*Solid State Phenomena Vol. 240 (2016) pp 74-80*
*© (2016) Trans Tech Publications, Switzerland*
*doi:10.4028/www.scientific.net/SSP.240.74*

*Submitted: 2014-11-14*
*Revised: 2015-04-17*
*Accepted: 2015-05-06*

# Method of determining the strain hardening of carburized elements in Ansys environment

Sebastian Lipa [1,a*], Jacek Sawicki [1,b], Emilia Wołowiec Korecka [1,c], Konrad Dybowski [1,d], Piotr Kula [1,e]

[1] Institute of Materials Science and Engineering,
Lodz University of Technology, Stefanowskiego 1/15 Lodz 90-924, Poland

[a]sebastian.lipa@p.lodz.pl, [b]jacek.sawicki@p.lodz.pl, [c]emilia.wolowiec.korecka@p.lodz.pl,
[d]konrad.dybowski@p.lodz.pl, [e]piotr.kula@p.lodz.pl

* please mark the corresponding author with an asterisk

**Keywords:** strain hardening, computational algorithm, thermochemical treatment, FEM analysis.

**Abstract.** The paper presents a method based on FEM analysis of determining the strain hardening of elements after the thermochemical treatment. A computational algorithm, which takes into account two factors, has been suggested. Firstly, the gradient of material properties resulting from the changes in the carbon content in the surface layer after the carburizing process. Secondly, the phase transformations occurring during the hardening of the material. The proposed flowchart did not include the flow effect, which greatly reduced the computation time by eliminating tedious CFD calculations. Thanks to an original program Converter coupled with a commercial software's SimHard, SimCarb J Math Pro and Ansys the carburizing process followed by quenching in gases was modeled. In parallel to the numerical calculations the results were verified experimentally. The results of FEM analysis for the tube made of stainless EN 16CrMn5 subjected to three processes of thermochemical treatment have also been presented in the paper. The experimental results were compared with the numerical simulations and a satisfactory convergence has been achieved.

## Introduction

The present development of commercial programs such as Hearts [1], Transt [2], Sysweld [3], Deform-HT [4], Dante [5], largely aims at creating a universal auxiliary tool, supporting the preparatory and final stage of thermochemical treatment. Its advantage is the ability to determine the initial treatment parameters before the processes are conducted in a furnace as well as to design treatment devices with the appropriate, required parameters.

Such activities largely involve performing a numerical simulation in which the process outcome is predicted and which enables better control of the whole thermochemical treatment process.

Majority of commercially available programs assume that the process to be designed consists of two basic stages. The first one involves formation of the top layer on the charge surface through diffusion. The other is a process of quenching, which brings about significant changes in the material properties and in the geometry of a detail under treatment. Usually, the first stage of the simulation is carried out in an original program developed specially for the analysis. It is used to define the phenomena which take place during the treatment. By purposefully entering known laws of physics, e.g. diffusion, phase transformations and their kinematics, TTT and CCT curves, one obtains data for further analysis. The second phase of simulation frequently employs a postprocessor of commercial numerical programs which is used to calculate the distribution and gradient of temperature for a specific part in combination with data from the first stage. This provides input for analyses, which usually concern deformations, as well as maps of stress and deformations. An example of such an application as the one described above is provided by coupling of two programs DANTE+ABAQUS, described in the paper by S.V. Warke and co-authors [6]. The geometry

entered initially into the numerical programme ABAQUS is discretized into finite elements. Subsequently, the external program DANTE is used to determine thermal elements, i.e. temperature distribution and gradient depending on the structure composition and phase transformations. The temporal step is also taken into consideration at this stage, associated with the cooling-down curve during the quenching process. A thermal analysis is followed in the next sequence of calculations by an analysis of stresses and deformations, which is affected by the phase density and the experimental constants determined in the DANTE program. The final analysis of the results of maps of stress and deformations is performed in the numerical program environment. Another example of this kind of approach is a computational system described in a paper by B.L. Ferguson and co-authors [7]. The DANTE /ABAQUS program was used with the support of the original ELTA program. All this served the purpose of developing a tool for simulation of inductive quenching. The ELTA software was used to compute the distribution of power and temperature during the inductive quenching process. Which in turn allowed to accumulate input data necessary to conduct the analysis in the DANTE / ABAQUS program.

Constant development of simulation programs increasingly often helps a user to formulate a precise definition of a phenomenon. Currently, a number of commercial programs and programs originally developed by researchers, which are later incorporated in them, can link the processes of conventional, vacuum, plasma carburizing with quenching in gas, oil, spray, or induction quenching. Moreover, program algorithms often help to take into account such factors as the effect of steel creep during heating and quenching, non-uniform carbon diffusion on the surface of parts under treatment (rollers, sleeves, cogwheels) or determine the place of a fatigue crack when a part after thermochemical treatment is in operation [8-10].

## Problem description

The authors of this paper attempted to develop a computational procedure which aims to model the phenomena which take place in the surface formed as a result of thermochemical treatment in gases. To this end, the authors developed a computational algorithm which combines several commercial programs by means of the original Converter program developed by them (Fig. 1) [11]. An analysis was performed of the treatment of EN 16CrMn5 steel which involved vacuum carburizing with gas quenching. Three carbon profiles (ECD = 0.8; 1.0; 1.2 %C) in the surface layer were taken into account in the calculations as well as changes of material properties at the assumed speed of the material cooling down process.

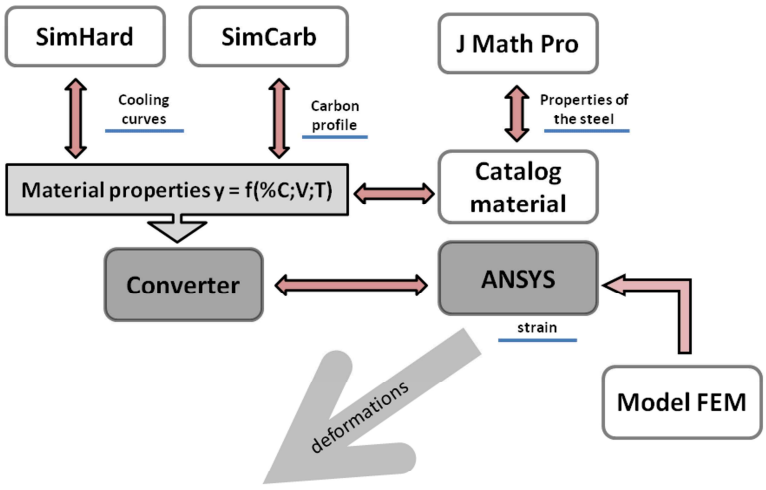


Fig. 1 Diagram of the algorithm [11]

**Description of the Converter program developed by the authors**

The Converter program was used to prepare input data to calculate quenching displacements and deformations of carburized steel parts, taking into account the gradient structure of the material (Fig. 2). In particular, the results of the program operation were adapted for work with the Ansys numerical program.

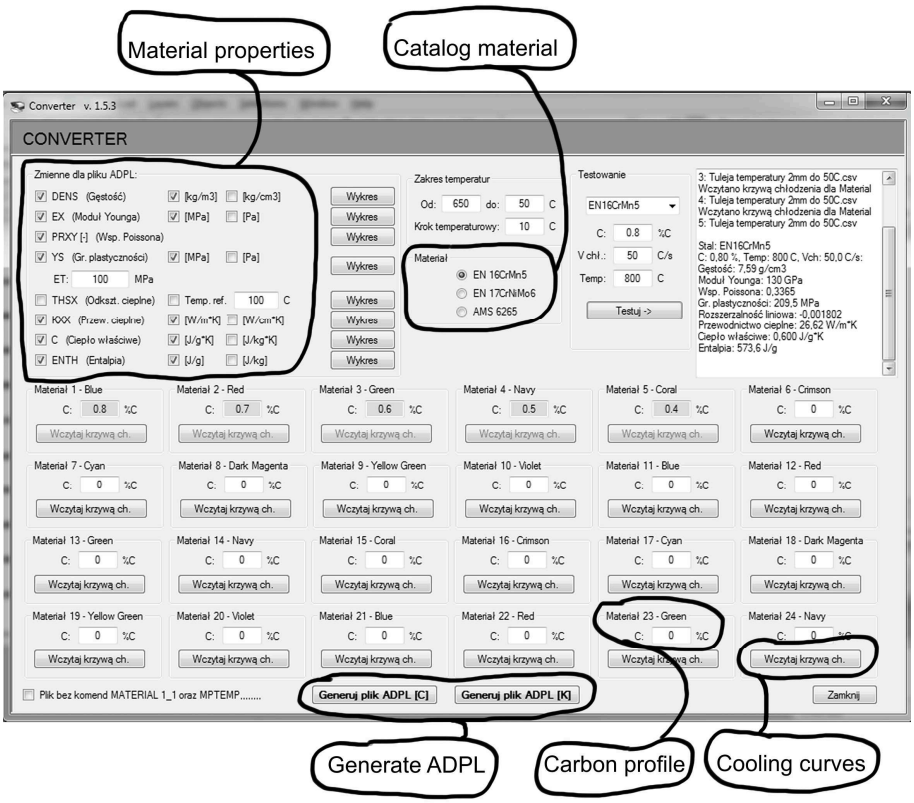


Fig. 2 Converter program

The program was based on material issue lists for steel, which describe the properties of the material, its density, Young's modulus, Poisson's ratio, the yield point, thermal deformation, thermal conductivity, specific heat and enthalpy. Their values depended on carbon concentration at a specific point of the material, temperature and cooling-down speed (Fig. 3).

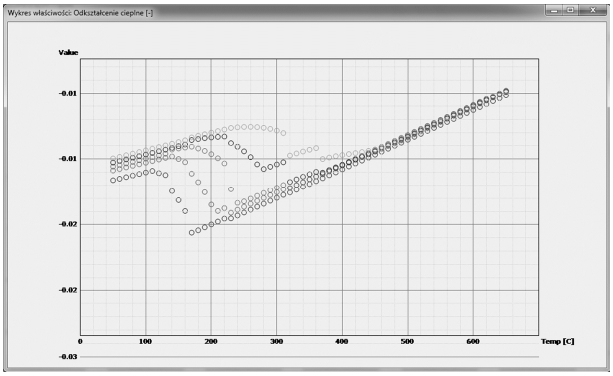

Fig. 3 Thermal deformation depending on carbon content and the cooling-down curve during the process of quenching the carburized layer

The gradient model was developed in four steps. In the first step material issue lists with the properties mentioned above were entered. Subsequently, the number of sub-layers of the gradient layer was determined (a layer could be divided into no more than 24 sub-layers) and the average

concentration of carbon was defined for each of them. The distribution of carbon concentration in the top layer was obtained from a carburization simulator SimVaCPlus® which is a module of a simulation program SimVaC Plus® ( Fig. 4). In the third step, a curve of temperature vs time (cool-down curve) is assigned to each sub-layer. The data were obtained from a quenching simulator SimHard® of the program SimVaCPlus®.

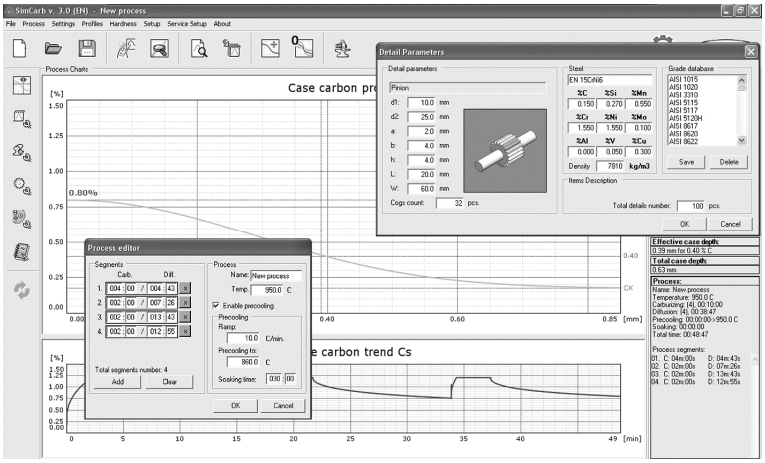


Fig. 4 A layer model generated from SimCarb® program.

In the last step, the values of all the properties mentioned above were determined for each sub-layer within the user-specified temperature range (by default it was the range between 50 and 850$^{o}$C, with a 10$^{o}$C increment). For each temperature, the Converter determined a momentary cooling-down speed and then it found the corresponding carbon concentration in the material issue lists and read out the value of the material property it looked for. The procedure was reiterated in order to define consecutive properties (i.e. density, Young's modulus, Poisson's ratio, yield point, thermal deformation, thermal conductivity, specific heat, enthalpy). This meant that for an example layer of 10 sub-layers the program found and read approximately 4800 different values from several different material issue sheets. This resulted in a model of a gradient material, characterised by eight properties, whose values took into account the state of carburization and the method of cooling down an element.

The program operation resulted in a file containing material data of the gradient model, used in the Ansys program to calculate quenching displacements and deformations.

## FEM Model

The material gradient model from the Converter program was entered into the ANSYS static structural program. A discrete model of a sleeve was created in the numerical environment discussed (inner diameter $\phi_w$=50 [mm], outer diameter $\phi_z$=67 [mm], height h=32 [mm]) with different carburized layer depth. A carburized layer was proposed in the simulations, defined by 10 sub-layers with the depth of 0.2 mm. A material with different properties was assigned to each of the sub-layers. The properties were differentiated with respect to different percentage content of carbon (carbon distribution in a layer) and the cooling-down speed. The discrete model was constructed with a hexagonal element Solid 185 (Fig. 5). Since the simulation was done in a static environment, a change of temperature as a result of "cooling down" was assigned to each node in a consecutive step of computation. The temperature was decreased from 850 $^{0}$C to 50 $^{0}$C every 10 $^{0}$C, i.e. 80 steps were calculated. In order to receive the required number of freedom, the model was "hinged" on element type: Combine 14. Moreover, in order to determine the initial deformations and stresses before the "cooling down" (quenching) process, the process of a sleeve heating up to the carburizing temperature was conducted numerically, where one core material was assigned to the sleeve. Such an approach to the problem allowed to obtain initial conditions for simulation of

carburizing and subsequent quenching. Calculations resulted in maps of deformations and stresses (Fig. 6). The values calculated in this manner allowed to determine the minimum and maximum displacements (Fig. 7).

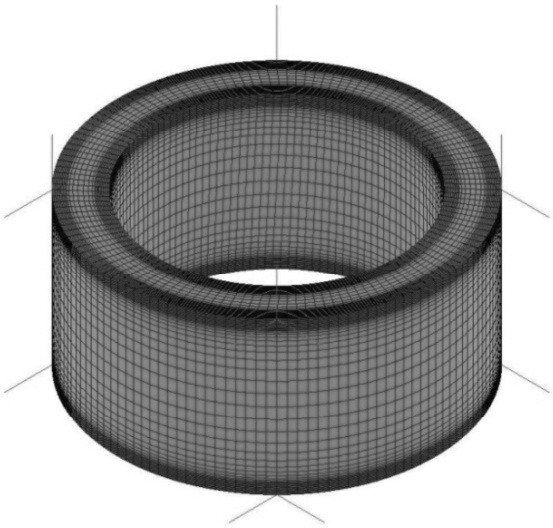

Fig. 5 A discrete model of a sleeve

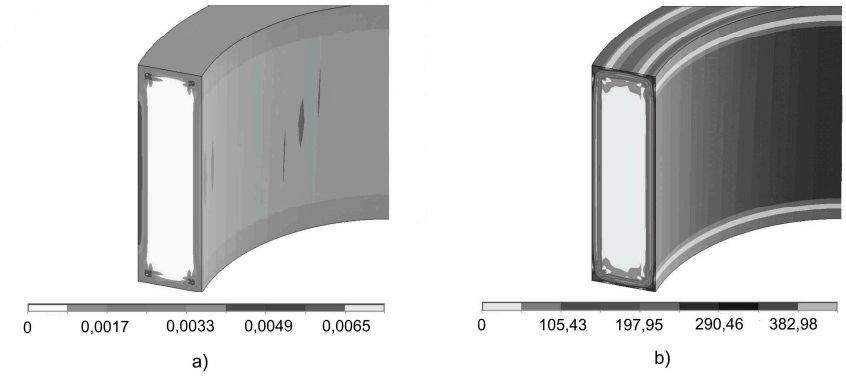


Fig. 6 A map of distribution of deformations (a) and equivalent stress (b)

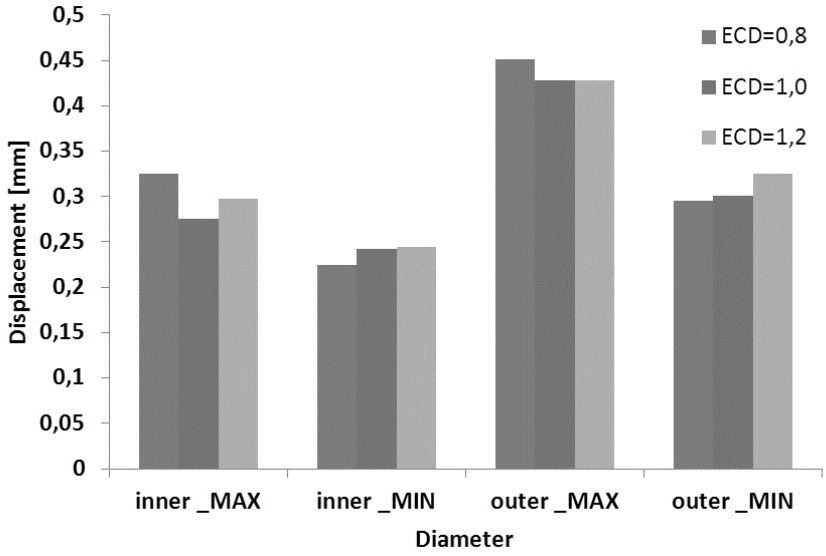


Fig. 7 Absolute minimum and maximum displacement of the inner and outer diameter of the sleeve as a function of the thickness of the carburized layer

## Construction of a comparative model - experiment

An experimental model of a sleeve was made with the same geometric parameters like in an FEM analysis. The sleeve was made from EN 16CrMn5 steel and subsequently subjected to thermal treatment. The hardened layer was formed by vacuum carburizing conducted in a single-chamber furnace with high-pressure nitrogen cooling, manufactured by Seco/Warwick S.A. The dimensions of the furnace working chamber were 600 x 600 x 900 mm. The sleeve was positioned vertically at the centre of the chamber on a specially designed device, which guaranteed uniform cooling on the entire length of the sample. The carburizing atmosphere was obtained from the mixture of acetylene – ethylene – hydrogen. Carburizing was done by the FineCarb® technology [12]. The temperature of the carburizing process was 1050°C. The parameters of the carburized layer: surface concentration: 0.75%C, effective case depth ECD=1.0 mm (for the criterion = 0.4%C). The duration of stages of saturation/diffusion to achieve the assumed profile of carbon concentration were selected on the basis of a simulation made in the SimVaCPlus® program. The carburization stage was followed by quenching from the temperature of 850°C in gas (nitrogen) at the pressure of 1.4 MPa.

Both before and after the thermal treatment, measurements were conducted of the sleeve diameter at three heights. Measurements were conducted with an amplifying gauge with the reading resolution *w*e=0.001 [mm] on an optical dividing head manufactured by Carl Zeiss Jena. The results of the experimental measurements and numerical computations for ECD=1.0 [mm] are presented in table 1.

Table 1 Comparison of experimental results with numerical computations

| Average absolute displacement[mm] | **∅50** | **∅67** |
|---|---|---|
| Experiment | **0.195** | **0.283** |
| Numerical algorithm | **0.258** | **0.364** |

The numerical computations and the experiment have shown that the difference between the findings is about 25%. This value shows that the procedure adopted and the computational algorithm can provide quite precise size of quenching displacements and deformations of carburised steel elements (they are of the same order)). The precision of results is affected by the processes that take place in the furnace chamber, which are not directly taken into account (e.g. radiation) in the FEM model, as well as the charge size. It is also believed that the precision of the gradient model can be improved by increasing the number of sub-layers and increasing the grid density in the discrete model.

## Summary

An analysis of the map of distribution of deformations and stresses shows that the layer is "working". The maximum of deformations is situated at the sleeve edges under the improved layer. This is caused by superposition of the layer deformations with the core (the layer is compressed). Moreover, it was observed that the maximum equivalent von Mises stresses of 429 MPa are concentrated in the top layer and they do not exceed the acceptable stresses resulting from thermochemical treatment (approx. $R_{e(acceptable)}$ = 1600-800 MPa for 0.8-0.6 %C).

In conclusion, it can be said that the simulations and experiments have shown that the computational algorithm used in the study provides results of simulation similar to the experimental findings. Therefore, employing the algorithm in numerical modelling enables one to predict distribution of stresses and deformations in elements subjected to thermal treatment, which helps to

make experiments more cost-effective, reduces the time of computations and experiments and reduces the need for final profiling.

The research was carried out within the project INNOTECH-K1/IN1/5/159396/NCBIR/12

**References**

[1] T. Inoue, D.Y. Ju, K. Arimoto, Metallo-thermo-mechanical simulation of quenching process—Theory and implementation of computer code HEARTS, in: G.E. Totten (Ed.), 1st International Conference on Quenching and Distortion Control, ASM International, 1992, pp. 205–212

[2] N. Jarvstrat, S. Sjostrom, Current status of TRAST: A material model subroutine system for the calculation of quench stresses in steel, in: ABAQUS Users_ Conference Proceeding, 1993, pp. 273–287

[3] SYSWELD—A Predictive Model for Heat Treat Distortion, Southwest Research Institute, 1992

[4] K. Arimoto et al., Development of heat treatment simulation system DEFORMTM-HT, in: R.A. Wallis, H. Walton (Eds.), Proceedings of the 18th Conference on Heat Treating, ASM International, Materials Park, OH, 1998, pp. 639–654

[5] B. Lynn Ferguson, A. Freborg, G. Petrus, Software simulates quenching, Advanced Materials and Processes (August) (2000) H31–H36

[6] S. V. Warke, J. Yuan, M. Maniruzzaman, M. M. Makhlouf, R. D. Sisson, Modeling the Heat Treatment of Powder Metallurgy Steels, Appendix B, Department of Mechanical Engineering Worcester Polytechnic Institute Worcester, MA 01609

[7] B. L. Ferguson, Z. Li, Stress and Deformation during Induction Hardening of Tubular Products, Proceedings from the 6th International Quenching and Control of Distortion, ASM International, September 9-13, 2012, Chicago, Illinois, USA

[8] Z. Li, B.L. Ferguson, Computer Modeling and Validations of Steel Gear Heat Treatment Processes Using Commercial Software DANTE, J. Shanghai Jiaotong Univ. (Sci), 2011, 16(2): 152-156

[9] J. Sawicki, M. Dudek, Ł. Kaczmarek, B. Więcek, T. Świątczak, R Olbrycht, Numerical analysis of thermal stresses in carbon films obtained by RF PECVD method on surface of cannulated screw. Archives of Metallurgy and Materials, 2013, 58(1), pp. 77-81

[10] J. Sawicki, M. Górecki, Ł. Kaczmarek, Z. Gawroński, K. Dybowski, R. Pietrasik, W. Pawlak, Increasing The Durability Of Pressure Dies By Modern Surface Treatment Methods. Chiang Mai Journal of Science, 2013, 40(5), pp.886-897

[11]J.Sawicki, E. Wołowic-Korecka, S. Lipa, K. Dybowski, P. Kula, P. Byczkowska: Method of determining the strain hardening of steel. Patent application No P.411176 date 02.02.2015

[12] P. Kula, J. Olejnik, P. Heilman, European Patent No.: EP1558780 (2007), United States Patent No.: US 7513958 (2009)