

D. JĘDRZEJCZYK*[#], M. HOJNY*, M. GŁOWACKI*

DEVELOPMENT OF SOFTWARE FOR THE SIMULATION OF ROLLING STEEL UNDER THE COEXISTENCE OF LIQUID AND SOLID STATE

ROZWÓJ OPROGRAMOWANIA DO SYMULACJI WALCOWANIA STALI W WARUNKACH WSPÓLISTNIENIA FAZY CIEKŁEJ I STAŁEJ

The paper presents the results of the application simulating the rolling process of steel in terms of coexistence of liquid and solid phases. The created mathematical models can be the basis for creation of systems that simulate the final phase of the continuous casting process relying on using a roller burnishing machine for continuous casting of steel. For a complete description of the performance of the material during deformation in these conditions, the constructed mathematical model is a fully three-dimensional model and consists of three parts: thermal, mechanical, and density variation submodels. The thermal model allows the prediction of temperature changes during plastic deformation of solidifying material. The mechanical model determines the kinetics of plastic continuum flow in the solid and semi-solid states, and the resulting deformation field. The temperature of the process forces supplementing the description of the performance of the material with a density variation model that allows the prediction of changes in the density of the material during the final phase of solidification with simultaneous plastic deformation. For the purpose built model, experimental studies were performed using a physical simulator Gleeble 3800[®]. They allowed the determination of the necessary physical properties of the metal within the temperature of change state. In addition to presenting the developed models the work also includes the description of the author's application that uses the above mathematical models. The application was written in the fully object-oriented language C++ and is based on the finite element method. The developed application beside the module data input, also consist of a module of three-dimensional visualization of the calculations results. Thanks to it, the analysis of the distribution of the particular rolling parameters in any cross-section of the rolled strip will be possible. The paper presents the results of the authors' research in the area of the advanced computer simulation.

Keywords: Semi-Solid State, Finite Elements Method, Mathematical Modelling, Physical Modelling

W pracy zaprezentowane zostały wyniki działania aplikacji pozwalającej na symulację procesu walcowania stali w warunkach współlistnienia fazy ciekłej i stałej. Opracowane modele matematyczne mogą być podstawą tworzenia systemów symulujących ostatnią fazę procesu ciągłego odlewania stali polegającej na dogniataniu odlanego pasma rolkami maszyny COS. W celu pełnego opisu zachowania się materiału podczas odkształcania w omawianych warunkach, zbudowany model matematyczny jest modelem w pełni trójwymiarowym i składa się z 3 części: termicznej, mechanicznej i zmiany gęstości. Model termiczny pozwala na przewidywanie zmian temperatury w trakcie odkształcania plastycznego krzepnącego materiału. Model mechaniczny określa kinetykę płynięcia plastycznego ośrodka ciągłego w stanie stałym i półciekłym oraz wynikające stąd pole odkształceń. Temperatura analizowanego procesu wymusza uzupełnienie opisu zachowania materiału o model zmiany gęstości pozwalający na przewidywanie zmian gęstości materiału w trakcie końcowej fazy krzepnięcia z równoczesnym odkształceniem plastycznym. Na potrzeby budowanego modelu wykonane zostały badania doświadczalne przy użyciu symulatora fizycznego GLEEBLE 3800[®]. Pozwoliły one na określenie niezbędnych własności fizycznych metalu w zakresie temperatur zmiany stanu skupienia. Poza przedstawieniem opracowanych modeli w pracy zwarty został również opis autorskiej aplikacji wykorzystującej w/w modele matematyczne. Aplikacja została napisana w pełni obiektowym języku C++ i bazuje na metodzie elementów skończonych. Opracowana aplikacja poza modulem wprowadzania danych wejściowych posiada również moduł trójwymiarowej wizualizacji wyników obliczeń. Dzięki temu możliwa jest analiza rozkładu poszczególnych parametrów walcowania w dowolnym przekroju walcowanego pasma. W pracy przedstawiono wyniki badań autorów w dziedzinie zaawansowanej symulacji komputerowej.

* AGH UNIVERSITY OF SCIENCE AND TECHNOLOGY, FACULTY OF METALS ENGINEERING AND INDUSTRIAL COMPUTER SCIENCE, AL. A. MICKIEWICZA 30, 30-059 KRAKOW, POLAND

[#] Corresponding author: d.jedrzej@agh.edu.pl

1. Introduction

At the present time, particular attention is paid to the production of products in such a way as to reduce the manufacturing cost while increasing productivity. At the same time due to the standards for CO₂ emissions of steel sheet production process should also be efficient in this respect. For several years, are created all sorts of solutions to reduce sheet production stages. The resulting technologies such as ISP (Inline Strip Production), DSC (Direct Strip Casting), UTHS (Ultra-Thin Hot-Strip), ESP (Endless Strip Production) and many others. Figure 1 presents the trend in the construction of the production line. On this basis the reduction of some steps which, in modern technologies can be omitted. This makes it possible to maintain the correct parameters stored product using the standards listed above.

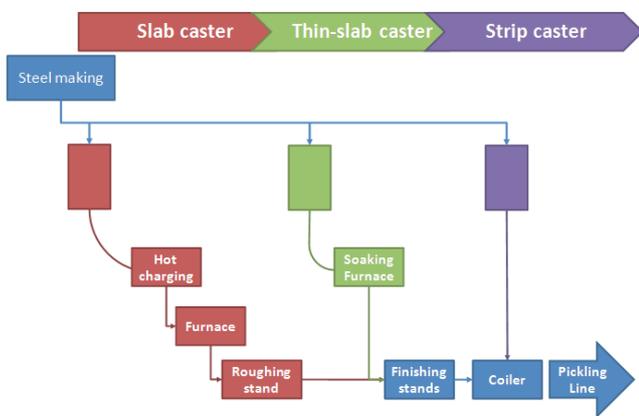


Fig. 1. Technological development in the casting and rolling stages of the steel production process

2. Testing of mushy steel rheology

The experimental work was done in Institute for Ferrous Metallurgy in Gliwice, Poland using Gleeble[®] thermo-mechanical simulator. The steel used for the experiments was the C45 grade steel having 0.45% of carbon content. In all cases, experiments were performed according to the following schedule: initial stage: sample preparation divided into several sub stages (e.g., thermocouple assembly); die selection, etc, melting procedure and deformation process. It is good practice to test materials in isothermal conditions [1]. Unfortunately, this is not possible for semi-solid steel. Nevertheless, the condition should be as close to isothermal as possible due to the very high sensitivity of material rheology to even small variations of temperature [1]. The basic reason for uneven temperature distribution inside the sample body on the Gleeble[®] simulator is the contact with hot copper handles presented in Figure 2.

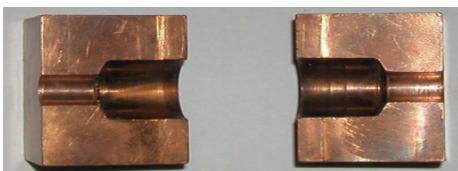


Fig. 2. The short contact zone handles used in experiments (hot handle)

The estimated liquidus and solidus temperature levels of the investigated steel are: 1495°C and 1410°C, respectively. Thermal solution of the theoretical model has crucial influence on simulation results, since the temperature has strong effect on remaining parameters. The resistance sample heating and contact of the sample with cold cooper handles cause non-uniform distribution of temperature inside heated material, especially along the sample. The semi-solid conditions in central parts of the sample cause even greater temperature gradient due to latent heat of transformation. Such non-uniform temperature distribution is the source of significant differences in the microstructure and hence in material rheological properties. During the experiments samples were heated to 1430°C and after maintaining at constant temperature were cooled down to the required deformation temperature. In case of heating the heat generated is usually not known because the Gleeble[®] equipment uses an adaptive procedure for resistive heating controlled by temperature instead of current flow. Hence, the actual heat generated by current flow (in fact the rate of heat generation Q) has to be calculated using inverse procedure. In this case the objective function (F) was defined as a norm of discrepancies between calculated (T_c) and measured (T_m) temperatures at a checkpoint (steering thermocouple position: TC4 in Figure 3) according to the following equation:

$$F(Q) = \int_{\tau_0}^{\tau_i} [T_c(Q, r, z, T) - T_m(r, z, T)] d\tau \quad (1)$$

where: τ is the time variable, Q is the rate of heat generation.

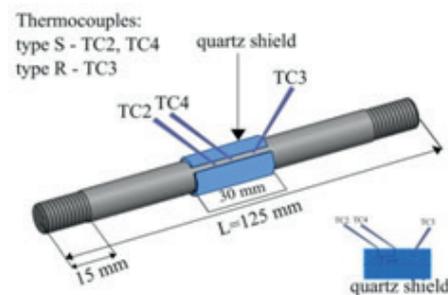


Fig. 3. Sample used for the experiments. TC2, TC3 and TC4 thermocouples

In the final stage of physical test, the temperature difference between core of the sample (TC3 thermocouple position) and its surface (TC4 thermocouple position) can be significant. In all cases the core temperature was higher than surface temperature. Differences between these two reach around 30°C for cold handle (handle with long contact zone) and about 40°C for hot handle. The results of numerical simulation are in agreement with experiments. Figure 4 presents the temperature distributions in the cross section of sample tested at 1380°C right before deformation (variant with hot handle). One can observe major temperature gradient between die-sample contact surface. However, difference between experimental and theoretical core temperatures for hot handles was only 3°C (calculated core temperature was equal to 1417°C and measured one was equal to 1420°C).

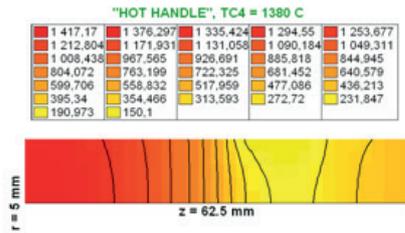


Fig. 4. Distribution of temperature in the cross section of sample tested at temperature 1380°C right before deformation (variant with hot handle)

Compression and tension tests were performed, according to the given methodology [2]. During experiments die displacement, force and temperature changes in the deformation zone were recorded. The computer simulations were performed as well. All series of tests and computer simulations were done using long contact zone between samples and simulator jaws (cold handle). The deformation zone had the initial height of 62.5mm. The sample diameter was 10mm. The samples were melted at 1430°C, and then cooled to deformation temperature. During the tests each sample was subjected to 10mm reduction of height. Results of each test were used for inverse analysis to compute yield stress curve parameters. Figure 5 shows strain-stress curves at several strain rate levels for temperature 1300°C. The relationships were calculated using experimental methodology detailed described in [2].

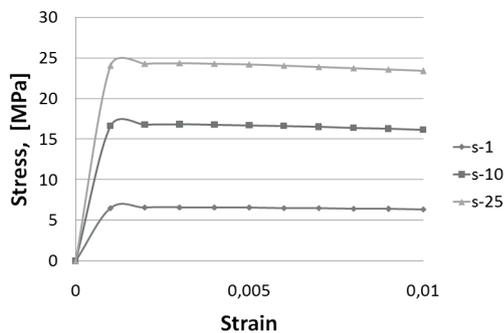


Fig. 5. Stress-strain curves at several strain rate levels for temperature 1300°C

3. Mathematical model

The mathematical model allowing to simulate the process of deformation of the steel must contain a model of changes in temperature and a deformation model (Fig. 6). To make the calculation results similar to the real ones, the created models must be supplemented with experimental studies. Such studies have been carried out. How to perform this research, the material used and the data obtained are presented in the previous chapter. Due to the fact that the research conducted by our team applies to deformations in extra high temperatures in which you can observe the coexistence of liquid and solid phases, the above models must be supplemented into the model of variable density.

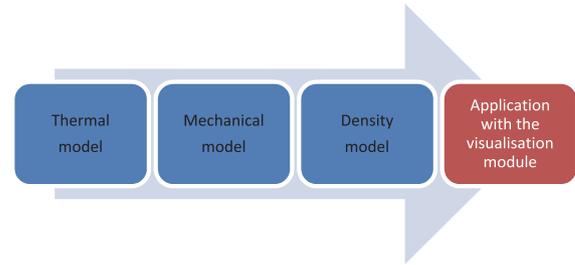


Fig. 6. Models included in application for simulation of deformation steel with semi-solid state

3.1. Thermal model

The material behavior above the solidus line is strongly temperature dependent. There are a few characteristic temperature levels between solidus and liquidus which decide about the steel crack resistance. The temperature has a great influence on almost all material properties. Hence, a proper thermal model is extremely important. In the presented model it was based on the general diffusion equation for a three-dimensional thermal problem, which is called Fourier-Kirchhoff equation, written in a form:

$$\frac{\partial}{\partial x} \left(\lambda_{xx} \frac{\partial T}{\partial x} \right) + \frac{\partial}{\partial y} \left(\lambda_{yy} \frac{\partial T}{\partial y} \right) + \frac{\partial}{\partial z} \left(\lambda_{zz} \frac{\partial T}{\partial z} \right) + \left(Q - \rho c_p \frac{\partial T}{\partial t} \right) = 0 \quad (2)$$

In (2) T is the temperature distribution, λ is the heat conduction coefficient, Q is the rate of heat generation due to plastic deformation, c_p denotes the specific heat and ρ is the density.

As a result, we obtain a matrix notation solution heat conduction equation for the case of unsteady state:

$$\left(2H + \frac{3}{\Delta t} C \right) T_1 + \left(H - \frac{3}{\Delta t} C \right) T_0 + 3P = 0 \quad (3)$$

where: C is a heat capacity matrix.

Equation (3) is an algebraic equation, which allows the calculation of the nodal temperatures T_i at the time Δt when the selected temperature T_0 at $t = 0$. Equation (3) is derived by assuming a linear relationship between the temperature in the interval of time Δt . The applied solution was used Galerkin integration scheme, due to the fact that it gives good results for the boundary conditions existing in the rolling. But in addition to the methods used are also other methods of integration, eg. Euler scheme and Crank-Nicholson. These methods differ from each other constant coefficients appearing in equation (3). Solution of equation (2) has to satisfy boundary conditions. For the rolling process in order to track changes in temperature, mixed boundary conditions applied, consisting of a second and third boundary condition [3].

3.2. Mechanical model

The mechanical model is responsible for calculation of strain, strain rate and stress tensors as well as their distribution in the controlled volume. The deformation process has been divided into deformation steps. Mechanical solution is based on the optimization of appropriate work functional

that depends on a number of parameters. The rigid-plastic formulation has been used to fulfil the Levy-Misses flow law. The work functional subjected to optimization in subsequent steps of deformation, consists of three main parts and can be written as follows:

$$W = \underbrace{\int_V \frac{\sigma_i \Delta \varepsilon_i dV}{W_\sigma}} + \underbrace{\int_V \lambda \left(\Delta \varepsilon_x + \Delta \varepsilon_y + \Delta \varepsilon_z - \frac{\Delta \rho}{\rho} \right) dV}_{W_\lambda} + \underbrace{\frac{m}{\sqrt{3}} \int_S \sigma_y \bar{\Delta} dS}_{W_f} \quad (4)$$

In equation (4) W_σ is the plastic work, W_λ is the penalty for failure to meet either the incompressibility or mass conservation condition. Both of them are responsible for the volume changes of the deformation zone. Finally, W_f is the work of friction forces. In the relation (4) σ_i is the effective stress which is calculated using unique yield stress functions developed by the authors [1]. The variable λ is the coefficient matching the hydrostatic stress, $\Delta \varepsilon_i$ the effective strain increment, $\Delta \varepsilon_x$, $\Delta \varepsilon_y$, and $\Delta \varepsilon_z$ are the step increments of strain tensor components, $\Delta \rho$ describes the density changes, m is the relative local displacement of metal and tool whilst m denotes the friction factor. In case of material incompressibility the work W_λ disappears step by step while the optimization is in progress. The sum of the three presented work components given by relationship (4) is the basis of the solution for both solid and mushy material. The discretization of the mechanical problem and the solution has been performed in usual finite-element manner [1].

3.1. Variable density model

Density is one of the critical factors affecting both the temperature and the mechanical model and has to be taken into consideration. Its changes that occurs with the progressive variations in physical state of steel are ruled by three main phenomena: the formation of the solid phase as a result of solidification, thermal shrinkage of solid and liquid phases and the movement displacement of the liquid phase in the matrix forming a porous solid phase.

The total derivative of density in any small area of the body with respect to time is a sum of equations (Eq. 5).

$$\frac{\partial \rho}{\partial t} = \underbrace{(\rho_s d_s + \rho_l d_l) \left(\frac{\rho_s}{\rho_l} - 1 \right) \frac{\partial d_s}{\partial t}}_{\text{part A}} - \underbrace{(\beta_s \rho_s d_s + \beta_l \rho_l d_l) \frac{\partial T}{\partial t}}_{\text{part B}} - \underbrace{\rho_l d_l \nabla v}_{\text{part C}} \quad (5)$$

The derivative of the density with respect to time during solidification is dependent on the fraction of solid phase and can be calculated from first part of equation (5) called as "part A", where: ρ_s describe density of the solid phase, ρ_l - density in the liquid phase and d_l - volume fraction liquid phase. Part B in equation (5) describe density changes due to thermal contraction can be described by the following relationship. In this formula symbol β_s describes volume contraction in the solid phase whereas β_l - volume contraction in the liquid phase.

Density changes caused by liquid phase flow through a porous medium consisting of solid phase skeleton can be expressed taking into account the Darcy flow law (Eq. 5 - part C). In this equation velocity v is given by (6):

$$v = -\xi_{op} \nabla (p - \rho_l g h) \quad (6)$$

Symbols used in equation (6) denote: ξ_{op} the coefficient of permeability of a porous medium and h the distance between solid particles. The relative velocity and pressure in the equation (5) introduce large computational problems.

4. Application for rolling simulation

As a result of theoretical considerations on the models appearing in this issue of rolling steel in semi-solid state and obtained after discretization of matrix notation. Thus prepared systems of equations must be solved by the methods of solving systems of linear equations. However, before the equations prepared in the form of matrices are solved, the matrices must be arranged in a suitable manner to create appropriate methods of operations on matrices, which will be used in the algorithms for solving systems of linear equations.

The developed computer software is a fully object-oriented application on all the three stages: preprocessor, solver and postprocessor. It was written using C++ and describes the process of rolling steel band rolling in three-dimensional space.

Computer programs that use the finite element method consists of three basic parts:

- preprocessor, which is built to solve the task,
- processor, which is part of the computing,
- postprocessor, used for graphical presentation of the results.

For users of such software, the most labour-intensive and time-consuming step in solving the problem is to determine the distribution of finite elements in the preprocessor. It should be noted that the wrong division into finite elements leads to erroneous results. The solution of the equations developed in the final stage of the preprocessor activity is included in the processor simulation application. To solve the equation system, the following methods for solving systems of equations (Gauss-Jordan method, CG, MinRES) have been used. The postprocessor allows for three-dimensional visualization of the calculation results. The OpenGL library was used for this purpose. At this stage of the application operation, the results can be presented both in three dimensional space and in any cross-deformable band.

The described graphics library usage helped to facilitate handling of the application because all the modifications for the geometry of the slab (rollers) as well as FEM mesh parameters are visible on the fly. This allows the user building the project to easily simulate the rolling process and decide the grid density and adjust it as necessary. In addition, the OpenGL library was used for visualization of the results, which undoubtedly caused:

- accelerating the analysis results,
- increased transparency in the presentation of the calculation results,
- the ability to view results in 3D space,
- the ability to plot any analyzed parameter in any cross-sections.

4.1. Numerical model

For solving common problems occurring in mechanical triaxial state of strain in the range of hot rolling rigid-plastic model of continuous medium [4] is often used. This approach is also valid for the considered model, since the elasticity of the material in a semi-liquid zone is small. The basis of the solution is usually to obtain the minimum value of the power supplied to the process, which is related to plastic deformation.

The solution to the thermal model involves the determination of the temperature distribution in the created FEM mesh nodes. The built thermal model is based on matrices and vectors created from forming a system of linear equations. The system so created in the form of a matrix was then solved using the CG (Conjugate Gradient) method.

In the case of the mechanical model after creating a proper system of equations and transforming it into a matrix form, the following non-linear equations system is solved using the MinRES (Minimal Residual) method.

The density variable model, based on the solution of Navier-Stokes, is complex, and its solution is associated with an additional increase in the computational complexity of the model. An alternative way is an empirical model of steel density variable model. Density is dependent on: temperature, chemical composition and structure of the steel. This model was used in the discussed solution.

4.2. Authors' original way to storing the matrices

The simulation process requires full square matrices. However, the use of such matrices for the calculation is not effective. In addition, it can be concluded that all the created matrices are symmetric. Therefore, the matrices of a certain bandwidth were used. The bandwidth is determined based on the number of variables for a given grid node and the maximum difference in the nodes' indexes in one element [4]. In this solution, matrices of non-zero elements were used by eliminating zero elements from the matrix band. In the present solution, we have used our own solution and data are stored in reduced matrices. Figure 7 illustrates a full matrix: the red colour indicates the diagonal elements of the matrix (elements with number: 1,6,8,13,14,18,20,22,24,25), while the blue colour – its non-zero elements (the other numbered fields). This matrix is a symmetric matrix. The matrix containing only non-zero elements of the full matrix was presented in Figure 8.

For the purpose of implementing such a method of storing information, our team have created two special classes: RMatrix - tackling the matrix, and Ritem - representing the element of the matrix. The essence of the RMatrix class is 1-dimensional array of size equal to the number of degrees of freedom of the modelled phenomenon of the Ritem type list of non-zero

elements in the row. A single element of the array represents a list of non-zero elements of the row. The RMatrix class has been created in such a way that if in any calculation a given element is zeroed, it is automatically removed from the structure. Such a structure of storing the data from the array is an organizing structure that stores only non-zero elements. For the purposes of FEM solvers and applied systems of equations, the class has been supplemented with mechanisms of arithmetic operations on matrices and vectors, and a special method of reducing the matrix in the case of the Dirichlet boundary condition. Thanks to arrays built in such a way, they include a smaller number of elements, thus matrix multiplication is much faster than in the case of the full and the band matrices. As a result, the need for working memory is reduced [3].

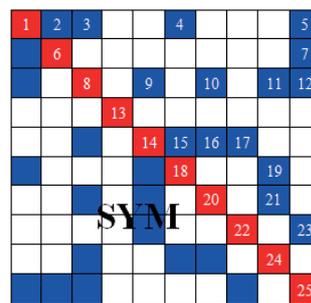


Fig. 7 View of a full matrix

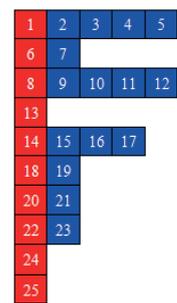


Fig. 8 View of a compressed matrix

4.3. Pre-&Postprocessor

The program is built as a “Wizard”, i.e., step by step, the user can move through the stages of the project. To go to the tab of the application that allows to perform calculations, you must sequentially proceed through the tabs “Options”, “Design and design options”, “Material parameters”, “Band/roller parameters” and “FEM grid parameters”. The final stage of the simulation process is the analysis of results, which is implemented through the “Calculation Results” tab [3].

The application includes a number of options in the “Options” tab. The options are grouped into sections and include: program options, language choice, creating shortcuts, setting the priority of the application, information about RAM memory and version control.

The first part of the creation of the rolling simulation process project is to build a computational project or to select an existing one.

A very important issue for modeling the rolling process is knowledge of the properties of deformed materials. This is especially important for high temperature processes. The process that runs in extra high temperatures is the subject of this work. Therefore, the created application not only has the possibility to read the relevant curves describing the characteristics of the material, but also allows to visualize the input curves. The parameters which characterize the rolled material are thermal properties (thermal conductivity, isotropy, the heat transfer coefficient), the density curve and the stress – strain curve.

The next step in building a project capable of carrying out the process of rolling simulation is to determine the geometrical

parameters of the band (initial length, initial width, initial height, final height) and rolls (radius, width). This module allows to visualize the input parameters. After changing any of the parameter, components of the rolling process are drawn on the tab. The band is drawn in three colours, due to the existence of three characteristic zones for the deformable band: the area before the deformation zone (in front of the rollers), the deformable zone (the rollers) and the deformed zone (after the roller). Additionally, the user can move and rotate the created figure created by using a pointer with special keys.

After entering the information about the geometry of the roll band and the roller, digitizing of the areas should be carried out. Establishing a grid of finite elements is done through the use of the "FEM grid" tab. All the built components are cubic elements in the current version. The division of the band into elements is fixed in all directions of the coordinate system (width, height, length of the band) independently for each of the zones of the band, while for the roller the number of elements varies along the length, radius and circumference. The parameters of division into elements are grouped separately for bandwidth and roll. All changes in the number of nodes for both the band and the roll are reflected in the number of elements, and the calculations are made in real time. In addition, each change of any parameter has an impact on the drawing of the tab network elements and roll band generated in the main part of the tab. Similar to geometrical parameters, the created grid of elements may be subjected to any geometric transformations.

The next stage of building the project is the "Start Calculations" tab. At this stage, all the information required to correctly simulate the rolling process is already prepared. This tab allows the user to save the created project and run calculations. In the main window there is an area of visualization of the results of the current calculation step. You can also choose a parallel or isometric view of the obtained results. In the case of parallel projection, beside the ability to modify the view settings bands in the display area, it is also possible to analyze the results on an outside wall using the pointer (mouse). Users can choose between the following choices of exterior walls: the wall along the length of the band (a vertical plane parallel to the length of the band), the top wall (a horizontal plane parallel to the length of the band), the front wall (a vertical plane perpendicular to the axis of the band, positioned in front of the entrance to the roll gap), the output wall (a vertical plane perpendicular to the axis of the strand after leaving the roll gap).

The last step in the process of rolling simulation is postprocessor, or the tab that allows the visualization of the obtained results of calculations (Fig. 9). In the „Calculations” tab, the main part of the window is the area where the calculation results are displayed as colour maps placed on the walls, and cross-sections of the band. In contrast, the right side of the tab is intended for visualization options and displaying performance results. The options to analyze the results of the calculation start in the „Presentation” tab, which has fields such as: the ability to display individual objects, shading mode, the method of projection and parameters describe the figure. After going through the „Calculation results” tab, the band with superimposed maps of calculated size distributions (Fig. 9) automatically displays. However, you can enable visualization of rolls involved in the rolling process by using

the “type of visualization” and disable network representation of shading and grid view (Fig. 10). In addition, the standard mode is shading, described as „Rainbow”. The paper presents several patterns of shading of the received distributions. Thanks to the authors of papers [5, 6] it is possible to use the following types of shaders: greyscale, rainbow, blue, red, neon, red - blue, inverse greyscale, rainbow extender, weather forecast, extremal, layers, solar spectrum. Shading is performed correctly within the calculated values. However, you can set restrictions on the displayed values. Another field located at the “Presentation” tab is an area dedicated to the projection. First, the user can decide on the type of projection. There are two possibilities: isometric projection and parallel one. The transition between these different kinds of projection is realized by a smooth animation. In the case of using parallel projection it is possible to use ready observation points, as in the case of the “Start calculation” tab. In addition, it is possible to visualize the rolling parameters not only for the full slab, but also for $\frac{1}{4}$, $\frac{1}{2}$ and $\frac{3}{4}$ of the area. Same place options are available for the analysis of next steps in the calculation of the “Run the calculation” tab. The last field in the “Presentation” tab is “Description of the picture”. This field allows the user to affect the ability to display such objects as coordinate system, legend, etc.. The user can change the view of the presentation of the band, to zoom in and out objects, rotate in any direction and move up - down, left - right). What is more, it is possible to rotate around the axis without changing the current settings.

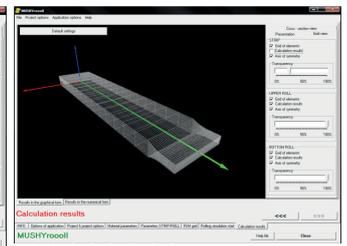
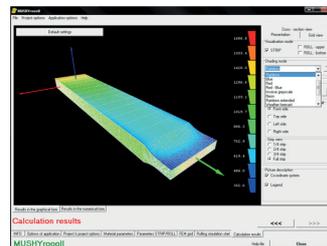


Fig. 9. Default view of the calculation results

In addition to the standard display of calculation results, it is also possible to analyze the results in the default planes carried along all the directions of the band: height (Fig. 11), width (Fig. 12) and length (Fig. 13). These planes can be moved along the said directions starting from the middle slab (Fig. 12) (Fig. 13). Displaying of the plane happens in real time, due to the original way to remember only the necessary information needed to draw the analyzed size distribution in the plane.

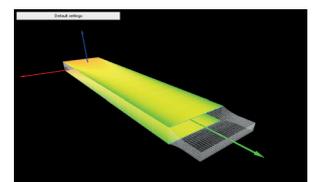
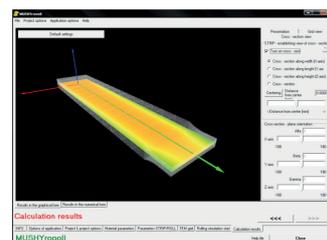


Fig. 11. Cross-section view - along height (Z-axis)

Fig. 12. Cross-section view - along width (two planes)

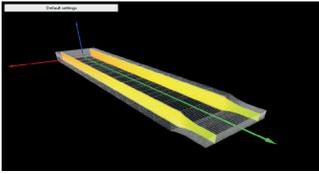


Fig. 13. Cross-section view - along length (Y-axis)

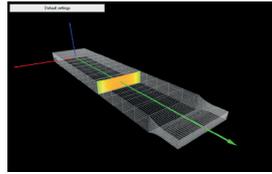


Fig. 14. Cross-section view - along height (Z-axis)

In the case that the user-defined area could not be analyzed in one of the default cross-sectional planes, it is possible to use any plane. At the beginning, the user selects an interesting cross-section plane (Fig. 15) and then it is possible to display the distribution of the analyzed parameter in a designated plane (Fig. 16).

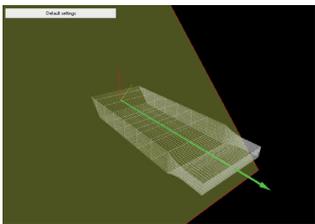


Fig. 15. Cross-section defined by user

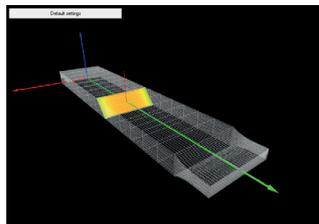


Fig. 16. Distribution analyzed parameter in a defined plane

5. Conclusions

In the context of the work, the ideas of the heat transfer mathematical models, deformation and changes in density have been characterized. On the basis of the theoretical models, a numerical model was developed, which was then implemented in an object-oriented technology to create an efficient computer program with an advanced graphic interface. The application built is based on the finite element method. Due to the coexistence of the liquid phase and the solid phase in the considered steel rolling process, the mechanical state prevailing within the zone of deformation is the full three dimensional state. Therefore, there was a need to build full spatial models, which, however, can easily be simplified in order to make quick preliminary calculations.

The application has a fully object-oriented structure, both in the implementation of mathematical models, as well as in the graphic user interface. For this reason, there is a possibility of its easy expansion. In addition, the classes created during the construction of the algorithm relate directly to the real

objects. In the previous applications structural approach was used, which has been widely used for the implementation of applications of models based on the finite element method.

In case of necessity for making adjustments to the application, the programming technique used frees programmers from creating a new entity from scratch, since there is a suitable adaptation of existing ones. The algorithm built in this way has many advantages such as uniformity, modularity, control based on the merits, hiding information, exception handling, static and dynamic type checking, inheritance, redefinition, or a polymorphism. The resulting program is fully operational for the calculation, but also has some disadvantages. In its present form, it does not account for all physical phenomena accompanying the process of rolling bands of semi-liquid core. Some models were consciously used in their simplified versions (e.g. an empirical model of density changes in the place of the full solution of differential equations speeds up calculations). Others, affecting the process, have not been implemented due to lack of physical theories and, consequently, the equations describing them.

Acknowledgment

The project has been supported by statutory activities, Decision number: AGH nr 11.11.110.300

REFERENCES

- [1] M. Głowacki M. Hojny, R. Kuziak, Komputerowo wspomagane badania właściwości mechanicznych stali w stanie półciekłym, Wydawnictwa AGH, Kraków 2012.
- [2] M. Hojny, M. Głowacki, Modeling of strain-stress relationship for carbon steel deformed at temperature exceeding hot rolling range, *Journal of Engineering Materials and Technology* **133**, 021008.1-021008.7 (2011).
- [3] D. Jędrzejczyk, Przestrzenny termomechaniczny model walcowania w stanie półciekłym, PhD thesis, AGH University of Science and Technology, Kraków (2008), (in Polish).
- [4] M. Pietrzyk, Metody numeryczne w przeróbce plastycznej metali, Wydawnictwa AGH, Kraków 1992.
- [5] A. Gumuła, M. Głowacki, Metody szybkiej wizualizacji wyników symulacji komputerowych, Materiały XIII konferencji "Informatyka w Technologii Metali", Szczawnica 15-18 maja 2006, 67-70.
- [6] A. Gumuła, M. Głowacki, Graficzna prezentacja rozkładu parametrów procesu przeróbki plastycznej, Zeszyt Towarzystwa Naukowego - „Laureaci VI edycji konkursu DIAMENTY, 30-36.AGH, Kraków 2006.

