Development of Dedicated Computer System for Gleeble 3800® Thermo-Mechanical Simulator

M. Hojny, M. Głowacki, R. Kuziak, and W. Zalecki

Abstract—**The paper reports the results of theoretical and experimental work leading to the construction of a FEM (Finite Element Method) oriented code allowing the computer simulation of physical phenomena accompanying the steel deformation at temperatures which are characteristic for direct rolling of continuously cast charge, as well as the graphical and database oriented pre- and post-processing modules completing the system and making it user-friendly. A coupled thermo-mechanical model including inverse analysis technique was adopted for the solver. The advantage of the solution was the analytical form of both incompressibility and mass conservation conditions. This can prevent usual FEM variational solution problems concerning unintentional specimen volume loss caused by the numerical errors. It is very important for the discussed modeling because the deformation process is running in temperature range that is characteristic for the last stage of transformation of steel aggregation state. The well known machine allowing tests in the discussed temperature range is the Gleeble® 3800 thermo-mechanical simulator. However, carrying out experiments with steel deformation in semi-solid state using this machine are very expensive. Therefore application of a dedicated computer simulation system is strictly required. Inverse analysis and appropriate modeling of the testing procedure makes tests possible, first of all, but it also results in lowering testing cost. The newest version of the Def_Semi_Solid v.5.0 is a unique FEM system supporting aided extra high temperature tests.**

Index Terms—**computer system, inverse analysis, numerical analysis, semi-solid state**

I. INTRODUCTION

Integrated rolling of plates which are products of continuous casting is a brand new and very proficient way of hot strip manufacturing. Only few companies all over the world are able to manage such processes. Among them one can mention the plant located in Cremona Italy which develops the ARVEDI Steel Technologies – new methods of steel strip manufacturing. They are called ISP and AST processes and are characterized by very high temperature allowed at the mill entry. The instant rolling of slabs which leave the casting machine allows the utilization of the heat stored in the strips during inline casting. The rolling equipment for the IST process consist of cast rolling machine, liquid core reduction equipment, high reduction mill, inductive heater, Cremona coiling station, descaler, traditional finishing mill and the cooling zone. The initial mould strip thickness is 74 mm and is reduced to 55 mm during liquid core reduction process. The region of maximum strip temperature for a high reduction mill is placed in the strip centre and varies from 1493.15 K to 1648.15 K depending on the casting speed. The main benefits of the technology are: inverse (in comparison to traditional rolling) temperature gradient in the cross-section of the strip, which is very useful for the rolling process, good product quality (1 mm strip with best shape and microstructure), very low level of heating energy consumption which drops to 0 when the casting speed is over around 0.14 m/s, up to 20 times lower water consumption, low level of installed mill power in high reduction mill (3 rolling stands with 0.5, 0.5 and 0.8 MW) providing reduction from 55 to 12.5 mm by the strip width of 1300 mm, compact rolling equipment layout (total rolling line is only 170 m long), no need of tunnel furnace, and very low investment costs. The AST technology is a result of further development of ISP to a real endless process. The main difference between these two technologies is the absence of the heating equipment in case of AST. The whole reduction process is running in one rolling mill consisting of 5 or 7 stands, which can reduce the strip thickness from 55÷70 mm to 0.8 mm. AST is the most compact hot strip production process using oscillating mould technology with excellent efficiency and cost. The total equipment length is 70 to 80 m including casting machine at the front and final coolers at the rear end. The maximal temperature of the strip occurs in central region of its cross-section and varies from 1613.15 K to 1693.15 K according to the casting speed. It suggests that the central region of the strand subjected to the rolling is still mushy. Both the technologies mentioned above ensure huge reduction of rolling forces, very high product quality and low investment costs and their details are usually classified.

The numerical modeling can be very helpful in developing “know how” theory for the mentioned processes. The lack of mathematical models describing the steel behaviour in the last phase of solidification with

---

Manuscript received November 10, 2010; revised December 14, 2010. This work has been supported by the Polish Ministry of Science and Higher Education Grant N N508410637.

M. Hojny is with the AGH-University of Science and Technology, Mickiewicza 30, 30-059 Kraków, POLAND (phone: 48-126174637; fax: 48-126172921; e-mail: mhojny@metal.agh.edu.pl).

M. Głowacki is with the AGH-University of Science and Technology, Mickiewicza 30, 30-059 Kraków, POLAND (e-mail:glowacki@agh.edu.pl).

R. Kuziak is with the Institute for Ferrous Metallurgy, K. Miarki 12-14, 44-100 Gliwice, POLAND (e-mail: rkuziak@imz.gliwice.pl).

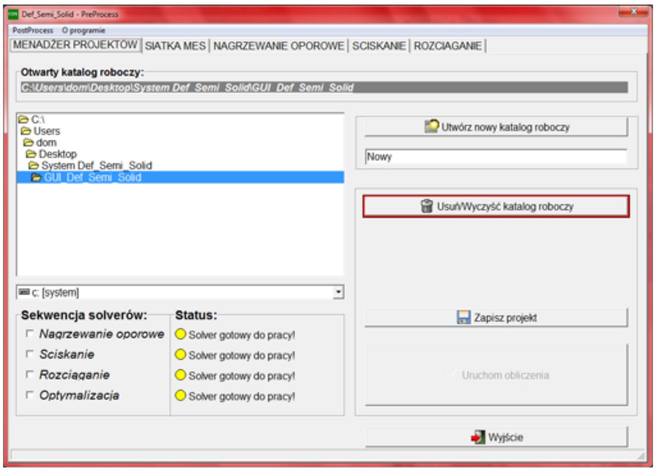
W. Zalecki is with the Institute for Ferrous Metallurgy, K. Miarki 12-14, 44-100 Gliwice, POLAND (e-mail: wzalecki@imz.gliwice.pl).

simultaneous plastic deformation was the inspiration of the investigation described in the presented paper. The authors have run further investigations leading to development of fully three-dimensional model of integrated casting and rolling processes, as well as the soft reduction process, that are part of strip casting technology. The contribution sheds light on these problems. It focuses on the axial-symmetrical computer model, which ensures the possibility of its experimental verification with the help of physical simulation conducted by using Gleeble® 3800 simulator.

II. THE COMPUTER SYSTEM

In aim to allow easy working with the Gleeble® 3800 simulator a user friendly system called Def_Semi_Solid v.5.0 was developed in the Department of Computer Science and Modeling of the Faculty of Metals Engineering and Computer Science in Industry, AGH. The numerical part of the program was developed in FORTRAN/C++ language, which guarantees fast computation and the graphical interface was written using visual version of C++ language, taking advantage of its object oriented character. This approach has sufficient usability both in Windows and Linux based systems. The newest version of Def_Semi_Solid system is equipped with full automatic installation unit (Fig. 1) and new graphical interface. It allows the computer aided testing of mechanical properties of steels at very high temperature using Gleeble® 3800 physical simulators to avoid problems which arise by traditional testing procedures. The first module allows the establishment of new projects or working with previously existing ones (Fig. 2). The integral parts of each project are: input data for a specific compression/tension test as well as the results of measurements and optimization. In the current version of the program the module permits application of a number of database engines (among other standard MSAcces, dBASE IV and Paradox 7-8 for PC-based systems) and allows the implementation of material databases and procedures of automatic data verification. The next module (the solver) gives user the possibility of managing the working conditions of the simulation process. The inverse analysis can be turned off or on using this part of the system.

Fig. 1. The main window of the newest version of Def_Semi_Solid system.



Otwarty katalog roboczy: C:\Users\dom\Desktop\System Def_Semi_Solid\GUI_Def_Semi_Solid

Nowy

c: [system]

Sekwencja solwerów:	Status:
☐ Nagrzewanie oporowe	○ Solver gotowy do pracy!
☐ Sciskanie	○ Solver gotowy do pracy!
☐ Rozciaganie	○ Solver gotowy do pracy!
☐ Optymalizacja	○ Solver gotowy do pracy!

Fig. 2. The Pre-processor of the newest version of Def_Semi_Solid system.

The last module (DSS/Post module) is dedicated to the visualisation of the numerical results and printing the final reports. In the current version the possibility of visualization was significant improved. The main are: shading options using OpenGL mode (2D and 3D) as shown in Fig. 3 and possibility make a full contour map (2D and 3D) as shown in Fig. 4.

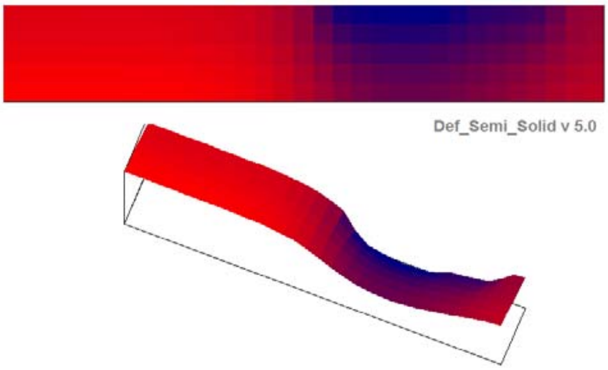


Fig. 3. The Post-processor of the newest version of Def_Semi_Solid system (shading option 2D and 3D).

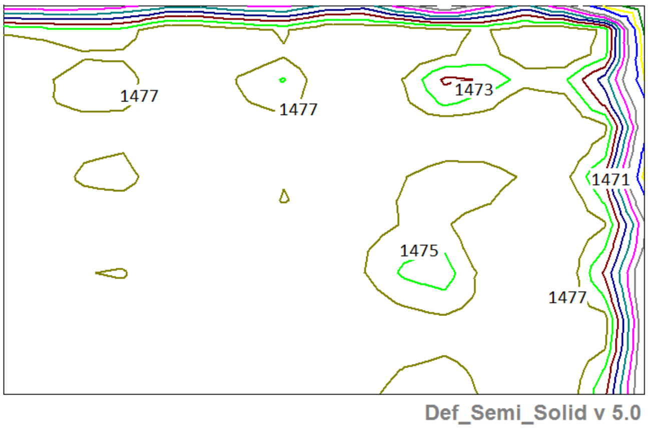


Fig.4. The Post-processor of the newest version of Def_Semi_Solid system (contour option).

The less visible but powerful heart of the system is of course the solver. The finite element code dedicated to the axial-symmetrical compression/tension tests has been developed. The solution is based on the thermo-mechanical approach with density changes described in [2]. Most of the rigid-plastic FEM systems apply a variational approach, which allow the calculation of strain

ISBN: 978-988-19251-2-1
ISSN: 2078-0958 (Print); ISSN: 2078-0966 (Online)

field and deviatoric part of stress tensor distribution according to subsequent functional:

$$J^*[v(r,z)] = W_\sigma + W_\lambda + W_t \tag{1}$$

where W_σ is the plastic deformation power, W_λ the penalty for the departure from the incompressibility or mass conservation conditions and W_t the friction power. The main idea of the presented solution is the lack of the second part of functional (1). Both the incompressibility and mass conservation conditions are given in an analytical form and constrain the velocity field components. The functional takes the following shape:

$$J^*[v(r,z)] = W_\sigma + W_t \tag{2}$$

In (1) and (2) v describes the velocity field distribution function in the deformation zone. The optimisation of functional (2) is much more effective than the functional (1) because numerical form of incompressibility condition generates a lot of local minimums and leads to wide flat neighbourhood of the global optimum. The accuracy of the proposed solution is much better because of negligible volume loss. This is important for materials with changing density. In classical solutions the numerical errors which are caused by volume loss can be comparable to those coming from real density changes. All that leads to solution with low accuracy. The model with analytical incompressibility condition is free from the described shortcoming. For solid regions of the sample the incompressibility condition has been described in cylindrical coordinate system with an equation:

$$\frac{\partial v_r}{\partial r} + \frac{v_r}{r} + \frac{\partial v_z}{\partial z} = 0 \tag{3}$$

where v_r and v_z are the velocity field components in cylindrical coordinate system r, θ, z. For the mushy zone equation (3) must be replaced by the mass conservation condition, which takes a form:

$$\frac{\partial v_r}{\partial r} + \frac{v_r}{r} + \frac{\partial v_z}{\partial z} - \frac{1}{\rho}\frac{\partial \rho}{\partial \tau} = 0 \tag{4}$$

where ρ is the temporary material density and τ - the time variable. Both the strain and stress models are based on Levy-Misses flow criterion. Condition (4), which is more general than relationship (3), was used for the purpose of the model. The model is completed with numerical solution of Navier stress equilibrium equations.

The temperature field is a solution of Fourier-Kirchhoff equation with convection. The most general form of this equation can be written as:

$$\nabla^T (k\nabla)T + \left[Q - c\rho\left(\frac{\partial T}{\partial \tau} + v \circ \nabla T \right) \right] = 0 \tag{5}$$

where T is the temperature distribution inside the controlled volume, k denotes the isotropic heat conduction coefficient, Q represents the rate of heat generation due to the plastic work done and c describes the specific heat. The solution of equation (6) has to satisfy the boundary conditions. The combined Hankel's boundary conditions have been adopted for the presented model:

$$k\frac{\partial T}{\partial n} + \alpha(T - T_0) + q = 0 \tag{6}$$

In equation (6) T_0 is the distribution of the border temperature, q describes the heat flux through the boundary of the deformation zone, α is the heat transfer coefficient and n is the normal to the boundary surface.
During the testing procedure the sample is melted down as a result of its resistance heating. The heat generated due to direct current flow was calculated using the Joule-Lenz law according to following equation:

$$Q = I^2 R \tau \tag{7}$$

where I is the current intensity, R is the electrical resistance and τ is the time.
The resistance was predicted using the following formula:

$$R = \rho_w \frac{l}{A} \tag{8}$$

In equation (8) *l*, *A* and ρ_w are the sample: length, area of the cross-section and specific resistance, respectively. The temperature changes have influence on the specific resistance. In the presented solutions the empirical equation was used to predict the specific resistance at a desire temperature:

$$\rho_w = \rho_0 \left[1 + \overline{\alpha}(t - t_0) \right] \tag{9}$$

In equation (9) ρ_0 is the specific resistance at the temperature $t_0 = 293.15K$, t is the current temperature and dashed α is a coefficient. One of the most important parameters of the solution is also the density. Its changes have influence on the mechanical part of the presented model and strongly depend on the temperature. The knowledge of effective density distribution is very important for modeling the deformation of porous materials. Description of the density changes has been presented in details in [2]. More details concerning the presented mathematical model were published in [1]-[5].

III. EXPERIMENTAL VERIFICATION OF THE MODEL

The experiments were done in Institute for Ferrous Metallurgy in Gliwice, Poland for low carbon steel using Gleeble® 3800 simulator (Fig. 5).

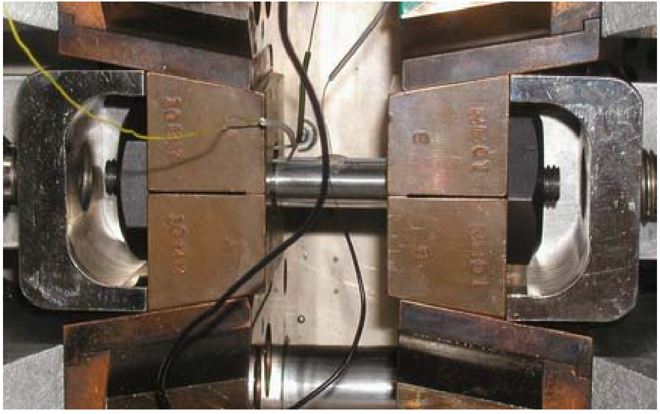


Fig.5. The standard Gleeble® equipment allowing deformation is semi-solid state.

In all cases, experiments ran according to schedule:
stage 1: the sample was prepared (e.g. mounting thermocouples, die selection),
stage 2: melting procedure of the sample was realized,
stage 3: at the end the deformation process was done.
Fig. 6 shows the shape of the testing samples and locations of thermocouples used during experiments.

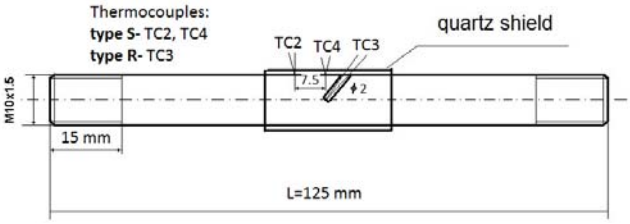


Fig.6. Samples used for the experiments. TC2, TC3 and TC4 thermocouples.

Material tests in the semi-solid state should be carried out in as isothermal conditions as possible due to the very high sensitivity of material rheology on small changes of temperature. This is why temperature distribution inside the tested samples should be analyzed. The basic reason for uneven temperature distribution inside samples on the Gleeble® simulator is their contact with cooper handles during resistance heating (Fig. 7 and Fig. 8).

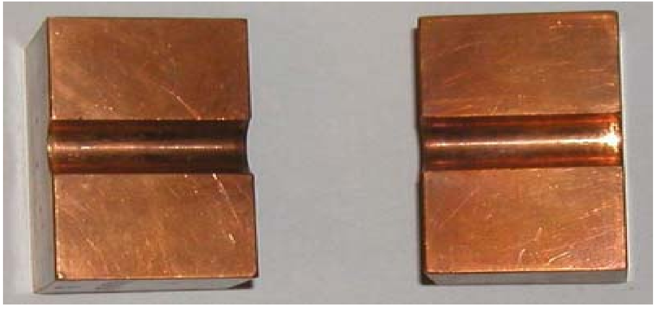


Fig.7. The handle with long contact zone.

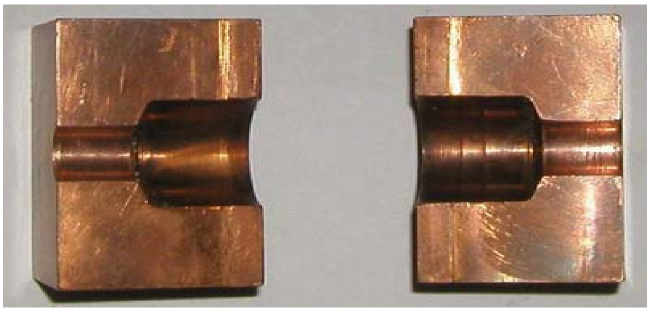


Fig.8. The handle with short contact zone.

The liquidus and solidus temperature of the investigated steel are 1763.15 K and 1683.15 K, respectively. The resistance heating processes cause non-uniform distribution of temperature inside heated materials especially in longitudinal section of the sample. In the case of the semi-solid steels, such distribution gives significant differences in the microstructure and rheology. The thermo-physical properties of this steel, necessary in calculations, were determined using JMatPro software. This software determines this properties on the basis of the chemical composition. In case of each physical and computer simulation samples were heated to 1703.15 K and after holding at constant temperature the sample was cooled to nominal deformation temperature. In the final stage of physical simulation for different holding time, the temperature difference between core (TC3 thermocouple) of the sample and the surface (TC4 thermocouple) was analyzed. In all cases the temperature difference between core of the sample and surface was about 303.15 K for variant with cold handles (Fig. 9) and 313.15 K for variant with hot handles (Fig. 10). The numerical simulation confirmed results obtained during experimental parts. In the Fig. 11 temperature distributions in the cross section of the sample tested at temperature 1658 K are presented for 3 and 5 seconds of heating and final distribution right before deformation. The one can observe, major temperature gradient between contact surface die-sample. The difference between experimental and theoretical core temperatures for hot handles was 277.15 K. The similar situation was observed in case of using cold handles – 280.15 K. The comparison between experimental results and numerical show that mathematical model of resistance heating right reflect back the physical simulation of resistance heating of samples using Gleeble® 3800 physical simulator.

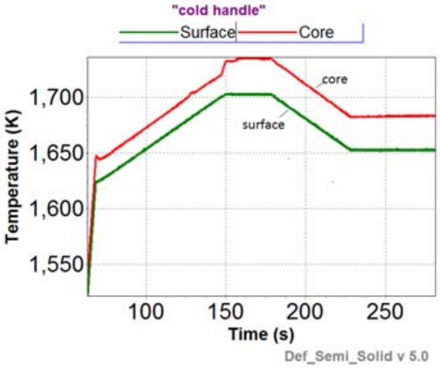


Fig.9. The temperature change versus time for cold handle (final stage of physical simulation right before deformation).

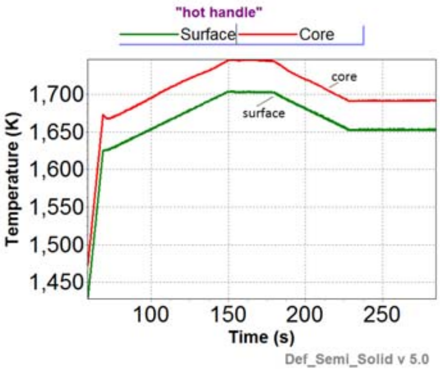


Fig.10. The temperature change versus time for hot handle (final stage of physical simulation right before deformation).

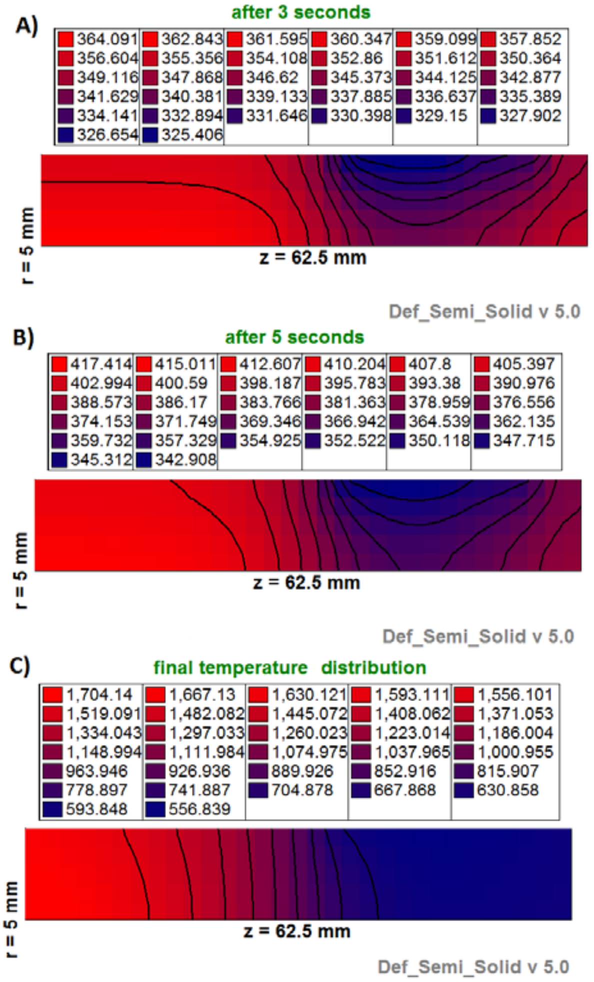


Fig.11. Distribution of temperature in the cross section of the sample tested at temperature 1658.15 K after time heating a) 3 seconds, b) 5 seconds, c) right before deformation. **Tools: hot handle.**

Finally, the microstructure of the tested samples was investigated. Fig. 12-13 show example microstructure in the cross-sections of two samples right before deformation deformed at 1658.15 K. One can observe that for analyzed temperature liquid phase particle exist in the central part of the sample.

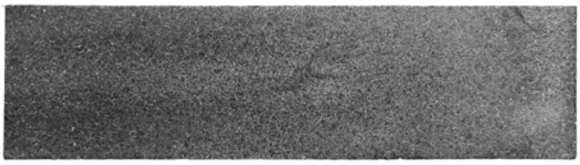


Fig.12. Sample pickled in Oberhoffer reagent – right before deformation at 1658.15 K . **Tools: "hot handle".**

Fig.13. Sample pickled in Oberhoffer reagent – right before deformation at 1658.15 K . **Tools: "cold handle".**

During experiments die displacement, force and temperature changes in the heating zone are recorded. Parallel, the computer simulations were realized in order to obtain optimal value parameters of process. The strain-stress curves were described by following equation:

$$\sigma_p = A\varepsilon^n \exp(B\varepsilon)\dot{\varepsilon}^m \exp(-CT) \qquad (10)$$

in the temperature range above 1473.15 K, where A, B, C, n, m are material constant, T - temperature, ε - strain and $\dot{\varepsilon}$ - strain rate. It is not easy to construct isothermal experiments for temperatures higher than 1473.15 K. Several serious experimental problems arise. First of all, keeping so high temperature constant during the whole experimental procedure is extremely difficult. There are also severe difficulties concerning interpretation of the measurement results. The significant inhomogeneity in the strain distribution in the deformation zone and distortion of the central part of the sample lead to poor accuracy of the stress field calculated using traditional methods, which are good for lower temperatures. The only possibility to have good coefficients of strain-stress formula is the inverse method. The long calculation time, which is usual by this kind of analysis requires sometimes parallel computation. The application such a method is considered for the future application.

The objective function was defined as a norm of discrepancies between calculated (F^c) and measured (F^m) loads in a number of subsequent stages of the compression according to the following equation:

$$\varphi(x) = \sum_{i=1}^{n}\left[F_i^c - F_i^m\right]^2 \qquad (11)$$

The theoretical forces F^c were calculated with the help of sophisticated FEM solver facilitating accurate computation of strain, stress and temperature fields for materials with variable density. The comparison between the calculated and measured loads is presented in Fig. 14, showing good agreement between both loads. The coefficients obtained during inverse analysis allow the construction of stress-strain curves, which are presented in Fig. 15.

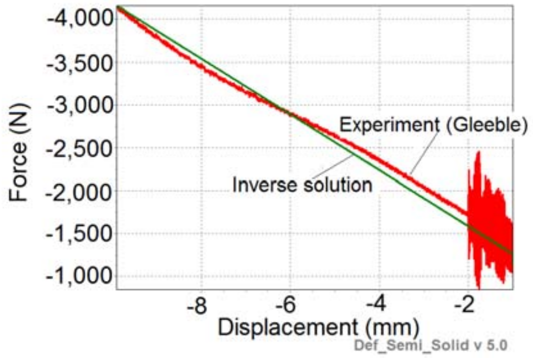


Fig.14. Comparison between measured and predicted loads at temperature 1623.15 K (quasi-static process).

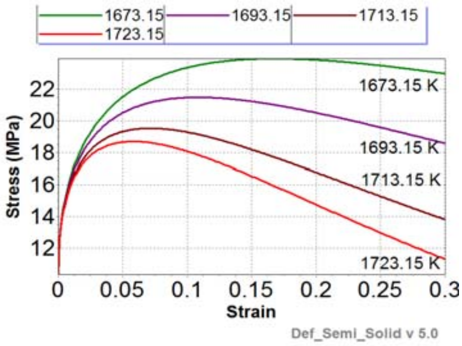


Fig.15. Stress-strain curves at several temperature levels from the range of 1673.15-1723.15 K (quasi-static process).

Using previously developed curves, example simulations of compression of cylindrical samples with mushy zone have been performed. The results of the tests demonstrate the possibilities of the computer system. For all series of tests the simulations were done using short contact zone between the sample and simulator jaws. The deformation zone had the initial height of 67 mm. The diameter of the sample was 10 mm. An example specimen was melted at 1703.15 K and then after cooling temperature deformed at demanding temperature. During the tests each sample was subjected to 10 mm reduction of height. For verification of the computer system two comparative criteria were used:

- comparison between the measured and calculated lengths of zone which was not subjected to the deformation,
- comparison of the maximum measured and calculated diameters of the sample. Analysis of sample deformation at 1673.15 K has shown that the error of the experimental result was 1.5 mm for the first criterion and 1.7 mm for the second, (see Fig. 16 and 17).

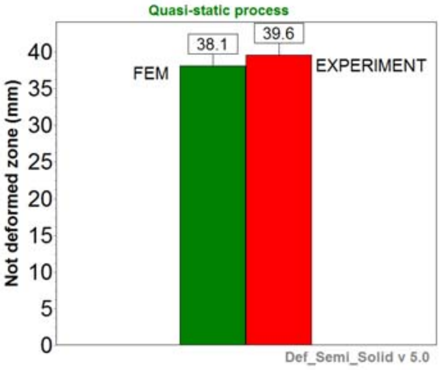


Fig.16. The comparison between the measured and calculated lengths of zone which was not subjected to the deformation – experiments at temperature of 1673.15 K for the quasi – static process.

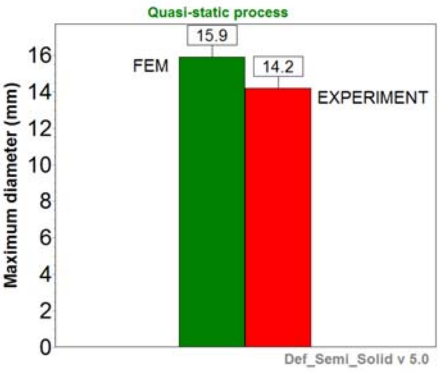


Fig.17. The comparison of the maximum measured and calculated diameters of the sample – experiments at temperature of 1673.15 K for the quasi – static process.

IV. CONCLUSION

Modeling and simulations of deformation of steel samples at extra high temperatures requires resolving a number of problems for the discussed temperature range: the difficulties in calculation of material constants, the necessity of determination of characteristic temperatures, avoiding the volume loss due to numerical form of incompressibility condition, which causes problems concerning optimization of the velocity field.

The presented dedicated computer system can be very helpful and may enable the right interpretation of results of very high temperature tests. It has shown good predictive ability of the computation regarding both shape and size of the deformation zone, mechanical properties of steels and temperature distribution in the cross section of the sample. It is possible with the help of developed CAE system thanks to computer aided testing, application of right model and implementation of the inverse analysis.

REFERENCES

[1] M. Hojny, M. Głowacki and Z. Malinowski, "Computer Aided Methodology of Strain-Stress Curve Construction for Steels Deformed at Extra High Temperature," *High Temperature Materials and Processes*, vol. 28, no. 4, pp. 245–252, Dec. 2009.

[2] M. Hojny and M. Głowacki, "The Physical and Computer Modelling of Plastic Deformation of Low Carbon Steel in Semi-Solid State," *Journal of Engineering Materials and Technology*, Trans. of the ASME, vol. 131, no. 4, pp. 041003-1–041003-7, Apr. 2009.

[3] M. Hojny and M. Głowacki, "Computer Modeling of Deformation of Steel Samples with Mushy Zone," *Steel Research Int*., vol. 79, no. 11, pp. 868-874, Nov. 2008.

[4] M. Hojny and M. Głowacki, "The Methodology of Strain – Stress Curves Determination for Steel in Semi-Solid State," *Archives of Metallurgy and Materials*, vol. 54, no. 2, pp. 475–483, Feb. 2009.

[5] M. Głowacki and M. Hojny, "Inverse Analysis Applied for Determination of Strain-Stress Curves for Steel Deformed in Semi-Solid State," *Inverse Problems in Science and Engineering*, vol. 17, no. 2, pp.159–174, Feb. 2009.