Prace Instytutu Mechaniki Górotworu PAN
Tom 18, nr 2, czerwiec 2016, s. 3-13

Numerical analysis of selected flow phenomena in the area of coal waste dumps

Przemysław Skotniczny

Strata Mechanics Research Institute of the Polish Academy of Sciences, ul. Reymonta 27, 30-059 Krakow, Poland

Abstract

The article presents the method of developing the numerical model, discretisation of the calculation domain, discusses the applied numerical model and provides the results of air flow simulations for the selected areas of the "Waleska" coal waste dump in Łaziska Górne, Poland. The presented calculations for the two dominating wind flow directions do not take into account the porosity of the coal waste dump, and therefore the mass exchange between the coal waste dump and the free stream, concentrating mainly on the flow effects around the coal waste dump. This simplification was introduced to facilitate the calculation process, at the same time providing information about the potential areas under the hazard of the disadvantageous effect of spontaneous combustion.

Keywords: coal waste dump, numerical analysis, static pressure at the coal waste dump bank, air flow velocity around coal waste dumps

1. Introduction

The calculations for cases with complicated geometry and flows should be accompanied with a series of experiments to verify the results of numerical simulations. However, for matters related to mass exchange in coal waste dumps, the process of experimental verification of the numerical results on laboratory scale is very complex. The reasons for that are primarily the size of the coal waste dump – the recreation of the dimensions to scale (for example, 1:50) requires the use of an aerodynamic tunnel with a measurement chamber of appropriate size as well as flow conditions different from laboratory setting (specification of the atmospheric boundary layer, etc.)

In case of the lack of detailed experimental data, a good solution is to use data from processes on a similar scale and with similar mechanics. Many studies regarding the numerical analysis of mass exchange processes in porous media related to mining (Sensogut & Ozdeniz, 2005; Ejlali at al., 2009; Siyakatshana at al., 2011; Krawczyk, 2014; Dziurzyński, 2014), discuss the effective application of implemented numerical models in the available commercial codes.

The article presents the method of developing the numerical model, discretisation of the calculation domain and discusses the numerical model used for the calculations of air flow in the area of the "Waleska" coal waste dump in Łaziska Górne, Poland. The calculations were made using the CFD Ansys Fluent commercial software package.

2. Numerical model development

Access to the updated surveying map of the object was required before the development of the model. The map was digitised and vectorised using appropriate dedicated software. The vectorisation of the map – creation of a digital image of the analysed area – enabled the creation of a 1:1 scale three-dimensional model for further processing in the CFD preprocessor. In this case, the GAMBIT software from the Ansys CFD package was applied.

The process of creating a three-dimensional model was presented based on the example of the "Waleska" coal waste dump located in Łaziska Górne, Poland.

The "Waleska" coal waste dump is a relatively new object (created in the 1990s) and is completely formed. The available surveying map in the 1:1000 scale was digitised using a large-format scanner. The digitisation result is presented in Fig. 1.

The map presented in Fig. 1 is a raster image of the coal waste dump. In order to create a three-dimensional model, it is necessary to provide information about the coordinates of certain points on the coal waste dump – i.e. vectorisation.

The result of the vectorisation process (Fig. 2), was a number of points representing the shape of the individual layers of the coal waste dump (Fig. 2b). The next step is the creation of the three-dimensional model.

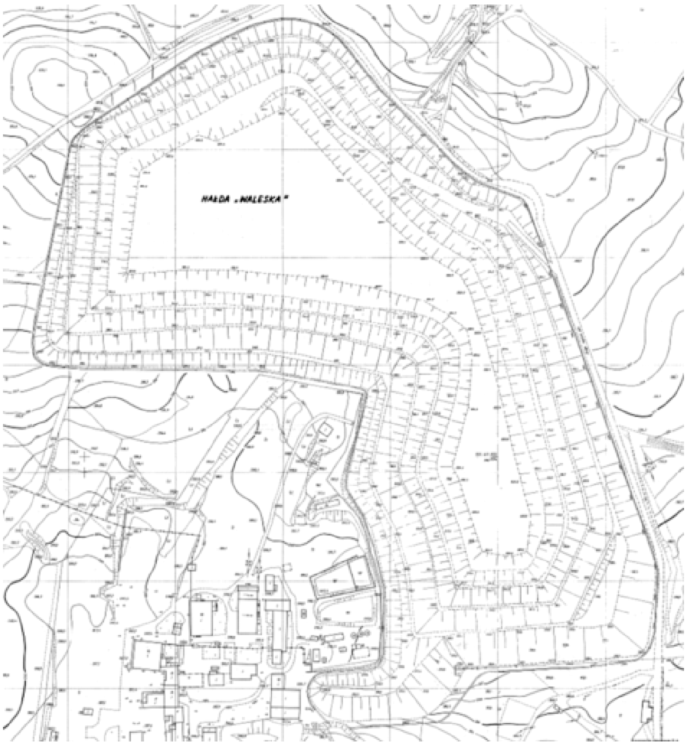


Fig. 1. Digital representation of the "Waleska" coal waste dump map

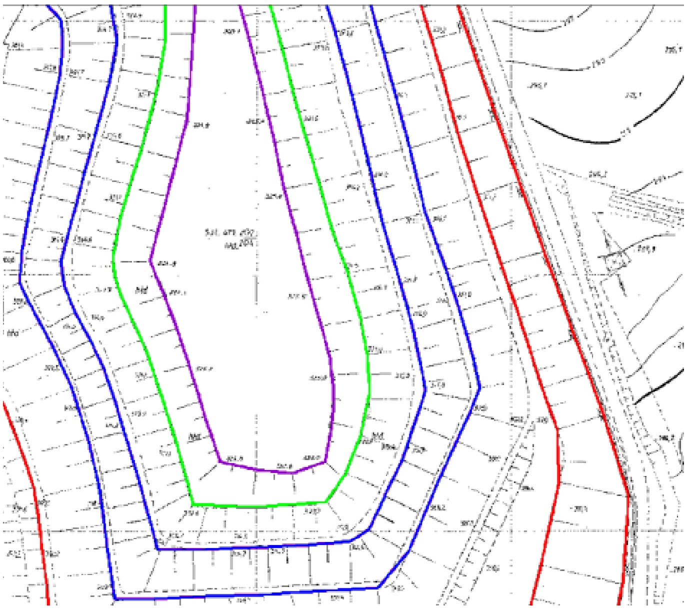


Fig. 2. Coal waste dump map vectorisation (part)

The reconstruction of individual layers of the coal waste dump was possible with the use of the GAMBIT preprocessor journal files (Fluent Use Manual, 2006). The resulting coordinates for individual points of the layer in the 1:1 scale were inserted in appropriate places of the journal file (Fig. 3). The result is presented in Fig. 4 – a screen shot of the GAMBIT software's main window together with the imported coordinates of the first layer of the coal waste dump.

```
/Journal File for GAMBIT 2.4.6,Database    2.4.4, ntx86
/Identifier wektoryzacja
...
vertex create coordinates 484.7254965637573 238.3096394493824 0
vertex create coordinates 499.4142815713807 234.8430515968607 0
vertex create coordinates 513.8359609064493 233.3453896114477 0
vertex create coordinates 526.6552595790505 238.0406068007173 0
vertex create coordinates 529.0589065846104 251.5146002903446 0
...
save
```

Fig. 3. Part of the Gambit journal file

After repeating the procedure for all layers of the coal waste dump and performing additional actions, such as verification of point locations, connecting points and creation of planes, a full, three-dimensional model of the analysed object in a 1:1 scale was created. The model is presented in Fig. 5.

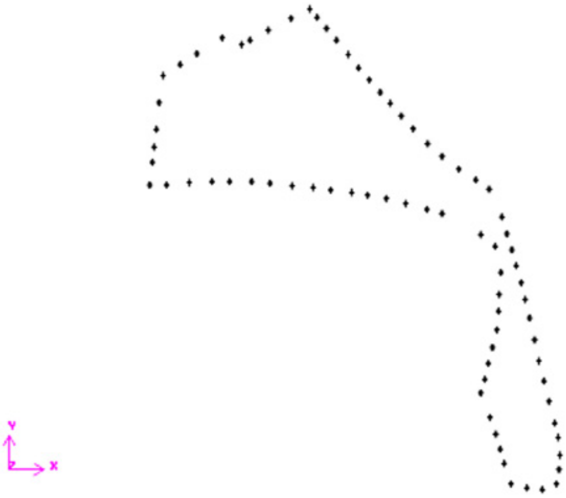


Fig. 4. Imported coordinates for the first layer of the coal waste dump

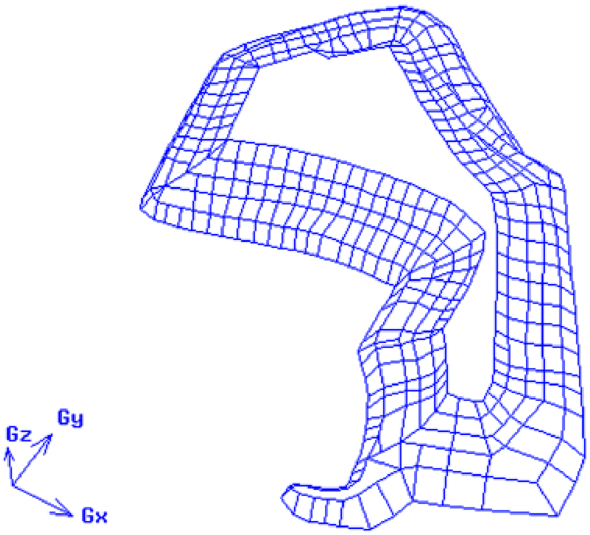


Fig. 5. Three-dimensional model of the coal waste dump

2.1. Determination of the calculation domain

In the matters related to fluid flows in atmospheric conditions, the selection of an appropriately formulated calculation domain is very important. The correctness of domain determination is affected by two major factors: the extent of the calculation domain (distances of the object around which the flow is analysed from the boundaries of the flow area) and the type of applied boundary conditions.

The values of coordinates of the area used in calculations are specified in the function of the dimensions of the object for which the values of individual flow parameters are determined. According to Hall (1997), the optimum distance between the inlet of the calculation domain and the analysed object, its left and right partition should be L1 ~ 5 h, outflow L ~ 10 h and top partition H ~ 6 h, with h as the height of the analysed object.

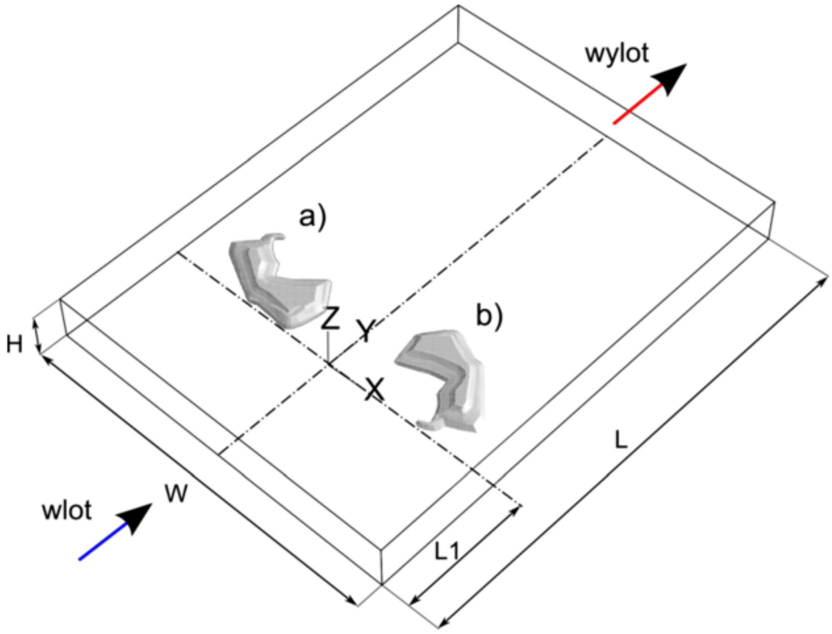


Fig. 6. Determined calculation domain. The figure presents two main positions of the coal waste dump in relation to the inlet of the calculation domain, a) E arrangement, b) SWW arrangement

Fig. 6 presents the schematic representation of the calculation domain for the analysed case of air flow around the mining coal waste dump. The a) case is for E air inflow and b) is for SWW air inflow. The selection of these two directions was not random – it was a result of the analysis of the average annual wind speed distribution (wind rose) for the discussed area. The values of the individual dimensions marked on Fig. 6 are as follows:

H = 330 m
W = 1100 m
L = 650 m
L1 = 300 m.

The described calculation area together with the coal waste dump model required preparation for further calculations. The next step involved the discretisation – dividing the total calculation domain volume into control volumes, for which numerical calculations could be provided.

2.2. Discretisation of the coal waste dump surface area and calculation domain

The discretisation of both the surface area of the coal waste dump and the volume of the calculation domain was completed using the Gambit preprocessor, part of the ANSYS Fluent package. In order to gain control over the calculation grid creation process, the discretisation was executed in steps. The first step involved the creation of a calculation grid for the coal waste dump surface area.

Due to the complex shape of the coal waste dump area, it was necessary to apply a TRI-PAVE non--structural grid (Fig. 7). The verification of the accuracy of the grid is critical during the discretisation of the calculation domain. This is achieved by analysing the value of the skewness index for individual grid elements. According to the notes in the FLUENT software package manual (Fluent Use Manual, 2006), the skewness of grid elements should not exceed the value of 0.85. In the discussed case, the skewness value did not exceed 0.65. After the creation of the calculation grid on the surface area of the coal waste dump, the entire calculation domain volume was discretised. Also in this case, a non-structural TRI = PAVE grid with the total number of elements of 2,046,726 was used.

2.3. Mathematical model, boundary conditions

Air flows in the area of the atmospheric boundary layer are categorised as turbulent flows. For this reason, the Navier–Stokes equation closure model was applied. The ***k-ω-SST*** model was selected from the multiple models available in modern CFD solvers (Fluent, CFX, etc.). The selection of this particular model

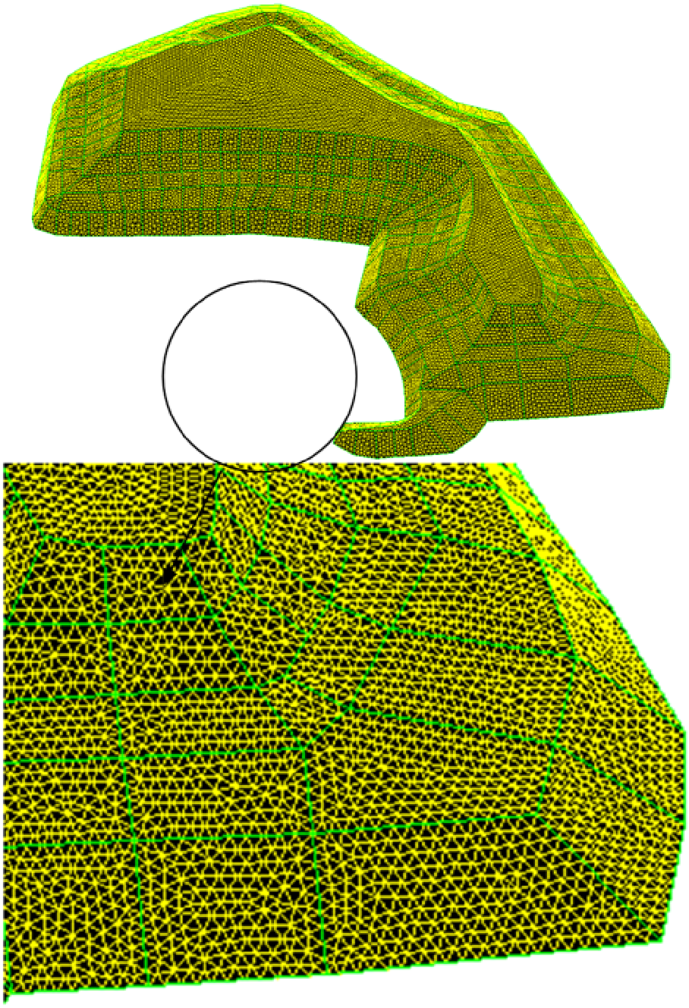


Fig. 7. Calculation grid for the coal waste dump surface area with a magnified part

is largely the result of its increased accuracy for the determination of turbulent values within the boundary layer compares to single and dual equation models.

The closure of the motion equations for this model is expressed with the following equations, describing the turbulence kinetic energy transport *k* [1] and dissipation speed w [2] (Fluent Use Manual, 2006).

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial x_i}(\rho k u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_k \frac{\partial k}{\partial x_j}\right)+\widetilde{G}_k - Y_k + S \qquad [1]$$

$$\frac{\partial}{\partial t}(\rho \omega)+\frac{\partial}{\partial x_i}(\rho \omega u_i)=\frac{\partial}{\partial x_j}\left(\Gamma_\omega \frac{\partial \omega}{\partial x_j}\right)+G_\omega - Y_\omega + S_\omega + D_\omega \qquad [2]$$

The type and application of boundary conditions of the discussed model are presented in Fig. 8. For the discussed flow case, for which the flow of air – a viscous and non-compressible fluid – is analysed, the velocity inlet partition was applied on the inlet to the calculation domain and the velocity outflow partition – on the outflow. The remaining partitions are as presented in Figure 8.

The values of the velocity for the inlet setpoint are derived from the average annual wind direction and speed analysis for the coal waste dump location (Meteorological data 2009-2010). Fig. 9 is a graph presenting the total wind rose recorded for the analysed area.

As presented in Fig. 9, the most frequent wind inflow direction to the analysed object is SWW. Therefore, it can be assumed to be the dominant direction.

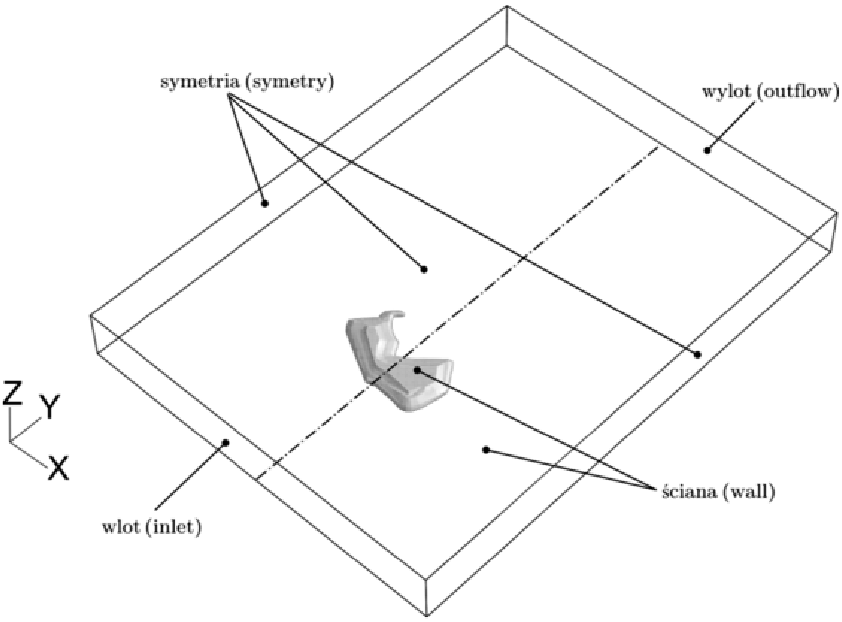


Fig. 8. Boundary conditions of the calculation domain

However, the analysis of the wind speeds presented in Fig. 10 shows that the second most frequent wind speed value is from the E-SEE direction. For this reason, the analysis provides calculations for two directions of wind inflow to the coal waste dump – E and SWW.

In order to make the comparison of simulation results easier, the same wind speed value of **U = 3.5 m/s** was assumed for both directions.

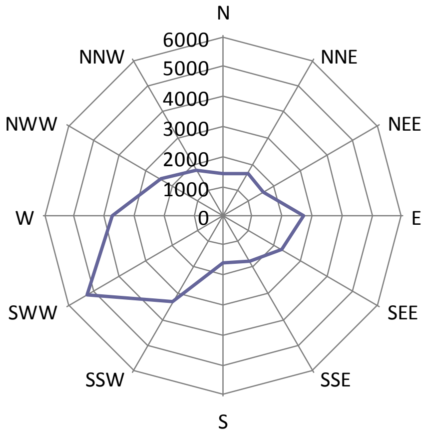


Fig. 9. Wind rose

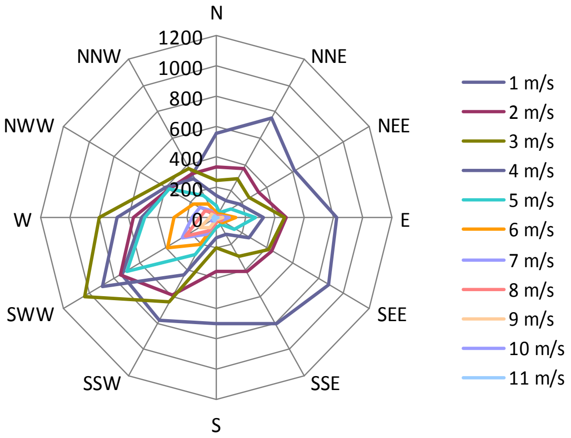


Fig. 10. Wind speeds distribution

3. Simulation results

The results of numerical simulations of air flows around the mining coal waste dump area for are presented below for the two selected wind directions.

The numerical analysis of air flows around the coal waste dump focused mainly on the static pressures distribution on the surface and the distribution of speeds around the coal waste dump. According to the data from model studies, the distributions of these two values have significant effects on adverse thermal phenomena in the coal waste dump.

The first case, marked in Fig. 6 as a) concerns the wind inflow with the speed of U = 3.5 m/s from the east (E). The schematic representation is provided in Fig. 11.

The calculations provided the distribution of static pressures on the banks of the coal waste dump, presented in Fig. 12.

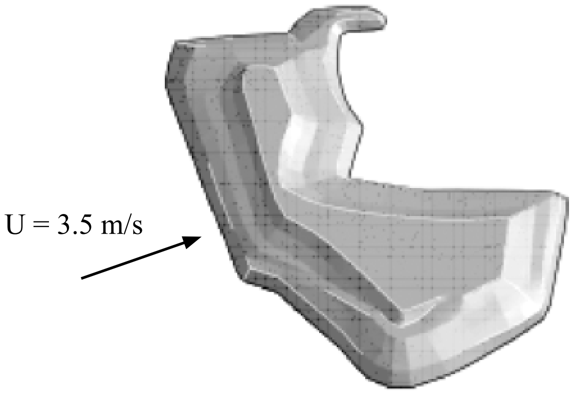


Fig. 11. Schematic representation of air inflow from the east

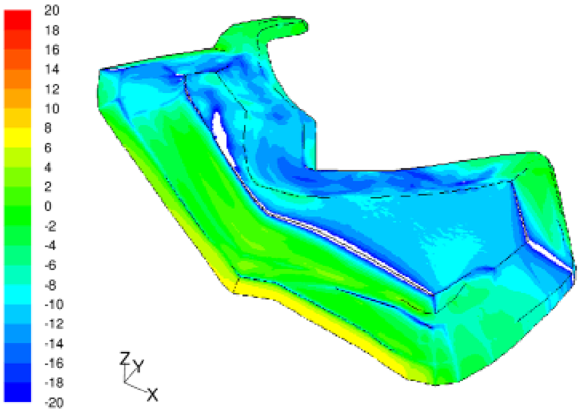


Fig. 12. Distribution of static pressures on the banks of the coal waste dump for the eastern wind direction

The values of relative static pressures (Fig. 12) are presented in the $-20 \div 20$ Pa scale. The minimum pressure values ($p_s = -20 \div -18$ Pa) can be observed mostly in the top parts of the coal waste dump, on the surfaces close to parallel on both sides of the coal waste dump and from the leeward side. The reason for this pressure distribution is the nature of air flow around the coal waste dump. The geometry of the coal waste dump body favours the creation of large recirculation zones on the leeward side of the coal waste dump. This can be observed by the analysis of speed distribution in select vertical cross-sections in the Y-Z plane depicted in Fig. 13 a), b), c) and d).

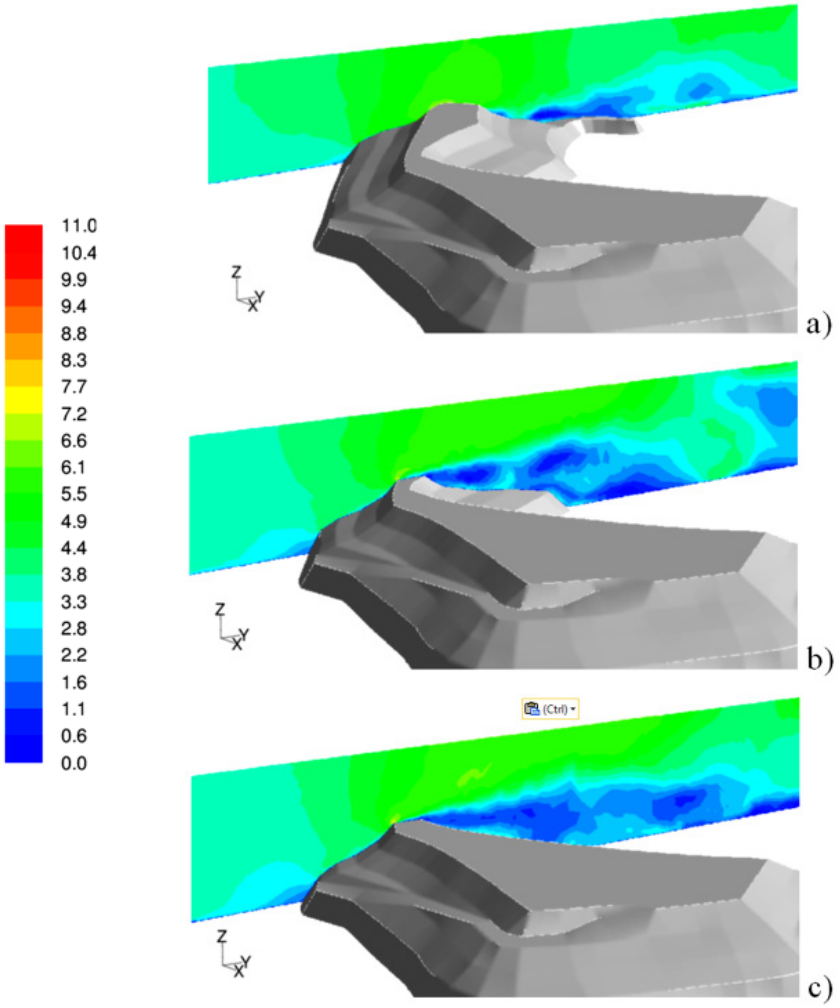
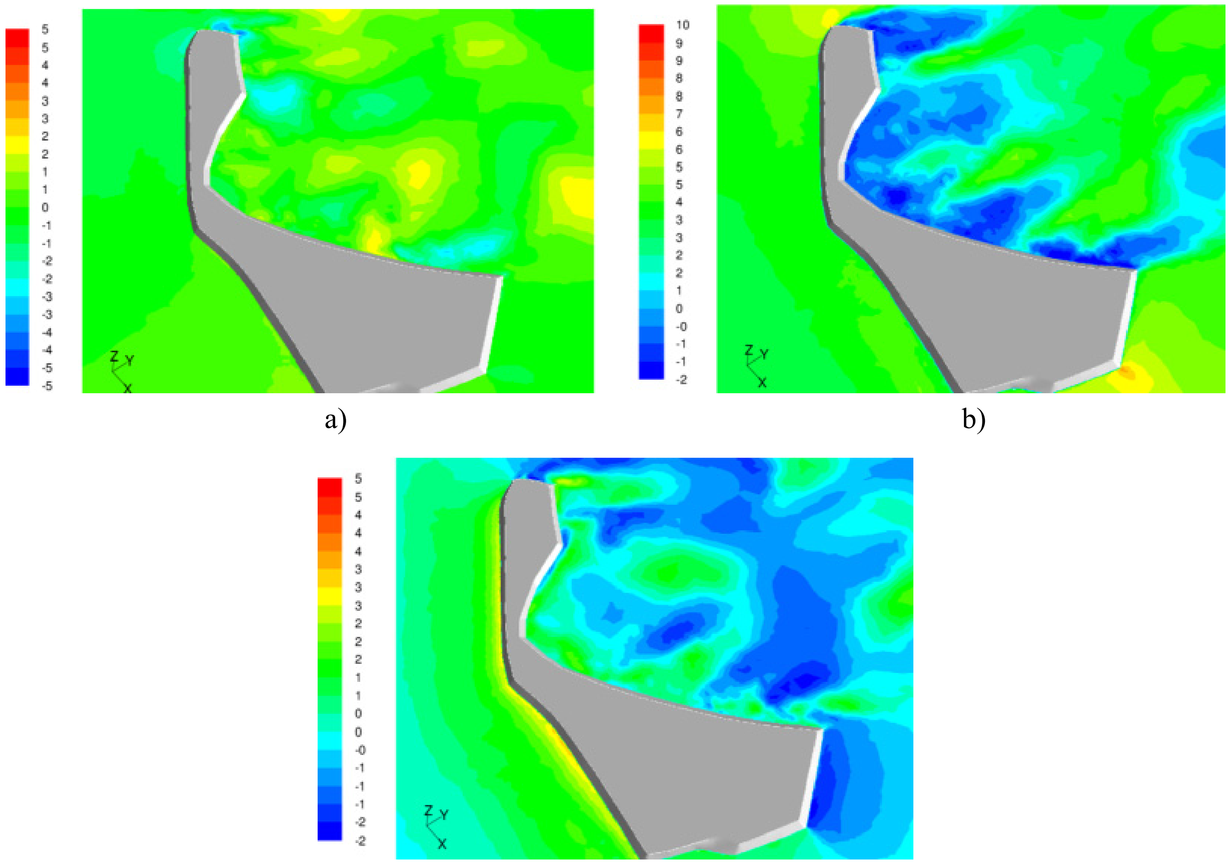


Fig. 13. Wind speed distribution in the YZ plane for a) X = 240 m, b) X = 400 m and c) X = 450 m



a) b)

a)

Fig. 14. Air recirculation on the south-western side of the coal waste dump, horizontal cross-section, 5 m from the top part of the coal waste dump. a) Ux, b) Uy, c) Uz [m/s]

The speed distribution in vertical cross-sections show an extensive air flow recirculation zone on the leeward (south-western) side from the sides of the coal waste dump.

The second case, marked in Fig. 6 as b) concerns the wind inflow with the speed of U = 3.5 m/s from the south-south-east (SSE). The schematic representation is provided in Fig. 15.

In this case, the distribution of static pressures on the banks of the coal waste dump (Fig. 16) is different from the distribution in case of an eastern wind direction. The extensive subatmospheric pressure

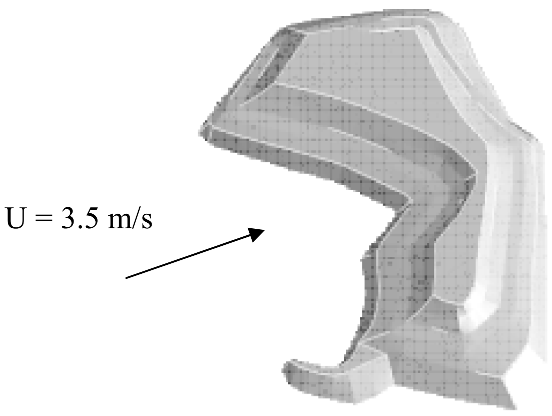


Fig. 15. Schematic representation of air inflow from the south-south-east

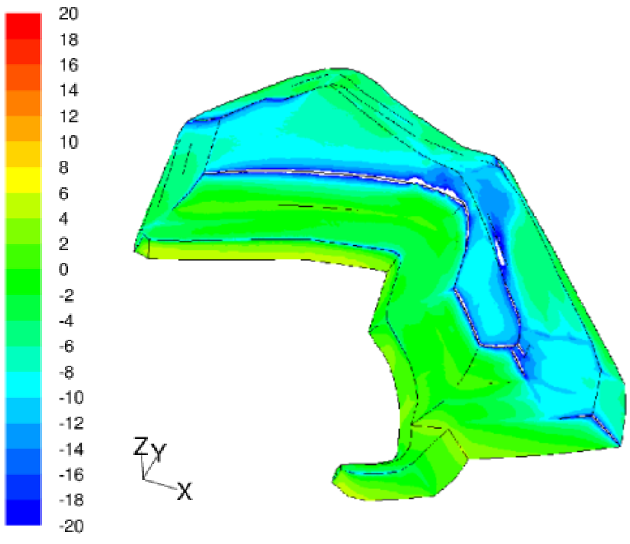


Fig. 16. Distribution of static pressures on the banks of the coal waste dump for the SSE wind direction

zone observed in the previous case is replaced with local areas with the relative static pressure values of around –18 Pa. This is the case primarily for the “leading edge” of the coal waste dump, from the windward side. It is a result of the better aerodynamic shape of the coal waste dump banks from the SSE direction.

The effect of the coal waste dump shape on the distribution of static pressures on the SSE banks is clearly visible during the analysis of the velocity vector analysis for the air flow around the coal waste dump (Fig. 17). For the 3 control cross-section included in Fig. 17 a), b) and c), the retraction of the air flow recirculation zone from the eastern banks of the coal waste dump can be observed. The lack (Fig. 17 a) or relocation of the recirculation zone (Fig. 17 b and c) towards the flow direction results in a decreased pressure gradient on the bank.

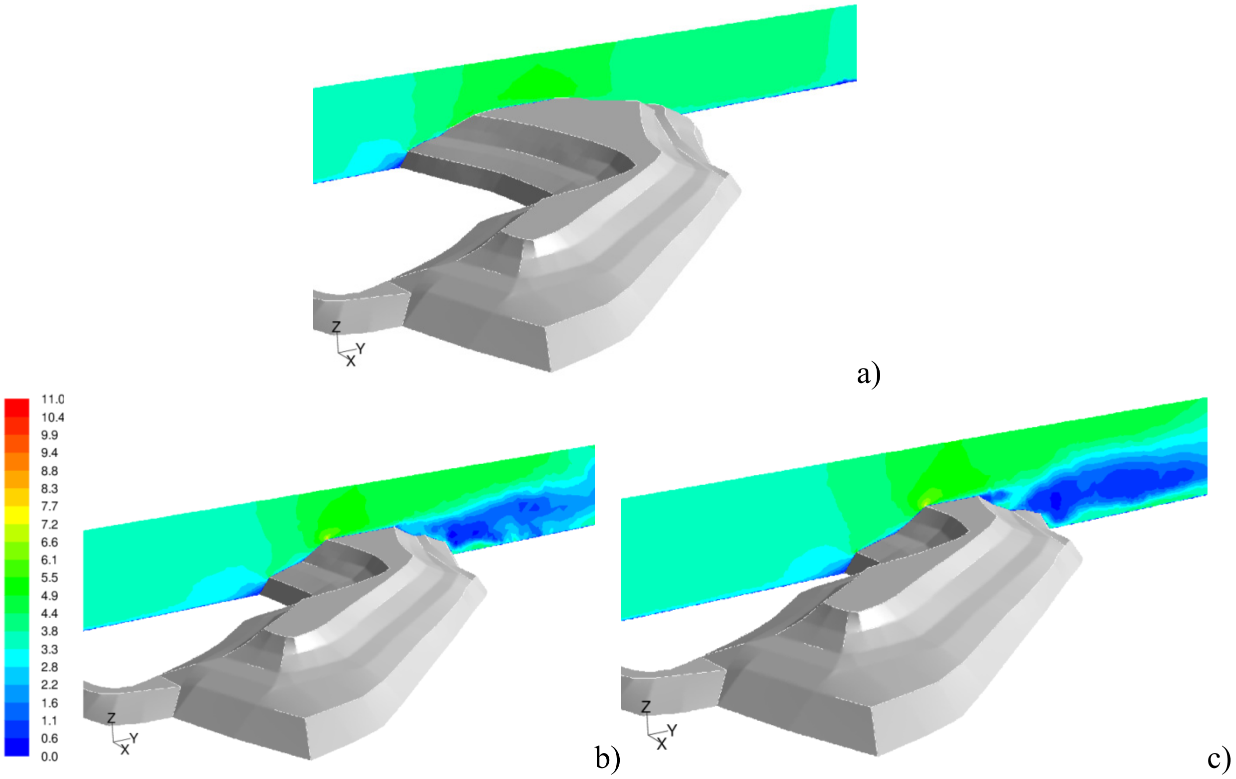


Fig. 17. Wind speed distribution in the YZ plane for a) X=240 m, b) X=400 m and c) X= 450 m

The difference in the flow between the two analysed cases is visible in Fig. 18. In this case, as before, a distribution of the individual values of the speed vector Ux, Uy and Uz parallel to the flow, 5 m from the top part of the coal waste dump are presented.

The distribution of the values of individual components of the velocity vector around the coal waste dump for SSE air flow shows a more coherent air recirculation zone, compared to the previous case. The Uy main flow component, particularly significant in the discussion, shows a smaller range of the occurrence of secondary (negative) speed value compared to the previous case.

4. Conclusions

The numerical analysis of the static pressures distribution and the velocity vector for the air flow around the coal waste dump can be significant for the correct assessment of the endogenous fire hazard in the coal waste dump. The basis for accurate assessment is the application of validated mathematical models describing the mass exchange in the area of the analysed objects as well as an extensive database of measurement cases.

The article shows the differences in the properties of air flow around the coal waste dump for the two main wind directions – E and SSE. In the case of SSE winds, a potentially lesser impact of the flow effects was found for the creation of endogenous fire starting points, due to better aerodynamic shape of the air

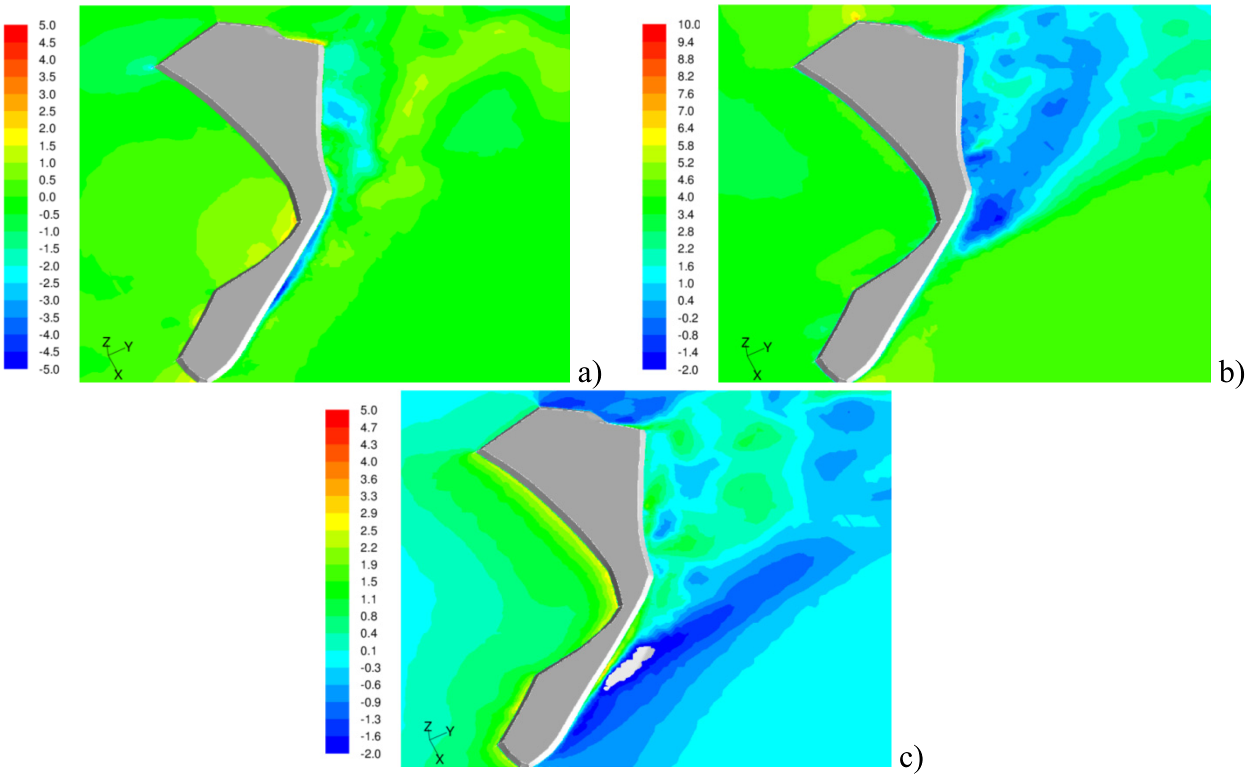


Fig. 18. Air recirculation on the eastern side of the coal waste dump, horizontal cross-section, 5 m from the top part of the coal waste dump. a) Ux, b) Uy, c) Uz [m/s]

flow both on the windward and leeward sides of the coal waste dump. Despite that, there is one know case of a single thermal incident in the analysed coal waste dump. A fire starting point occurred at the base of the coal waste dump, from the NEE side.

The numerical analysis shows that for the air flow from the SSE direction, on the leeward side (NEE), a relatively extensive recirculation flow zone occurs at the base of the coal waste dump, with potential adverse impact on the intensity of the oxidation processes for the material stored in the area.

The “Waleska” coal mining coal waste dump is a relatively new object, constructed with proper care given to the distribution of the material, bank slopes and appropriate compactness of the material. However, the results presented in this article show a number of air flow effects, including static pressures distribution on the banks resulting from the particular wind rose distribution in the coal waste dump area and adverse air flow recirculation zones from the leeward side of the coal waste dump due to the shape of the banks, which could result in an increased susceptibility of the stored material to self-heating. The reduction of the adverse flow effects may be indirectly accomplished by:

Maintenance of the technical condition of coal waste dump banks – the naturally occurring erosion crevices, due to natural phenomena, require ongoing maintenance. Furthermore, a poorly documented phenomenon of stored material dampening occurs, probably due to a weak drainage system or the hygroscopic properties of the material. Examples of such an effect for the “Waleska” coal waste dump are the north-eastern and northern sides, where the material is damp regardless of the weather conditions. This condition may in the future lead to the creation of an extensive network of underground channels, facilitating the aeration of the material in the coal waste dump.

Planting high-growing plants on the coal waste dump (trees, bushes). This will decrease the air flow velocity in the direct area of the coal waste dump surface, which can effectively reduce the intensity of aeration processes. The possible increase in the porosity and permeability of soil in the forestation area due to the roots network should, however, be taken into consideration.

Acknowledgements

Author would like to thank the Directorate of Coal Mine “Boleslaw Śmiały” and employees of the Department of Environmental Protection for providing the maps and all needed information on “Waleska” coal waste dump.

Praca została wykonana w ramach prac statutowych 2016 realizowanych w IMG PAN w Krakowie, finansowanych przez Ministerstwo Nauki i Szkolnictwa Wyższego.

Bibliography

Dziurzynski W., Krach A., Palka T., 2014: *Computer simulation of the propagation of heat in abandoned workings insulated with slurries and mineral substances*. Archives of Mining Sciences, vol. 59, issue 4, No 1, p. 3-23.

Ejlali A, S. M. Aminossadati, Ejlali A *Numerical analysis of fluid flow and heat transfer through a reactive coal stokpile,* Seventh International Conference on CFD in the Minerals and Process Industries

CSIRO, Melbourne, Australia 9-11 December 2009.

Fluent User Manual, ANSYS Fluent, 2006.

Hall RC (Ed.), 1997: Evaluation of modeling uncertainty. CFD modeling of near-field atmospheric dispersion. Project EMU final report European Commission Directorate General XII Science, Research and Development Contract EV5V--CT94-0531, WS Atkins Consultants Ltd., Surrey, 1997

Krawczyk J., Janus J., 2014: *Modeling of the Propagation of Methane from the Longwall Goaf, Performed by Means of a Two-Dimensional Description*. Archives of Mining Sciences, vol. 59, issue 4 p. 851-868.

Meteorological data 2009-2010, Katowice Muchowiec.

Sensogut C., Ozdeniz A.H., 2005: *Statistical modelling of stockpile behaviour under different atmospheric conditions – Western Lignite Corporation (WLC) case*. Fuel, 84:1858-1863.

Siyakatshana N., Onno Ubbink, Kekana J., Wessels G., 2011: *Steady state cfd modelling of carbon dioxide release and Methananogenesis in coal heaps* Second African Conference on Computational Mechanics – An International Conference – AfriCOMP11 January 5-8, 2011, Cape Town.

Analiza numeryczna wybranych zjawisk przepływowych w otoczeniu zwałowisk odpadów poweglowych

Streszczenie

W artykule zaprezentowano metodę konstrukcji geometrii modelu numerycznego, dyskretyzację domeny obliczeniowej oraz omówiono wyniki obliczeń wielkości przepływowych dla wybranych obszarów zwałowiska odpadów pogórniczych „Waleska” zlokalizowanego w Łaziskach Górnych. Przedstawione wyniki obliczeń dla dwóch dominujących kierunków przepływu powietrza nie uwzględniają wymiany masy pomiędzy bryłą zwałowiska a omywająca go struga powietrza (zwałowisko traktowane jest jako ośrodek nieporowaty). Wprowadzone uproszczenie przypadku przepływowego miało na celu zmniejszenie czasu obliczeń, przy równoczesnym dostarczeniu informacji na temat obszarów zwałowiska o potencjalnie zwiększonym ryzyku wystąpienia zjawiska samozagrzewania materiału deponowanego.

Słowa kluczowe: zwałowisko odpadów powęglowych, analiza numeryczna, rozkład ciśnień statycznych na zboczu zwałowiska, rozkład wektora prędkości w otoczeniu zwałowiska