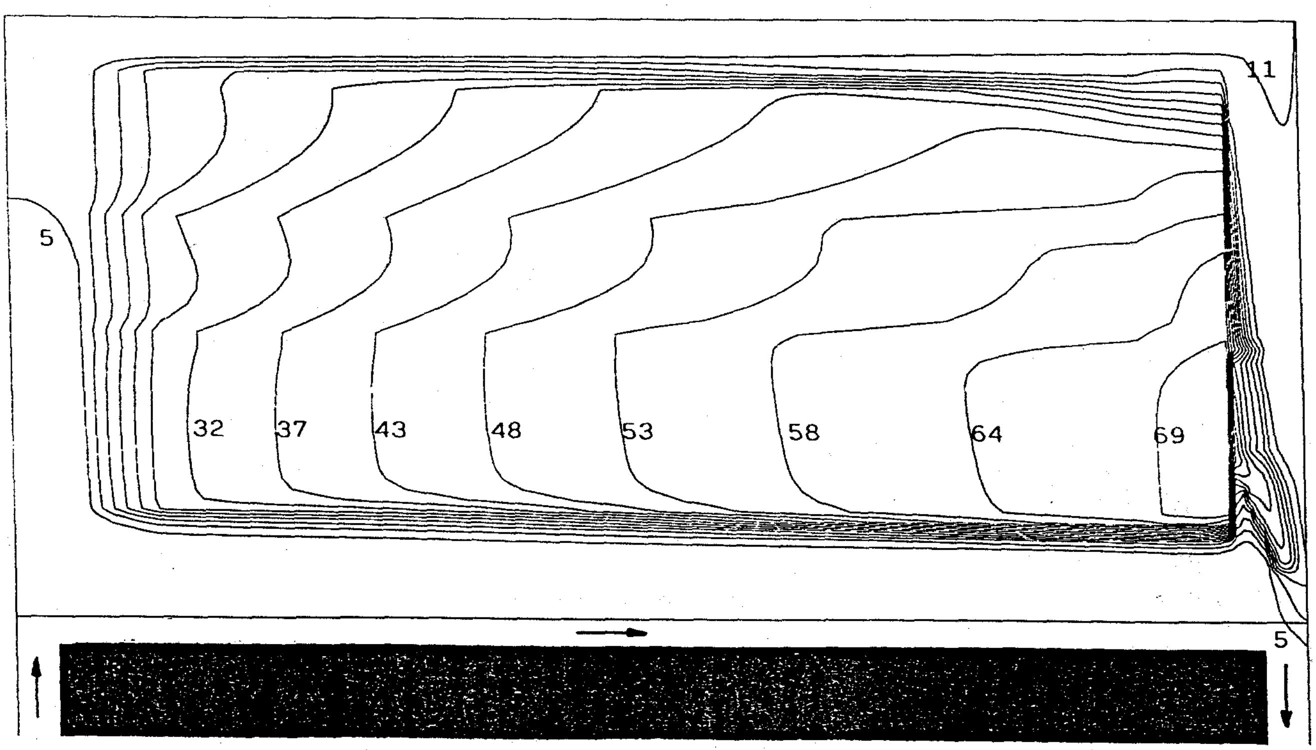
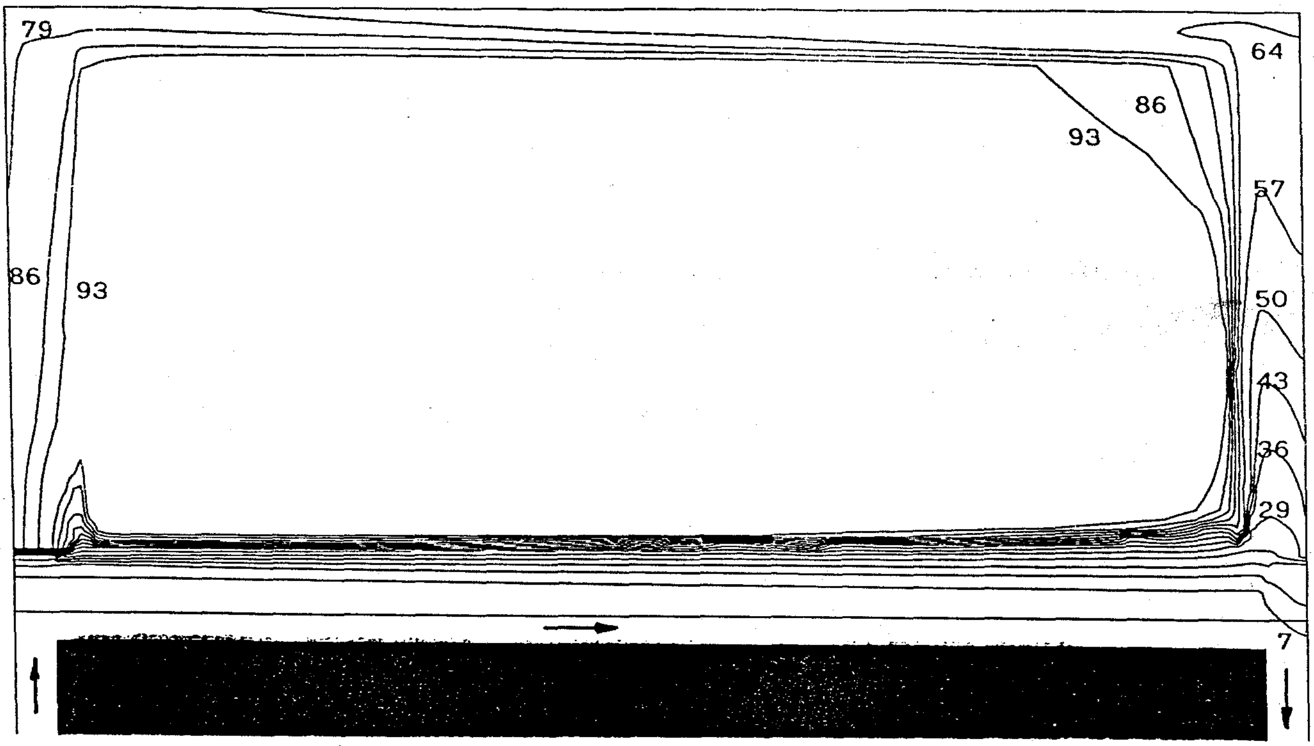
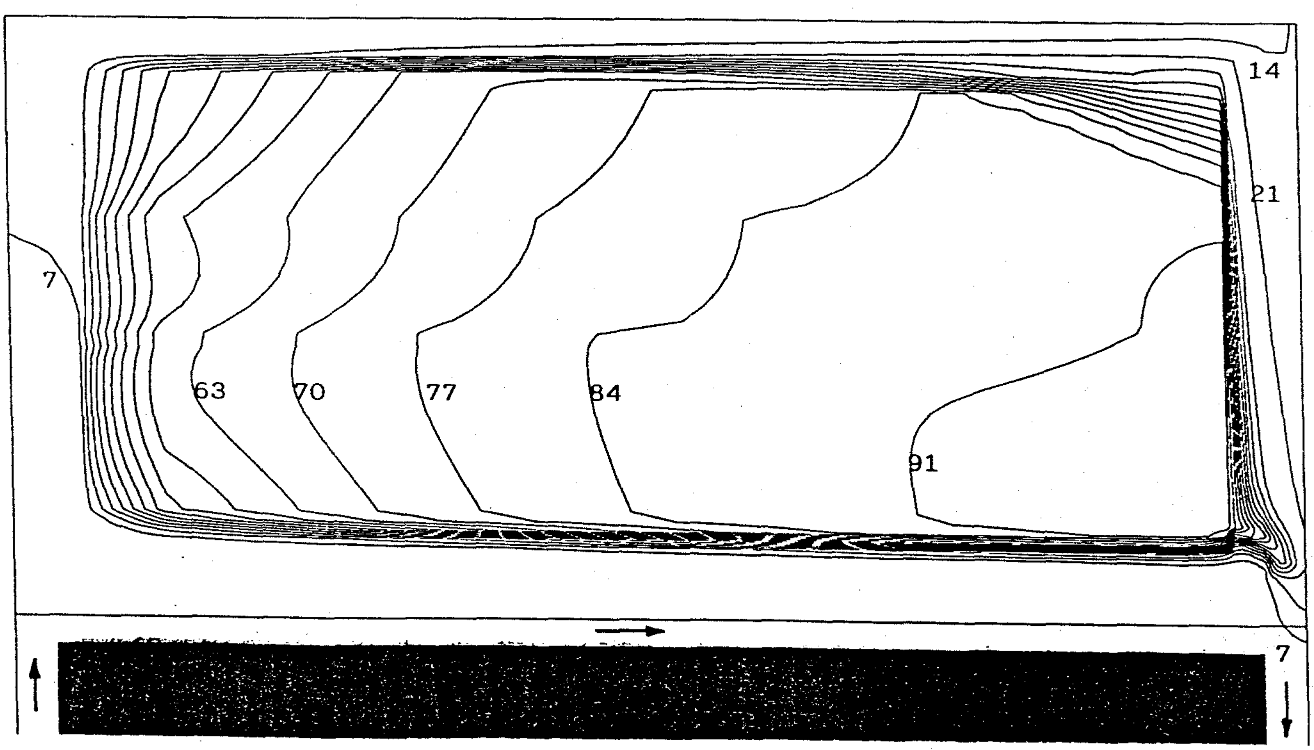


Figure 3 : Distribution of CH_4 concentrations in the goaf (%)

DOWNCAST VENTILATION -10°



HORIZONTAL VENTILATION 0°



<u>Figure 3</u> (cont'd) : Distribution of concentrations in the goaf (%)

CONCLUSIONS

The results of the first simulations carried out are in perfect agreement with observations and measurements made in the past. Although these results are only purely qualitative, they nevertheless provide an initial verification of the capacity of the model proposed to reproduce effects such as those which result from the density of CH_4.

For a better validation of the model, a complete adjustment has to be carried out on one or more real cases. The calculation parameters have to be adjusted (such as the relative permeability values between the different zones, for example, or the parameters of the turbulent diffusion model) until the best correspondence is obtained between the measuring results and the calculation results.

Anyway, the previous results indicate that CFD codes, such PHOENICS, are very promising tools for improving our knowledge of flows in the goaf. As a result, they will enable mine operators to perfect the different processes and techniques used to control firedamp and spontaneous combustion.

ACKNOWLEDGEMENTS

The work whose results are presented here received financial backing from the Ministry of Industry and from Charbonnages de France.

The authors would like to thank the engineers and technicians from the Lorraine Coalfield for their efficient assistance in carrying out this work.

BIBLIOGRAPHY

BRADLEY C.P., PROCARIONE J.A. McCARTER M.K, 1989, "Prediction of air flow through broken rock by finite difference grids", University of Utah.

BRITISH COAL, 1993, "Methane control for retreat faces", CEC Research Project 7220-AC/838.

BRUNNER M.J., 1985, "Ventilation models for longwall gob leakage simulation", 2nd US Mine Ventilation Symposium, Reno.

CARMAN P.C., 1961, "L'écoulement des gaz à travers les milieux poreux, traduit de l'anglais par Jacques MACHEFER", Institut National des Sciences et Techniques Nucléaires, Presse Universitaire de France.

CUSSLER E.L., 1984, "Mass transfer in fluid system", Cambridge University Press.

CWIKLINSKI D. et JOSIEN J.P., 1984 "Influence of descensional ventilation in a face on environmental conditions and distributions of gas and airborne dust", Third international mine ventilation congress, Harrogate, England.

DULLIEN F.A.L., 1975, "Single phase flow through porous media and pore structure", The chemical engeneering journal 1-34.

EDIZ I.G. and EDWARDS J.S., 1991, "Numerical simulation of time dependent methane flow", Mining Science and Technology.

HINZE J.O., 1975, "Turbulence", Second edition, Mac Graw-Hill Book Company New York.

HOUILLERES DU BASSIN DE LORRAINE, 1990, "Gestion moderne de la conduite de l'exploitation", Projet de recherche CCE 7220-AF/309.

JEGER C., 1988, "Will firedamp emission stop progress in longwall production ?", 21st century higher production coal mining systems symposium, Woollongong, Australia.

KIM S.J. and RUSSEL W.B., 1985, "Modelling of porous media by renormalisation of the Stokes equations", Journal of Fluid Mechanics, vol 154, pp 269-286.

LAUNDER B.E. and SPALDING D.B., 1972, "Mathematical models of turbulence", Academic Press, London and New York.

NIELD D.A., 1991, "The limitations of the Brinkman - Forchheimer equation in modelling flow in a saturated porous medium and at interface", International Journal of Heat and Fluid Flow, vol 12, n°3, pp 269-272.

PATENKAR S.V., 1979, "Numerical Heat Transfer and Fluid Flow", Mac Graw Hill Book Company, New York.

SIMODE E., 1976, "Valeurs pratiques des résistances", in Aérage, document S.I.M. n°3. Société de l'Industrie Minérale, Saint Etienne.

SINCLAIR R.J. and POTTER O.E., 1965, "The dispersion of gas in flow through a bed of packed solids", Transactions of the Institution of Chemical Engeneers, vol 43, n°1, pp T3 à T9.

STOKES A.W., 1985, "Air leakage through longwall wastes in the Sydney coalfield", 2nd US Mine Ventilation Symposium, Reno.

TAUZIEDE C. et POKRYSZKA Z., 1993, "Dynamic prediction of CH_4 emission in Longwalls", 25th International Conférence in Safety in Mines Research Institutes, Pretoria.

VAFAI K. and KIM S.J., 1990, "Fluid mechanics of the interface region between a porous medium and a fluid layer", International Journal of Heat and Fluid Flow, vol 11, n°3, pp 254-256.