Modelling and Simulation of Simultaneous Forming and Quenching

GREGER BERGMAN

Department of Mechanical Engineering
Division of Computer Aided Design

1999:28 • ISSN: 1402 - 1544 • ISRN: LTU - DT - - 99/28 - - SE
Modelling and Simulation of Simultaneous Forming and Quenching

GREGER BERGMAN

Division of Computer Aided Design
Department of Mechanical Engineering
Luleå University of Technology
SE-971 87 Luleå, Sweden

Akademisk avhandling som med vederbörligt tillstånd från Tekniska fakultetsnämnden vid Luleå tekniska universitet för avläggande av teknologie doktorsexamen kommer att offentligt försvaras i sal E 246, E-huset vid Luleå tekniska universitet, måndagen den 13 december 1999, kl. 10.00.

Fakultetsopponent: Lennart Josefsson, Chalmers Tekniska Högskola, Göteborg.

Doctoral Thesis 1999:28
ISSN: 1402 - 1544
ISRN: LTU - DT -- 99/28 -- SE
Modelling and Simulation of Simultaneous Forming and Quenching

GREGER BERGMAN

Doctoral Thesis
Division of Computer Aided Design
Department of Mechanical Engineering
Luleå University of Technology
SE-971 87 Luleå, Sweden

Luleå 1999
Preface

The work presented in this thesis has been carried out at the Division of Computer Aided Design, at the Department of Mechanical Engineering of Luleå University of Technology.

The financial support for this work has been provided by the Research Council of Norrbotten.

First of all, I would like to thank my supervisor, Professor Mats Oldenburg, for his support and guidance during the course of this work. I would also like to thank Lars Sandberg and Martin Jonsson at SSAB HardTech for stimulating co-operation. Many thanks also to Jan Granström for participating in the experimental part of this work. I also wish to thank Professor Lennart Karlsson, Head of the Division of Computer Aided Design. Finally, I would like to thank all present and past colleagues at the Division of Computer Aided Design for contributing to pleasant working conditions.

Luleå in November 1999

Greger Bergman
Abstract

The objective of this thesis is to develop and evaluate numerical methods for modelling and simulation of simultaneous forming and quenching within an integrated product development environment. Simultaneous forming and quenching, also referred to as hot-stamping, is a manufacturing process for high strength automotive components such as side impact protection beams.

A concept for integrated product and process development is proposed. The prototype system consists of a CAD system with finite element modelling included, a relational database management system, program interfaces, database administration programs and nonlinear finite element programs. The integration is based on a relational database with a data model able to store complete finite element models for nonlinear analysis of thermal and mechanical problems.

A thermal model based on explicit time integration is developed and implemented into the explicit finite element code DYNA3D to solve coupled thermomechanical problems. A staggered approach is used for coupling the thermal and mechanical analysis, wherein each analysis is performed with different time step sizes. The thermal model includes two element formulations, a standard 8-node brick element and a shell element with linear temperature approximation in the plane and quadratic in the thickness direction. The geometry of the thermal shell element is defined as in the DYNA3D implementation of the Hughes-Liu shell. Stability characteristics of the shell element is investigated with respect to the explicit forward difference method and a lumped capacity matrix. Thermal contact, in which heat transfer depend on the mechanical deformation, is included in the thermal model. A pressure dependent contact conductance model is employed. The material behaviour is described by a thermo-elastic-plastic material model. The effective-stress-function algorithm is used to update the stresses. Transformation plasticity is included in the model.

The implemented methods are evaluated by comparison with corresponding experimental results. In one of the developed experiments, pre-heated steel plates are simultaneously formed and quenched by a cold tool. The analyses show good agreement during the initial forming stage, followed by an overestimation of the tool force at sequential times. It is shown that the computed tool force is very sensitive to the sequence of cooling in different parts of the plate.

Keywords: finite element method, explicit time integration, shell element, effective-stress-function algorithm, simultaneous forming and quenching, integration, relational database
This thesis comprises a survey and the following five appended papers:


D  G. Bergman and M. Oldenburg, A finite element model for thermomechanical analysis of sheet metal forming, to be published.

E  G. Bergman, L. Sandberg and M. Oldenburg, Finite element analysis of simultaneous forming and quenching of thin-walled structures, to be published.
### Contents

<table>
<thead>
<tr>
<th>Section</th>
<th>Title</th>
<th>Page</th>
</tr>
</thead>
<tbody>
<tr>
<td>Preface</td>
<td></td>
<td>i</td>
</tr>
<tr>
<td>Abstract</td>
<td></td>
<td>ii</td>
</tr>
<tr>
<td>Thesis</td>
<td></td>
<td>iii</td>
</tr>
<tr>
<td>1</td>
<td>Introduction</td>
<td>1</td>
</tr>
<tr>
<td>2</td>
<td>Integrated product development</td>
<td>2</td>
</tr>
<tr>
<td>3</td>
<td>Modelling of simultaneous forming and quenching</td>
<td>3</td>
</tr>
<tr>
<td></td>
<td>3.1 Material modelling</td>
<td>4</td>
</tr>
<tr>
<td></td>
<td>3.2 Modelling of boundary conditions</td>
<td>6</td>
</tr>
<tr>
<td>4</td>
<td>Numerical procedures</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>4.1 Finite element formulation</td>
<td>7</td>
</tr>
<tr>
<td></td>
<td>4.2 Linear-quadratic thermal shell element</td>
<td>8</td>
</tr>
<tr>
<td></td>
<td>4.3 Integration of the constitutive equations</td>
<td>9</td>
</tr>
<tr>
<td>5</td>
<td>Experiments and evaluations</td>
<td>9</td>
</tr>
<tr>
<td></td>
<td>5.1 One-sided spray cooling</td>
<td>10</td>
</tr>
<tr>
<td></td>
<td>5.2 Simultaneous forming and quenching</td>
<td>11</td>
</tr>
<tr>
<td>6</td>
<td>Summary of appended papers</td>
<td>12</td>
</tr>
<tr>
<td></td>
<td>6.1 Paper A</td>
<td>12</td>
</tr>
<tr>
<td></td>
<td>6.2 Paper B</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>6.3 Paper C</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>6.4 Paper D</td>
<td>13</td>
</tr>
<tr>
<td></td>
<td>6.5 Paper E</td>
<td>14</td>
</tr>
<tr>
<td>7</td>
<td>Conclusions and future work</td>
<td>14</td>
</tr>
<tr>
<td>References</td>
<td></td>
<td>15</td>
</tr>
</tbody>
</table>

### Appended papers

<table>
<thead>
<tr>
<th>Paper</th>
<th>Title</th>
</tr>
</thead>
<tbody>
<tr>
<td>A</td>
<td>Integration of a product design system and nonlinear finite element codes via a relational database</td>
</tr>
<tr>
<td>B</td>
<td>Experiments with one-sided spray-cooling of steel plates for evaluation of thermo-mechanical material models</td>
</tr>
<tr>
<td>C</td>
<td>Verification of thermomechanical material models by thin-plate quenching simulations</td>
</tr>
<tr>
<td>D</td>
<td>A finite element model for thermomechanical analysis of sheet metal forming</td>
</tr>
<tr>
<td>E</td>
<td>Finite element analysis of simultaneous forming and quenching of thin-walled structures</td>
</tr>
</tbody>
</table>
1 Introduction

Simultaneous forming and quenching is a manufacturing process for low weight and high strength automotive components. The process, also referred to as hot-stamping, has been used mostly for safety related structural components such as side impact protection beams and bumper systems. An increased use of high strength hardened components in vehicle structures can be foreseen due to the possibilities of weight reductions, while simultaneously fulfilling requirements for enhanced vehicle safety. The major challenge is how to manufacture low weight and high strength components. High strength steels have limited formability and can not be cold formed into complex shapes. On the other hand, mild steels can be formed but its yield strength is often too low. Alternatively, it is possible to use a hardenable steel, forming it in the cold state, followed by hardening and tempering. This procedure can, however, lead to distortion which may require that the quenching is done in a special tool that prevent the part from deforming or that the part is straightened afterwards. These additional operations are expensive and generally raise the price of the product above the level the market can support. The hot-stamping process uses boron steel blanks which are first austenitized and then subsequently formed and quenched between cooled tools. Complex shapes can be formed since the heated material have high formability and the subsequent quenching gives a material with very high yield strength. Furthermore, the dimensional accuracy of the process is comparable to conventional forming of mild steels.

The design of forming tools and determination of process parameters for new products constitute a considerable cost. This design process is often iterative, a prototype is manufactured and tested, and if it fails to fulfil the functional requirements it is re-designed. With numerical simulations in the design stage, the number of iterations can be minimised and the lead time and cost for product development can be reduced [1]. Numerical methods such as the finite element method [2, 3] has proved to be a viable approach for simulation of manufacturing processes and simulation of component functionality. Because of the complexity of these analyses, there is no general-purpose software that can cover the entire range of applications. Instead, various special-purpose software modules have to be used. Hence, integration between these software modules is required to be able to transfer data from one software to another.

The aim of the present work is to develop and evaluate numerical methods for modelling and simulation of the hot-stamping manufacturing process within an integrated product development environment.
2 Integrated product development

The product development process consists of many tasks dependent on computer based tools which operate on, more or less, the same set of data. The different activities involved in the development of a mechanical component may be geometry definition, simulation of the manufacturing process or simulation of component functionality. Ideally, one general-purpose system that provides solutions to all aspects of the product development process would be preferred. Unfortunately, there is no system available with such qualities. However, there are many powerful computer based tools suited for specific tasks, e.g. geometric modelling, finite element analysis, rigid/flexible body dynamics and visualisation. A large part of the profit arising from the use of such systems comes from integration, giving the possibility to efficiently transfer data from one tool to another. In general, the computer integrated manufacturing (CIM) environment need to consist of programs from different suppliers. It is therefore important that the integration is achieved by use of protocols and tools which conform to present standards where it is possible. The data structures used in communication and storage must be as general as possible. In such an environment, one program can be replaced with another with the same functionality without affecting the complete environment. The integration can be done either by program specific interfaces or by using a neutral database, see Figures 1 and 2.

![Diagram](image1)

**Figure 1.** CIM environment with program specific interfaces [Paper A].

![Diagram](image2)

**Figure 2.** CIM environment with interfaces linked to a neutral database [Paper A].

Using the first approach, a large number of interfaces need to be installed in the system, and replacing one program will affect the complete environment. In the second approach, the neutral database serves as a link between the programs in the system. A database for finite element analyses serves two main purposes. The analysis models and results within a project can be managed in an efficient manner.
The second purpose is to simplify the integration process. The neutral data structure makes it easy to produce interfaces to the programs included in the integrated environment. If a new program is added, only one interface program need to be produced. A prototype system for integrated product and process development is presented in Paper A.

3 Modelling of simultaneous forming and quenching

Simulation of hot-stamping involves additional modelling aspects compared to conventional cold sheet metal forming. The blank, initially heated to austenitising temperature, cools during the process due to heat transfer to the tools, and the steel undergoes martensitic transformation. Consequently, a realistic simulation of combined forming and quenching must consider interactions between the mechanical field, the thermal field and the microstructure evolution. The possible couplings are shown in Figure 3 and are explained in Table 1. Table 1 also shows couplings considered in the present work. Some of these couplings are more important than others. For example, heat generation due to plastic dissipation can be neglected since it is small compared to the heat flow between the blank and the tools. However, there may be couplings not included in this work which can have a substantial influence on the results of the simulations. Most of the couplings listed in Table 1 are related to material modelling, with the exception of coupling 1a which is related to the modelling of boundary conditions.

![Figure 3. Couplings between mechanical field, thermal field and microstructure evolution.](image-url)
Table 1: Couplings in Figure 3.

<table>
<thead>
<tr>
<th>Coupling</th>
<th>Explanation</th>
<th>Considered</th>
</tr>
</thead>
<tbody>
<tr>
<td>1a</td>
<td>Thermal boundary conditions depend on deformation</td>
<td>x</td>
</tr>
<tr>
<td>1b</td>
<td>Heat generation due to plastic dissipation</td>
<td></td>
</tr>
<tr>
<td>2</td>
<td>Thermal expansion</td>
<td>x</td>
</tr>
<tr>
<td>3a</td>
<td>Thermal material properties depend on microstructure evolution</td>
<td>x</td>
</tr>
<tr>
<td>3b</td>
<td>Latent heat due to phase transformations</td>
<td>x</td>
</tr>
<tr>
<td>4</td>
<td>Microstructure evolution depends on temperature</td>
<td>x</td>
</tr>
<tr>
<td>5a</td>
<td>Mechanical material properties depend on microstructure evolution</td>
<td>x</td>
</tr>
<tr>
<td>5b</td>
<td>Volume change due to phase transformations</td>
<td>x</td>
</tr>
<tr>
<td>5c</td>
<td>Transformation plasticity</td>
<td>x</td>
</tr>
<tr>
<td>5d</td>
<td>Memory of plastic strains during phase transformations</td>
<td></td>
</tr>
<tr>
<td>6</td>
<td>Phase transformations depend on stresses and strains</td>
<td></td>
</tr>
</tbody>
</table>

3.1 Material modelling

There are mainly two methods that can be used for the determination of how the thermal and mechanical material properties depend on the microstructure evolution (couplings 3a and 5a). Firstly, based on information of the volume fractions of phases (austenite, ferrite, pearlite, bainite and martensite) and their properties it is possible to estimate the overall material properties using mixture rules [4, 5]. The microstructure evolution (coupling 4) is computed from the temperature history. An algorithm for microstructural predictions is presented in [6]. Martensite formation is computed using the Koistinen-Marburger equation [7]. Such an approach is considered to be the most general since it is not limited to a specific temperature history. Secondly, the dependency on the microstructure of the material can be included directly in its properties by performing material characterisation experiments with an appropriate temperature history [8]. The temperature dependent thermal and mechanical properties of the steel material used in this work, SS 142550-02, have been obtained mostly by the second approach and are shown in Figures 4 and 5. The latent heat release (coupling 3b) during austenite to martensite transformation is included in the heat capacity curve. The thermal
dilatation, which is the driving force for thermal stresses, includes both thermal expansion and volume change due to martensite formation (coupling 2 and 5b).

Transformation plasticity (coupling 5c) is an irreversible deformation that occurs when a material undergoes phase transformation under applied stresses well below the yield strength of the material. Many authors have shown that transformation plasticity can be important in determining the nature and magnitude of residual stresses, see e.g. [9, 10]. Different transformation plasticity models have been proposed, a review can be found in [11]. A model proposed by Leblond et al. [12] is used in the present work, see Paper C.

![Figure 4. Heat capacity (c), thermal conductivity (k), Young’s modulus (E) and thermal dilatation ($e^T$) [Paper E].](image)

![Figure 5. Yield stress versus effective plastic strain at elevated temperatures [Paper E].](image)
Memory of plastic strains during phase transformations (coupling 5d) is related to modelling the yield strength of the material. The plastic strain tensor and effective plastic strain are state variables commonly used to describe the plastic history and hardening that has occurred in the material. In [4, 13], these state variables are replaced by new quantities to account for the possible recovery of strain hardening during phase transformations. That is, the fact that the newly formed phase can have only partial memory, or even no memory at all, of the previous hardening. How much of the dislocation structure that should be remembered depends on the transformation. In [14], it is stated that the memory of previous plastic deformation disappears for ferritic and bainitic transformations. Martensite formation, which involves only small displacements of atoms, is best described by full memory of the dislocation structure.

The influence of stresses on the kinetics of phase transformations (coupling 6) during quenching have been investigated in e.g. [10, 13]. For martensitic transformations, an increase in the $M_s$ temperature for uniaxial tensile and compressive stresses, although less by compressive stresses, and a decrease in $M_s$ for hydrostatic stresses can be observed [15]. Plastic deformation of the austenite prior to martensite transformation can aid both nucleation and growth of martensite, but too much plastic deformation may suppress the transformation [16].

### 3.2 Modelling of boundary conditions

Cooling of the material takes place mainly by heat transfer through the contact interface between the workpiece and the tools. Accurate treatment of the evolving thermomechanical contact is essential in processes where both the workpiece and tool behaviour are affected strongly by the temperature fields. The resistance to heat transfer when rough surfaces are pressed together is mainly due to the low percentage of surface area really in contact. The heat transfer through the contact interface takes place by conduction through the contacting spots, conduction through the interstitial gas, and radiation across the gaps. The heat flux between the contact surfaces is usually formulated in terms of the contact conductance and the surface temperatures, see Paper D. The difficult part in the modelling of thermomechanical contact is to obtain relevant values of the contact conductance coefficient. A large number of contact conductance models are available, see [17, 18] for a review. These models indicate that the contact conductance depends on the material of the bodies, the microscopic shape of the surfaces, the contact pressure and other factors. A method based on experiments and inverse modelling techniques for determination of the contact conductance as a function of
temperature and pressure is presented in [19].

4 Numerical procedures

The modelling of the hot-stamping manufacturing process involves several sources of nonlinearities, which has to be properly handled by the numerical procedures employed in order to reach satisfactory results. Useful numerical models have two requirements. The first is accurate relations to describe the physical phenomena involved, which has been discussed in the previous section. The second is effective and physically realistic numerical procedures. The discussion that follows concentrates on the implemented numerical procedures made to be able to simulate combined forming and quenching.

4.1 Finite element formulation

In practice, both sheet metal forming and quenching operations are sufficient slow to be classified as quasi-static processes. The apparent choice would therefore be to use an implicit finite element method. However, due to the material, geometrical and contact nonlinearities involved in the sheet forming process, very short time steps are required to reach convergence. This has led to the use of explicit dynamic methods where the above mentioned issues do not affect the cost of the analysis significantly [20]. On the other hand, a subsequent springback analysis is probably done more efficiently with an implicit method. The major drawback of using an explicit method in simulation of the forming stage is that the stable time step size is very small compared to the natural time of the process. To increase the computational efficiency of the explicit method, two numerical artifices are generally employed: i) the stability limit is increased by increasing the material density; ii) the natural time of the process is artificially reduced by increasing the velocity of the tool. The amounts by which the material density and tool velocity can be increased is, however, limited. The chosen density and tool velocity must be such that the inertia forces do not influence the results in an unacceptable way. In the present work, the mechanical part of the problem has been solved by explicit time integration, using DYNA3D [21].

Thermomechanical analyses in standard DYNA3D are based on an uncoupled approach in which the thermal problem, based on the initial geometry, is solved by an external implicit code, and the resulting temperature history is used as thermal loads in the mechanical problem. The intended application requires a thermal model applicable to heat transfer analysis of thin-walled structures subjected to thermal
contact. Furthermore, a coupled analysis is required since the deformation changes the thermal boundary conditions. In Paper D, a thermal model based on explicit time integration is developed and implemented into DYNA3D. A staggered approach is used for coupling the thermal and mechanical problem. In this approach, separate analyses are performed with data exchange at the end of each time step. In particular, the thermal analysis is based on the geometry and contact data calculated at the end of the previous mechanical step. The resulting thermal field is then used in the mechanical analysis. Since the stable time step size in the thermal analysis is normally much larger than in the mechanical analysis, see Paper D, each analysis is performed with different time step sizes. That is, between two consecutive thermal steps, many mechanical steps are performed. During these intermediate mechanical steps a linearly interpolated temperature field is used.

4.2 Linear-quadratic thermal shell element

In the application considered here, shell elements is the natural choice for modelling the structural behaviour of the blank. It is therefore advantageous if the same geometry description can be used in both the mechanical and thermal analysis. Shell elements for heat conduction are normally derived from three-dimensional isoparametric solid elements where two faces of arbitrary order are connected by linear edges [22], resulting in a linear temperature approximation in the thickness direction. In applications where the shell surfaces are subjected to one-sided or double-sided thermal contact, the linear temperature approximation in the thickness direction is not adequate.

Figure 6. Linear-quadratic thermal shell element.
For accurate heat transfer analysis of such applications, an element with linear temperature approximation in the plane and quadratic in the thickness direction is proposed in *Paper D*. The temperature approximation is derived by introducing additional temperature nodes in the thickness direction, see Figure 6. For the linear-quadratic element, 12 temperature nodes are required.

### 4.3 Integration of the constitutive equations

The choice of constitutive equations and stress calculation algorithm will have a substantial influence on the predictive capabilities of the numerical model. The quality of the results achieved depends strongly on a correct representation of the material's stress-strain relationship. However, the results obtained can be no better than the accuracy that the available material properties allow. Constitutive models for thermomechanical analysis concern the effects of strain, temperature and phase transformation on stress. For a given increment of these quantities, the stress increment is calculated at the Gauss points by integration of the constitutive equations. The basic parts of a constitutive model are a yield function, a flow rule and a hardening law. In the present work, a rate-independent thermo-elastic-plastic constitutive model is used, see *Paper C* and *Paper D*. The plastic behaviour of the material is described by von Mises isotropic yield condition, an associated flow rule, and mixed linear isotropic-kinematic hardening or nonlinear isotropic hardening. Transformation plasticity is included in the model as an additional strain component related to the stress state and the progress of transformation. Once the increments of total strain and thermal strain are known it becomes possible to compute the elastic and inelastic strain increments, and finally the stress. The problem of finding the final stress state, which in the case of plastic loading must lie on the yield surface, can be solved in a number of ways. In the present work, a modified version of the effective-stress-function algorithm [23] is used, see *Paper C* and *Paper D*.

### 5 Experiments and evaluations

Experiments have always been very important in the development of numerical methods. They are needed for obtaining input data to the computational models, e.g. material properties, and for verification of the computational models. To evaluate the implemented methods presented in *Paper C* and *Paper D*, two experiments have been developed which are described in *Paper B* and *Paper E*. A brief presentation of the experiments and corresponding results from simulations is given here.
5.1 One-sided spray cooling

The purpose of the experiment developed in Paper B is to evaluate material models for simulation of quenching. With this experiment, the mechanical response can be monitored throughout the time-history of the test. The experimental set-up shown in Figure 7 consists of a housing with water spray nozzles and measurement devices for temperature and deformation. The test specimen is austenitized and thereafter subjected to one-sided spray cooling. The simulations in Paper B and Paper C are performed in two steps. The thermal analysis is followed by the mechanical analysis. The implicit codes TOPAZ2D [24] and NIKE2D [25] are used in the thermal and mechanical analysis, respectively.

![Experimental set-up for one-sided spray cooling experiments](image)

The experimental set-up and measurement equipment is validated using a specimen produced from Inconel 600 which does not undergo phase transformations during the cooling process. The measured and computed deformation histories for the Inconel 600 plate are shown in Figure 8. The simulations show acceptable agreement with the measured deformation. The difference between the calculated and measured deformation is believed to be caused by a deviation from an even distribution of the water spray over the plate. The measured and computed deformation histories for the SS 142550-02 plate are shown in Figure 9. The results from these analyses confirm the interpretation of the results from the Inconel 600 experiment. The experiment seems to be very sensitive to an uneven distribution of the cooling spray, causing the phase transformations to occur in different moments of time over the plane of the plate. Since the deformation is a sum of the response of every point in the plate, the large amplitude observed in the simulations is not captured in the experiment. For the conditions examined, transformation plasticity must be taken into account. The analyses of the present experiment show that the
permanent deformation of the plate are exclusively due to transformation plasticity.

Figure 8. Measured and calculated deformation in the Inconel 600 experiment [Paper B].

Figure 9. Measured and calculated deformation in the SS 142550-02 experiment [Paper B].

5.2 Simultaneous forming and quenching

The purpose of the experiment developed in Paper E is to evaluate the developed and implemented methods presented in Paper C and Paper D. The experimental set-up shown in Figure 10 consists of a cylindrically shaped tool and a plate support designed to avoid draw-in of material during the test.

Figure 10. Experimental set-up for simultaneous forming and quenching experiments.
The set-up is designed to include most of the characteristics of the industrial application of hot-stamping under well defined boundary conditions. In the experiment, pre-heated steel plates are simultaneously formed and quenched by the cold tool. The plate is modelled with shell elements in the mechanical part of the analysis. The linear-quadratic thermal shell element is used for the heat transfer analysis in the plate. The tool is considered to be rigid in the mechanical analysis, but is modelled with 8-node brick elements since the same mesh is used in both the mechanical and thermal analysis. The measured and computed tool force are shown in Figure 11. The tool force is computed with and without transformation plasticity (trp), in both cases including the transformation volume change. After the forming stage is completed, i.e., after approximately 1 s, the force increases due to thermal shrinkage. When the $M_s$ temperature is reached in the plate, the force decreases. The simulations shows acceptable agreement during the initial forming stage, followed by an overestimation of the force for times between 2 s and 10 s. The computed force without trp shows acceptable agreement at sequential times. When trp is included the calculated force decreases rapidly when $M_s$ is reached.

![Figure 11. Measured and computed tool force [Paper E].](image)

**6 Summary of appended papers**

**6.1 Paper A**

A concept for integrated product and process development is presented. The prototype system consists of a CAD system with finite element modelling included, a relational database management system, program interfaces, database
administration programs and nonlinear finite element programs. The integration is based on a relational database with a data model able to store complete finite element models for nonlinear analysis of thermal and mechanical problems. The communication with the database management system is based on the structured query language (SQL). The program interfaces and database administration programs are written in the C language with embedded SQL commands.

6.2 Paper B

In this paper a one-sided spray cooling experiment is developed and evaluated. The purpose of the experiment is to evaluate material models for simulation of quenching. The set-up and measurement equipment is validated using a specimen produced from Inconel 600 which does not undergo phase transformations during the cooling process. The simulation of the Inconel 600 test show acceptable agreement with the measured deformation. The difference between the calculated and measured deformation is believed to be caused by a deviation from an even distribution of the water spray over the plate. A comparison between measured and computed deformation for a material that undergoes martensitic transformation during the cooling process shows discrepancies during the transient stage due to the uneven distribution of the water spray. The computed final deformation does, however, agree well with the experiment.

6.3 Paper C

A thermomechanical material model applicable to quenching simulations is presented. The material behaviour is described by a temperature dependent elastic-plastic model with a mixed isotropic-kinematic hardening law. Transformation plasticity is included in the model. The effective-stress-function algorithm is used to update the stresses. Simulations and results from the one-sided spray cooling experiment are used to evaluate the material model.

6.4 Paper D

This paper describes the development and evaluation of methods for modelling and simulation of the hot-stamping process. A thermal model based on explicit time integration is developed and implemented into the explicit finite element code DYNA3D to solve coupled thermomechanical problems. The implementation includes a thermal shell element with linear temperature approximation in the plane and quadratic in the thickness direction, as well as algorithms for contact heat
transfer.

6.5 Paper E

Finite element analyses of simultaneous forming and quenching is presented. A forming and quenching experiment is developed in order to evaluate the developed methods in Paper D. In the experiment, pre-heated steel plates are simultaneously formed and quenched by the cold tool. Tensile tests at elevated temperatures are performed in order to characterise the material response during quenching. The analyses show good agreement during the initial forming stage, followed by an overestimation of the tool force at sequential times. It is shown that the computed tool force is very sensitive to the sequence of cooling in different parts of the plate.

7 Conclusions and future work

The objective of this thesis has been to develop and evaluate numerical methods for modelling and simulation of simultaneous forming and quenching within an integrated product development environment. The outgrowth of this work is considered to be a potential tool that can assist the design of forming tools and the determination of process parameters for high strength steel components. It is also the base for further investigations and developments in this research field.

The use of integration in the product development process has a substantial significance for the productivity. In the case of numerical simulations, the computational models can be defined in one single system independent of which finite element program is going to be used. The concept of a neutral database decreases the number of interfaces in the system. The exchange of programs is facilitated, if a new program is selected, only one interface need to be altered. The system developed in Paper A has been used to define all the computational models used in Paper B to Paper E.

The developments and implementations presented in Paper C and Paper D are necessary parts in a future production analysis code for the industrial application of hot-stamping. The evaluation has shown that explicit methods can be used for simulation of hot-stamping. It is also shown that the linear-quadratic thermal shell element is capable of accurate modelling of heat transfer in thin-walled structures subjected to thermal contact. To extend the predictive capabilities of the developed methods future work is needed, mainly in the field of material testing and material modelling methods.
References


INTEGRATION OF A PRODUCT DESIGN SYSTEM AND NONLINEAR FINITE ELEMENT CODES VIA A RELATIONAL DATABASE

G. BERGMAN, M. OLDENBURG AND P. JEPPSSON
Department of Mechanical Engineering, Luleå University of Technology, S-97187 Luleå, Sweden

ABSTRACT
A database for finite element models and related data is developed and incorporated into a prototype system for integration of non-linear finite element codes with a product design system. In the prototype system, the database is used as a link for integrating commercial, public domain as well as in-house codes. In the present system, the public domain finite element codes NIKE2D, NIKE3D, DYNA2D, DYNA3D and TOPAZ2D are integrated with the CIM-system I-DEAS. The prototype system is primarily intended as a platform in research projects for development of integrated environments tuned for simulations of specific manufacturing processes such as quenching, welding, hot rolling, metal powder compaction and hot isostatic pressing.

KEY WORDS Integration Relational database Product design system Finite element analyses

INTRODUCTION
The product development process consists of many tasks dependent on computer based tools which operate on, more or less, the same set of data. The different activities involved in the product development of a mechanical component may be geometry definition, simulation of functionality or simulation of manufacturing. One goal for the future in the present developments of CIM-systems is to create a complete integration of the different computer based tools used in product development. This goal has been achieved to some extent in commercial products. However, there is in general no single commercial CIM-system that provides solutions to all aspects of the development process of a specific industrial product.

There are many powerful computer based tools suited for specific tasks, e.g.,

- Geometric modelling
- Structural analysis, linear and nonlinear
- Rigid/flexible body dynamics
- Drawing production
- Visualisation and animation
- Preparation of computer controlled manufacturing

A large part of the profit arising from the use of such systems comes from integration, giving the possibility to efficiently transfer data from one tool to another. In general, the CIM environment need to consist of products from different suppliers. It is important that the
integration is achieved by use of protocols and tools which conform to present standards where it is possible. The data structures used in communication and storage must be as general as possible. In such an environment, one program can be replaced with another with the same functionality without affecting the complete environment.

**GENERAL**

The development of the presented system is based on the need to efficiently perform non-linear analyses of manufacturing processes. The present CIM systems typically include integration with linear FE-solvers, and some systems provide interfaces to commercial nonlinear FE-codes. These interfaces vary in quality and often lack functionality for e.g. material specifications or contact definitions. With interfaces connecting specific codes, a large number of interfaces need to be installed in the CIM-environment, see Figure 1. In the present work, the integration is based on the functionality needed for the modelling and the nonlinear analysis. Each commercial program in the integrated system is considered as a tool for providing a function to the integrated environment. Thus, one program can be changed for another program with the same functionality. The basic functions needed in a computer-based environment for manufacturing simulations are model generation based on a geometric model, material and process definitions, nonlinear structural analysis and result presentation.

In order to meet the requirements stated above, the integration need to be built on a neutral database, independent from suppliers of commercial CIM systems, see Figure 2. By using a database manager based on present standards, e.g. the structured query language (SQL), the database manager can be obtained from several suppliers as well. Another advantage of SQL-based databases for technical environments is that it can be integrated with present systems for economy and production administration. A database for structural analyses serves two main purposes. The administration of analysis models and results within a project can be centralised. Within this environment, the different versions and models of the product can be validated by the project management in an efficient manner. The second purpose is to simplify the integration process. The neutral data structure makes it easy to produce interfaces to the programs included in the integrated environment. If a new program is added, only one interface program need to be produced. The process of searching and collecting data for the interfaces is facilitated by the use of the relational data model\(^1\).

---

**Figure 1** CIM-environment with program specific interfaces

**Figure 2** CIM-environment with interfaces linked to a neutral database
INTEGRATION VIA A RELATIONAL DATABASE

THE PROTOTYPE SYSTEM

System configuration
The present integrated environment consists of a geometric modeller with finite element modelling included, a relational database manager, structural analysis codes for nonlinear analysis, integration interfaces and database administration programs, see Figure 3. In the prototype system, the following codes are included:

- I-DEAS CIM-system\(^2\).
- Informix-OnLine database manager\(^3\).
- NIKE2D two-dimensional implicit finite element code\(^4\).
- DYNA2D two-dimensional explicit finite element code\(^5\).
- NIKE3D three-dimensional implicit finite element code\(^6\).
- DYNA3D three-dimensional explicit finite element code\(^7\).
- TOPAZ2D two-dimensional heat transfer code\(^8\).
- In-house codes for the user interface, program interfaces, material data and model administration.

The I-DEAS system is a general CIM system with modules for geometric modelling including solid and surface modelling, a general purpose finite element model generator, static and dynamic finite element solver, tools for kinematic and dynamic simulations of assembled rigid bodies and modules for simulation and preparation of manufacturing. Transfer of data to and from the system can be performed with use of the complete and well documented universal files or via the local relational database Pearl. Geometric data can also be transferred to some extent with use of IGES files. The functions used in the integrated system are geometric modelling and finite element model generation. The data is transferred using universal files.

The DYNA2D and DYNA3D codes are nonlinear, explicit finite element codes for two- and three-dimensional analysis, respectively. They have been used extensively for fast transient problems such as high velocity impact analyses and crashworthiness analyses where the effects from wave propagation are significant. DYNA3D has good capabilities concerning analyses of general contacting systems. Explicit codes are also used in highly nonlinear large deformation analyses of manufacturing processes such as metal powder pressing and hot rolling. NIKE2D and NIKE3D are nonlinear, implicit finite element codes for two- and three-dimensional analysis,
respectively. These programs are used for nonlinear, quasi-static and dynamic solid and structural analysis. TOPAZ2D is an implicit finite element code for heat transfer and mathematically equivalent problems. The resulting temperature histories can be used as temperature loading in the structural analyses performed by DYNA2D or NIKE2D. In this case, the model generated in I-DEAS and stored in the database can be used both for the heat transfer analysis and the structural analysis.

The Informix-OnLine system is a relational database management system. The communication with the database manager is based on the structured query language (SQL). The database can be accessed in a client-server computer configuration and one database can be distributed to, and accessed from, many servers. The database can be created and accessed from an interactive SQL-interpreter or from C or Fortran programs with embedded SQL commands. Another option is to create programs with a fourth generation language tool (4GL).

User interface and communication programs

The user interface is event-driven and uses the XView window system based on the X-window protocol. There is a main window where the user can choose between different activities within the CIM-environment. Dependent on the choice done in the main window, the selected communication program is executed. An example of the window appearances is shown in Figure 4. The execution of communication programs could be included in user defined menus in the CAD-system as well. All user interface and communication programs in the system are written in the C-language and the programs that communicate with the database manager have embedded SQL commands.

![Figure 4](image-url) Example of the user interface window layout
The interface between the CIM-system and the selected finite element code is divided into two steps. The model is read from a universal file created by I-DEAS and then stored into the database. When a finite element code is selected, one program searches in the database for the relevant data and creates an input file for the selected FE-code.

THE FINITE ELEMENT DATA MODEL

The finite element data model used in the database is defined as general as possible. The intent is to be able to use the data model in environments with other CIM-systems, finite element solvers and result presentation programs. In the present system, it is possible to define a three-dimensional model with boundary conditions, loads, material definitions and contact surface definitions and use the model directly for both DYNA3D and NIKE3D. For a large class of problems, the complete input file can be generated from the information stored in the model database.

Supported finite element entities

The data items that can be defined in the I-DEAS system and transferred to the database are:

- Elements.
  The supported elements are plane, axisymmetric, shell and solid isoparametric elements.
- Nodes.
- Local coordinate systems.
- Material properties.
  The material name and the static coefficient of friction are the only items in the material property list that are transferred to the database. The material name refers to a combination of a material type and a material model defined in the material database.
- Physical properties.
  Thickness of plane and shell elements.
- Boundary and initial conditions.
  Restraints, initial velocities, prescribed velocities and prescribed displacements as well as initial and prescribed temperatures are transferable to the database. Each set of prescribed velocities, displacements and temperatures are associated to load curves that can be defined in the database.
- Loads.
  Nodal structural loads as well as edge and face pressures can be transferred to the database. Edge heat radiation and convection can also be defined. All loads are associated with load curves.
- Contact surface definitions.
  Contact surfaces in three-dimensional models can be defined in I-DEAS as membrane elements covering the contact surface on the solid elements. These membrane elements are defined as contact surfaces in the database. The type of algorithm used for contact detection and contact force evaluation is determined by the material name associated with the membrane elements. Contact edges are defined for the two-dimensional models in the same manner in the database but must be defined in a different way in the I-DEAS system. In this case nodes associated with slide-lines are collected into groups with names defining the algorithm type.

A detailed description of each data type stored in the finite element model database is found in Bergman et al.¹⁰.
The relational data model

In the relational data model, data is stored in tables and the information in different tables are related by the content in the table columns. With the relational data model defined in the presented database, an hierarchical model can be stored in a relational database. Many databases can be defined in the system and each database can store several models. A similar strategy has been used by Jeppsson and Oldenburg for the storage of geometry by a boundary representation in a relational data model. A relational data model for finite element data has previously been proposed by Yang and a finite element system with model storage in a relational database.

![Figure 5 Tables and columns of the finite element entity data model](image-url)
have been presented by Kröplin et al.\textsuperscript{13}. The data model in the present system is defined in a general manner. For example, there are no assumptions or restrictions made on the number of nodes of each individual element in the model. Furthermore, a search for the elements connected to a specific node is as straightforward as a search for the nodes of an element. The tables of the data model are shown in Figure 5. The material database is stored separate from the finite element models and the material definitions can be defined, updated and deleted from a special program in the system. The material data model is shown in Figure 6.

**EXAMPLES**

The examples are shown in order to present the capabilities of the integrated system and the manner in which the models are defined in the I-DEAS system. Up to this date, the integrated environment has been used in different development and research projects as well as in education situations. Experience from these projects tells us that the integrated environment significantly improves the productivity. In the research projects, it is most common that special versions of the finite element programs are used. In some of the presented examples, new material models and contact algorithms are tested.

**Metal powder pressing**

The pressing of a hard metal powder component\textsuperscript{14} is analysed with a special version of DYNA2D. The component is analysed for the density distribution which influences the quality of the sintered product. A special cap-plasticity material model is implemented into DYNA2D in order to simulate the behaviour of the powder during the compaction process. However, the definition of this material model is not included in the present version of the material database. In this example a single surface contact algorithm is used, see Oldenburg and Häggblad\textsuperscript{15}. The definition of contact segments for this algorithm is included in the database. The model definition includes mapped and free mesh generation, two material definitions, prescribed velocities, restraints and contact surfaces, see Figure 7.
Hot isostatic pressing of a turbine hub

A steel container filled with metal powder is subjected to hot isostatic pressing (HIP) in order to obtain a fully densified turbine hub with the desired final shape, see Jeppsson and Svoboda\(^1\). In this case TOPAZ2D and a special version of NIKE2D is used. The powder behaviour is described by a material model depending on the applied pressure, the temperature and the time. Free mesh generation is used and temperature loading as well as pressure loading are generated and transferred to the database, see Figure 8. The same FE-model is used for the thermal and the mechanical analyses.

Assembly of an electrical connection device

In this analysis a small cylinder is pressed and deformed by a pressing tool onto an electrical wire in an electronic assembly. The established pressure between the wire and the cylinder is analysed with respect to dimensional tolerances using DYNA3D. An elastoplastic material model is used for the wire and for the cylinder. The pressing tool is modelled as a rigid shell structure with a prescribed velocity. The symmetry conditions give rise to a skew symmetry plane and on this plane the boundary conditions are defined in a local coordinate system. The master-slave contact algorithm is defined on two contact surfaces, between the tool and the cylinder and between the cylinder and the wire, see Figure 9.

Impact between a rod and a wall

A rod with initial velocity impacing a wall is analysed. The definition of this problem follows a demonstration example in the NIKE3D users manual\(^6\). The model is generated with boundary conditions, initial velocities, elasto-plastic material model of the rod and specification of a contact interface using the master-slave algorithm, see Figure 10. A complete input file is transferred and the problem is solved with implicit integration of the equations of motion in NIKE3D.
CONCLUSIONS

An example of an integrated environment for product design and non-linear finite element simulations is presented. The integrated environment consist of commercial and public domain codes linked together with use of a relational database, a data model for finite element models, a data model for material descriptions and communication programs. The program system is operated with use of a window based user interface.
The use of integration in the product development process has a substantial significance for the productivity. In the case of numerical simulations, the analysis models can be defined in one single system independent of which finite element program is going to be used. Traditionally, the geometry of a product need to be redefined for each analysis in different programs. In the integrated environment, the geometry is taken from the product definition created by a solid modeller. In addition to the increased productivity and ease of use, it is easy to expand the system with new programs. The concept of a neutral database decreases the number of interfaces in the system. The exchange of programs is facilitated, e.g. if a new CIM-system is selected, only one interface need to be altered. Furthermore, the relational database facilitates the search for relevant data for the selected finite element program. The use of the structured query language for technical database applications makes it straightforward to integrate the product development system with systems for economy and administration.

The future development will include standard protocols such as PDES/STEP\textsuperscript{17} both for data transfer and data storage. There will also be a development of a more refined material database with separate definitions of material behaviour and parameters for material models. Interfaces for alternative CIM-systems, finite element codes and result presentation programs will be developed. Most of the future development will take place in research projects where the system and the analysis programs are tuned for use in simulations of specific manufacturing processes such as metal powder pressing, hot isostatic pressing, quenching or welding.

ACKNOWLEDGEMENT

The financial support obtained from the CIM-Institute at Luleå University of Technology is gratefully acknowledged.

REFERENCES

INTEGRATION VIA A RELATIONAL DATABASE

2 I-DEAS. Structural Dynamics Research Corporation. 200 Eastman Drive. Milford, Ohio 45150-2789, USA
3 INFORMIX. Informix Software Inc. 4100 Bohannon Drive. Menlo Park, CA 94025, USA
16 Jeppsson, P. and Svoboda, A. Integrated design and verification system for finite element modelling. to be published in Int. J. of Concurrent Eng.: Research and Applications (CERA)
EXPERIMENTS WITH ONE-SIDED SPRAY-COOLING OF STEEL PLATES FOR EVALUATION OF THERMO-MECHANICAL MATERIAL MODELS

M. Oldenburg, L. Sandberg*, G. Bergman and M. Jonsson*

Department of Mechanical Engineering, Luleå University of Technology
S-971 87 Luleå, Sweden

* Plannja HardTech AB
S-971 88 Luleå, Sweden

Abstract

The quenching process of thin plates is studied. In many industrial applications of quenching, there is a need for simple tests for evaluation of analysis methods and material models for simulations. The presented study is part of a program for the simulation of the combined pressing and quenching process of thin-walled structures. This type of manufacturing process is used in e.g. the production of side-impact protection beams for cars. By the use of an experiment with one-sided spray-cooling of a thin strip, large deformations can be studied and followed in time. The prediction of the deformation of the strip can be regarded as a one-dimensional problem where the curvature of the strip is sought. The thermo-mechanical behaviour of the material can be evaluated throughout the time-history of the test. In order to match the deformation of the plate, the material model must predict the temperature distribution through the thickness of the plate in each point in time, the phase transformations taking place in each layer of the thin plate as well as the stress and strain components in each layer. Experiments have been performed with a material without phase transformations in the actual temperature and time scale and with a material with only martensitic transformations. Analyses with a thermo-elastic-plastic material model with transformation plasticity included have been performed.

This study is initiated by the demand from the industrial application of simultaneous quenching and sheet metal forming. This process is used for forming products with low weight and very high strength for the car industry. The parts are in general used for functions related to the safety of the passengers, e.g. side impact protection beams. The sheet metal material is a steel alloy with very high strength after quenching. If the forming takes place before the quenching process with standard sheet metal forming methods, there will be considerable geometry changes due to residual stresses from the forming process combined with the dilatation due to phase changes during quenching. On the other hand, if the quenching takes place before forming, there will be considerable springback that is difficult to predict due to the high strength material. In addition, subsequent heat treatment would cause further distortions. In order to avoid most of the described problems, the process of simultaneous forming and quenching is used for low weight, high strength sheet metal products for the car industry.

The quenching of steel has previously been subjected to intensive studies. Most research conducted in this area have concerned the quenching of massive steel components (Sjöström 1982), (Hasselquist 1993). In the case of massive components, the structural effects of quenching are residual stresses. However, the corresponding deformations are usually small.

In order to decrease the lead times during product development and to avoid part of the costly prototype manufacturing and testing of new products, the material behaviour with respect to the phase changes and structural response need to be studied. The aim of the studies is to obtain sufficient material and process characteristics to be able to analyse and simulate the combined quenching and pressing process with the actual material.

In the present work, experiments with quenching of thin plates are studied. In order to use simulations in the industrial product development process, there is a need for low cost, uncomplicated and reliable verification methods for the material data that are to be used in simulations. The structural effects of heat treatment are usually residual stresses and/or permanent deformations in the component. In the quenching process, usually only the final state can be evaluated by comparison of results from analyses and experiments, e.g. (Argyris et al. 1985), (Rammerstorfer et al. 1983). This is due to the difficulties involved in measuring the stresses with respect to time during the process. Furthermore, the precision of residual stress measurements are not always sufficient for the verification of material characteristics and complex material models for thermo-mechanical analyses.
The purpose of this study is to evaluate an experiment for verification of material data and material models intended for the simulation of quenching. The experiment is a setup with one-sided spray-cooling of a thin strip. The deformation history of the strip is measured during the cooling time. The temperature of the specimen is continuously measured on each side as well as in the middle of the specimen. The evaluation of the deformation is dependent on that the spray-cooling is equally distributed over the plate in order to obtain constant curvature of the thin strip.

The reasons for using this method are that deformation is usually measured with higher accuracy than stresses and that the deformation can be used for evaluation of the micromechanical and structural changes in the material during the time history of the test. Martensitic hardening can be studied by appropriate choice of material type and cooling time. The cooling time is dependent on the thickness of the strip and the water flux. Time dependent transformations can be studied if the cooling time is long enough for the selected material.

The process can be analyzed with one dimensional theory both for the non-linear transient thermal analysis and the non-linear structural analysis. Thus, special analysis programs for evaluation of the experiment can be designed for investigations of material characteristics and material models.

**Experimental Setup**

The experimental setup consists of a furnace for heating the specimens to approximately 850 °C and a special housing for the spray nozzles with specimen support at positions 25 mm and 305 mm from the lower edge and a deformation measurement device, see Figure 1. The geometry of the strips used in this work is 360 x 32 x 10 mm.

The temperatures at the cooled side and in middle of the specimen are measured with thermocouple wires which are installed in specially designed probes. The probes are inserted into the plate according to Figure 2. Thus, the measurement of the temperature on the cooling side is done 0.1 mm into the strip. The reason for using this method is that the temperature measurement with thermocouples installed directly on the surface is disturbed by the cooling spray. The measured temperature with use of surface mounted thermocouples is effectively the water spray temperature.

The probes are manufactured in the same material as the studied material in order to minimize the disturbance of the temperature field by the probes. The measurement equipment consists of a thermal measurement device, Analog Devices 3B47, an RTI 800/815 A/D-converter card and a computer for sampling and storage. The displacement at a position 159 mm from the lower edge of the strip is measured with a DCT/2000/A displacement transducer from the RDP-group, see Figure 3. The air pressure and the water temperature in the spray-cooling equipment are also measured. In order to minimize the thermal flow at the supports of the plate, the plate is simply supported by aluminium oxide bars. The housing is designed to minimize the presence of water spray on the back side of the plate, in order to obtain true one-sided cooling.

**Spray-cooling Experiments**

The experiment setup and measurement equipment is validated with use of a specimen material which does not change phases during the cooling process. For this purpose, strips produced from Inconel 600 is used. The measured thermal history of the sprayed side of the specimen is used as a prescribed temperature function on the cooled surface in an analysis of the heat transfer in the plate. The temperature field through the thickness of the plate is then used as thermal loading in the
subsequent structural analysis. The strip of Inconel 600 does not show any permanent deformation due to the cooling process. The thermal history and deformation history of the strip obtained from measurements are shown in Figure 9 and Figure 10, respectively.

The purpose of the experiments is to be able to validate material models and material parameters of materials used in industrial quenching processes. Initially, a material that show only martensitic transformations during the quenching process is used. The material used is SS2550-02, equivalent to the steel referred to as 60 NCD 11 steel in Denis et al. (1982). An experiment with the same geometry as the Inconel 600 strip is performed with the SS2550-02 material. The thermal history and the deformation history of the strip are shown in Figure 14 and Figure 15. The SS2550-02 material shows permanent deformations as a result of the one-sided cooling of the strip. By using an adequate material model and appropriate material parameters, the deformation history and the resulting permanent deformation should be possible to predict.

Analysis of Experiments

The quenching process is analysed with non-linear finite element codes for thermal and structural analysis. The used codes are the public domain programs TOPAZ2D (Shapiro and Edwards 1990) and NIKE2D (Engelmann and Hallquist 1991). TOPAZ2D is used for non-linear heat transfer analyses of two-dimensional and axi-symmetric problems. It permits material parameters such as conductivity, specific heat and convection coefficient to be dependent of the temperature. NIKE2D is an implicit code for non-linear, two-dimensional or axi-symmetric conditions. Problems with non-linear material behaviour, large deformations and contacts can be analysed. The thermal and structural analyses are uncoupled, i.e. the complete temperature history is calculated in TOPAZ2D before it is used as thermal loading in NIKE2D. The used finite element codes are part of an integrated CIM-environment (Bergman et al. 1995).

Analysis of Inconel 600 Experiments. Since the experiment can be idealized as a one dimensional heat transfer problem, in principle, only one row of four-node elements is needed in the TOPAZ2D analysis. Sixteen elements are used through the thickness of the 10 mm thick strip. The material data used in the thermal analysis is shown in Figure 4. It can be shown that the deformation of a specimen that does not show any transformations in the actual temperature interval and remain elastic during the test, is strongly correlated to the heat flux on the cooled side. Beam theory, the Fourier law and an assumption of the temperature field gives a relation between heat flux and deformation. If, for simplicity, a linearly varying temperature field is assumed through the thickness of the strip, the relation between the heat flux as a function of time, $f(t)$, and the deformation of the specimen as a function of time, $\delta(t)$, is

$$\delta(t) = \frac{\alpha(T) L^2}{8k(T)} \cdot f(t)$$

where $\alpha(T)$ and $k(T)$ are the thermal expansion coefficient and the conductivity as functions of the temperature, respectively,
and $L$ is the length of the specimen. The calculated heat flux as a function of time is shown in Figure 5. This heat flux history is based on the calculated temperature field obtained from an analysis using the measured cooled side temperature as input.

![Figure 5. Calculated heat flux based on the cooled Inconel 600 surface as a function of time.](image)

Figure 6 is high when the temperature on the surface is below 350 °C. However, the region with high convection coefficient appears at lower temperatures than obtained by using the function from Hodgson et al. (1992), see Figure 6.

![Figure 6. Calculated convection coefficient function and function proposed by Hodgson et al. (1992).](image)

The deformation of the Inconel 600 strip is almost proportional to the heat flux on the sprayed side. The factor between the heat flux and the deformation may vary linearly with the temperature since the conductivity is almost linear with respect to the temperature for the Inconel 600 material. The convection coefficient is shown together with the function proposed by Hodgson et al. (1992) for spray-cooling with approximately the same water and air pressure as in the present experiment. The water flux is estimated to be 4.1 litre/(s m²). The resulting convection coefficient from the spray-cooling experiment shown in

![Figure 7. Inconel 600 thermal expansion function (Näsström et al. 1992).](image)

![Figure 8. Structural material data used in Inconel 600 analyses (Näsström et al. 1992).](image)

The temperature field from the heat flow analysis is used as thermal loading in the structural analysis. The structural material data for the Inconel 600 material is shown in Figure 7 and 8 (Näsström et al. 1992). The resulting deformation together with experimental results are shown in Figure 10.

**Analysis of the SS2550-02 Experiments.** The thickness of the SS2550-02 specimen is 10 mm as was the case for the Inconel 600 strip. The finite element model used in the analysis is the same as in the analysis described above. The temperature dependent conductivity coefficient and specific heat coefficient are shown in Figure 11. These functions are derived from the fraction of martensite and austenite with respect to the temperature (Koistinen and Marburger 1959). The conductivity and
Figure 9. Temperatures obtained from experiment and from analysis. Inconel 600.

Figure 10. Deformation obtained from experiment and analysis. Inconel 600.

specific heat functions used for martensite and austenite, respectively, are found in Sjöström (1982). The total latent heat from the transformation from austenite to martensite is derived from measurements of the total energy in a specimen of SS2550-02 at 850 °C (Bergman and Oldenburg 1996). The latent heat as a function of temperature is then derived from the fraction of martensite with respect to temperature. The latent heat function is then included in the function for the specific heat coefficient. A linear isotropic-kinematic hardening thermo-ela-sto-plastic material model is used in the deformation analysis. In one of the analyses, transformation plasticity is included in the model (Bergman and Oldenburg 1996). The thermal expansion function used is shown in Figure 12 (Denis et al. 1982) and includes the volumetric change caused by the martensitic transformation. The yield stress and the hardening modulus with respect to the temperature are shown in Figure 13 (Denis et al. 1982). Young’s modulus and Poisson’s ratio are

Figure 11. Thermal material data used in the analyses for the SS2550-02 material.

Figure 12. Thermal expansion function used in the SS2550-02 analysis (Denis et al. 1982).

Figure 13. Yield stress and hardening modulus functions used in the SS2550-02 analysis (Denis et al. 1982).
considered to be constant, equal to 145.5 GPa and 0.3, respectively. The temperature histories from experiment and analysis are shown in Figure 14. The resulting deformation in the middle of the plate obtained from experiment and from analysis

Figure 14. Measured and calculated temperature histories from the experiment with a 10 mm thick SS2550-02 plate.

have been used to establish a function for the convection coefficient which is valid for this experimental setup.

The deviation between the calculated and the measured deformation in the Inconel 600 experiment may be due to errors in the temperature or deformation measurements. There may also be errors in the material data used in the calculations. However, it is believed that uneven distribution of the water spray over the plate is the major cause to the observed deviations in figure 10 and this will be considered in future experiments.

The results from the analyses of the SS2550-02 plate confirm the interpretation of the results from the previous experiment. The experiment seems to be very sensitive to an uneven distribution of the cooling spray, causing the phase transformations to occur in different moments of time over the plane of the specimen. Since the measured deformation is a sum of the response of every point in the plate, the large amplitude in the analysis result is not captured. However, since the phase transformation in this case is not time dependent, this effect will not have any effect in the end of the quenching process. The calculated deformation history with transformation plasticity included, supports this interpretation of the experimental results.

With some enhancements concerning the uniformity of spray-cooling over the surface of the plate, the presented experiments with one-sided spray-cooling of steel strips can be used as a relatively simple method of verifying material data and models both for the thermal and the deformation process of quenching. The history of the deformation can be measured more easily than the history of stresses during the cooling process. The measured deformations are also expected to be more accurate than measurements of residual stresses. In order to predict the deformation of the strip, the phase changes and the dilatation as well as the plastic deformation throughout the thickness of the strip need to be modelled accurately. So far, one material without phase changes and one with only martensitic transformations have been tested and analysed. However, the method will be very useful in the verification of models for time-dependent transformations as well. Different cooling histories can be obtained by changing the water flux or the thickness of the plates used in the experiment.

**Discussion and Conclusions**

The analysis of the Inconel 600 experiment has shown almost direct proportionality between the heat flux on the spray-cooled side of the specimen and the deformation of the strip. The correlation between heat flux and deformation is due to the elastic behaviour of the material and the relatively small deformations. The measured temperature function and the calculated heat flux function on the cooled side of the specimen

Figure 15. Measured and calculated deformation histories from the experiment with a 10 mm thick SS2550-02 plate.

with the calculated temperature field as thermal loading are shown in Figure 15.

**References**


A computational model for quenching simulations of thin plates has been developed. The model is examined by comparisons with experiments with one-sided water spray cooling. With this experiment, the thermomechanical behavior of the material can be monitored throughout the time history of the test. Experiments have been performed with a material that undergoes only martensitic transformation during quenching. For the conditions examined, the plate exhibits permanent deformation after quenching. In the stress calculation, transformation plasticity is included in the effective-stress-function (ESF) algorithm as an additional strain component related to the stress and to the progress of transformation. Analyses of the present experiments show that the permanent deformation of the plate is exclusively due to transformation plasticity.

The study presented here is part of a research program for the simulation of combined quenching and sheet metal forming of thin-walled structures. This manufacturing process is used for the production of components with low weight and high strength, for example, side-impact protection beams for cars. The performance of the components is greatly influenced by the microstructure, hardness, and residual stresses developed during the process. Industrial objectives of the studies are to obtain material and process characteristics in order to achieve good quality in the manufacturing process and service behavior. Using finite element simulations, part of the costly prototype manufacturing and testing of new products can be avoided. A finite element simulation of the complete quenching and pressing process requires accurate predictions of temperature distribution, microstructure evolution, stresses and strains, as well as the thermomechanical contact between the tool and the sheet metal. There is also a need for uncomplicated and reliable verification methods for the computational models and material data used. Furthermore, the experimental methods should, if possible, monitor the process in time.

This work addresses the modeling of the material behavior during quenching and comparisons with experimental results. The fact that the structural transformation influences the mechanical material properties is well known. The importance
of considering phase transformation effects in modeling heat treatment processes has been shown by a number of authors, including Rammerstorfer et al. [1], Denis et al. [2, 3], and Sjöström [4]. Most research conducted on quenching placed a particular emphasis on predicting residual stresses in massive components. However, the deformation from the thermal loading is usually small. The results from these simulations are usually compared with the residual stresses determined by experiments [5]. Due to difficulties involved in measuring stresses and strains with respect to time, usually only the final state can be evaluated by a comparison of the results from analyses and experiments. In this work, an experiment with one-sided water spray cooling of thin plates is used to verify the analysis. This experiment includes most of the phenomena that are present in the industrial quenching process such as large deformation, plastic deformation, and transformation-induced plasticity. Using an experiment with one-sided spray cooling of steel plates, the thermomechanical behavior of the material can be monitored throughout the time history of the test. Using the deformation history instead of stresses as a measure of the material response, verification of the material model is not restricted to the final state of the experiment.

**SPRAY COOLING EXPERIMENTS**

The experiment that is our subject for analysis is a setup with one-sided water spray cooling of a thin strip; see Figure 1. The experimental setup is a housing consisting of water spray nozzles and measurement devices for temperature and deformation in which the test specimen, after austenitization, is subjected to one-sided spray cooling. During cooling from approximately 850°C to 20°C, temperatures at the water-cooled side and in the middle of the specimen are measured with thermocouples. The measured temperature history at the water-cooled side is
used for boundary conditions in the thermal analysis. The deformation history is measured at a distance of 159 mm from the lower edge of the plate. The evaluation of the deformation is dependent on the fact that the water spray is equally distributed in order to obtain a constant curvature of the plate. The distance from the lower edge to the lower support is 25 mm; the distance between the supports is 280 mm. A more extensive description of the experiments can be found in Oldenberg et al. [6].

**COMPUTATIONAL MODEL**

The simulation of quenching consists of two parts, the temperature analysis and the mechanical analysis based on the thermal loads generated from the temperature analysis. Temperature-dependent material properties are considered in the model. For the nonlinear thermal and mechanical analyses, TOPAZ2D [7] and NIKE2D [8] are used, respectively. The thermomechanical analysis is uncoupled; that is, the complete temperature history is calculated in TOPAZ2D before it is used as thermal loads in NIKE2D. The used finite element codes are part of an integrated CIM-environment in which nonlinear finite element codes are integrated with a CAD-system via a relational database; see Bergman et al. [9].

Plates with a thickness of 4.5 mm and 10 mm are analyzed. The height and depth of the plates are 360 mm and 32 mm, respectively. Assuming that heat flow occurs only in the thickness direction of the plate, the thermal analyses can be performed with a one-dimensional model. A one-dimensional model can be used in the mechanical analyses, as well, if special constraints are applied. However, for practical reasons, a two-dimensional model is used both for the thermal and mechanical analyses since the calculations are not time critical. The finite element models of the 4.5-mm and 10-mm specimen have 8 and 16 four-node elements through the thickness, respectively. Both models have 300 elements in the transverse direction.

**Thermal Analysis**

The boundary conditions considered are water spray cooling and natural air cooling. The water spray cooling can be modeled in a number of ways, such as by prescribing the nodal temperatures at the surface or by convection heat transfer between the surface and the quenching medium. The use of the surface heat transfer coefficient as a boundary condition is only applicable in an industrial manufacturing process after careful investigations of the heat transfer conditions in each special case. In the present work, a prescribed surface temperature according to a measured temperature history is used. The natural air cooling consists of both radiative and convective components. Values for free convection are obtained from standard formulas for vertical plates. The surface emissivity is assumed to be 0.8. The surrounding temperature is 20°C. The time step size used in the thermal analysis is 0.05 sec.
Thermal Material Properties

The plate material is SS 142550-02, which is equivalent to the tool steel referred to as 60 NCD 11 in Denis et al. [2]. The chemical composition of the steel is 0.55% C, 0.3% Si, 0.4% Mn, 1.3% Cr, 0.3% Mo, and 3.0% Ni. This steel undergoes only martensitic transformation during the actual quenching process, with a martensite start temperature of 247°C. The absence of thermal material data for this specific material imposed the use of a linear mixture rule in estimating thermal material relations. The single phase data for austenite and martensite are obtained from Sjöström [4]. These data originate from measurements on British Standard Steel En 352. Although the data from [4] are valid for a material with a different chemical composition, it is considered to be the best set of thermal material data available. As shown in Figures 2 and 3, both the thermal conductivity $\lambda$ and the heat capacity $C$ for each phase vary with the temperature. The resulting material properties are assumed to be a linear combination of the corresponding properties of each phase such that

$$\lambda(T) = V_A \lambda_A(T) + V_M \lambda_M(T)$$  \hspace{1cm} (1)$$

and

$$C(T) = V_A C_A(T) + V_M C_M(T)$$  \hspace{1cm} (2)$$

where $V_A$ and $V_M$ are fractions of austenite and martensite. The volume fraction of martensite is given by Koistinen–Marburger’s [10] equation

$$V_M = 1 - \exp(-0.011(M_s - T))$$  \hspace{1cm} (3)$$

Figure 2. Thermal conductivity of austenite and martensite as a function of temperature.
where $V_M$ is the volume fraction of martensite, $M_s$ is the martensite start temperature, and $T$ is the temperature. The mixture law does not account for latent heat release due to phase transformation, which is simulated by a modified temperature dependence of the heat capacity. After austenitization at $850^\circ$C for 30 min, a small specimen of the steel was cooled in water in order to determine the amount of heat released. The increase in temperature of the water was used as a measure of the heat content of the material. The difference between measured heat content and the enthalphy given by the mixture rule corresponds to latent heat released due to transformation. The latent heat from austenite to martensite was estimated to be $58.5$ kJ/kg. The distribution of latent heat over the temperature interval where the transformation occurs depends on the rate of transformation that is given by Eq. (3). The calculated thermal material relations are shown in Figure 4.

**Mechanical Analysis**

During quenching the material may undergo plastic deformation due to thermal stresses arising from variations of the temperature and variations of the phase proportions. A thermo-elastic-plastic material model in NIKE2D is used for the mechanical analysis. The material model has been modified to use the ESF algorithm for the stress calculations together with a mixed linear isotropic-kinematic hardening rule. Transformation plasticity is also implemented in the model. Plane stress conditions are assumed in the analyses of the one-sided spray cooling experiment. The time step size used in the mechanical analysis is 0.10 sec.
Transformation Plasticity

Transformation plasticity is a phenomenon where a material undergoing phase transformation can exhibit permanent deformation even if the applied stress is much lower than the yield stress. The additional plastic strain during phase transformation is the result of two mechanisms. The first is known as the Greenwood–Johnson [11] mechanism. The volume difference between the phases generates internal stresses that are large enough to cause plastic deformation in the weaker phase. The interpretation of the second mechanism is due to Magee [12] and occurs in martensitic transformations. The formation of martensite occurs in a preferred orientation that affects the global shape of the body.

A model proposed by Leblond et al. [13] for the transformation plasticity strain rate that we use in this work is

\[
\dot{\varepsilon}_{ij}^{trp} = -\frac{\Delta V}{V} \frac{1}{\sigma_y} (\mathbf{S}_{ij} - \mathbf{a}_{ij}) h \left( \frac{\bar{\sigma}}{\sigma_y^g} \right) \ln(z) \dot{z}
\]  

(4)

where \( \Delta V/V \) is the relative volume change associated with the transformation, \( \sigma_y \) is the yield stress of the weaker phase, \( \mathbf{S}_{ij} \) is the deviatoric stress tensor, \( \mathbf{a}_{ij} \) is the center of the yield surface, \( h \) is a function describing the nonlinear dependence on large applied stresses, \( z \) is a fraction of formed phase, and \( \dot{z} \) is the rate of transformation. The nonlinear dependence on large applied stresses starts at \( \bar{\sigma}/\sigma_y^g \geq 0.5 \) and has the form

\[
h \left( \frac{\bar{\sigma}}{\sigma_y^g} \right) = 1 + 3.5 \left( \frac{\bar{\sigma}}{\sigma_y^g} - \frac{1}{2} \right)
\]  

(5)
where $\bar{\sigma}$ is the effective stress and $\sigma^g_y$ is the global yield stress. In Eq. (4) it is assumed that the weaker phase (austenite) behaves according to an ideally plastic material. If the hardening parameters for the individual phases are known, hardening of the austenite can be taken into consideration when using the model proposed by Leblond [14]. The back stress tensor in Eq. (4), currently defined in Eq. (17), will then be the back stress tensor for the weaker phase. The incremental form of the transformation plasticity strain is obtained by integrating Eq. (4), assuming that the stress is constant, as

$$\Delta \varepsilon_{ij}^{\text{trp}} = t^+ \Delta t \rho \left[ t^+ \Delta t \frac{\bar{\sigma}}{\sigma^p} \right] \left[ t^+ \Delta t S_{ij} - t^+ \Delta t \alpha_{ij} \right]$$

where

$$t^+ \Delta \rho = \frac{\Delta V}{V} \frac{1}{\sigma_y} \left( t^+ \Delta t z \left( 1 - \ln \left( t^+ \Delta t z \right) \right) - t^* \left( 1 - \ln \left( t^* z \right) \right) \right)$$

and

$$\sigma^g_y = t^+ \Delta t \sigma_{yv} + \beta t^+ \Delta t H' \varepsilon_p$$

The global yield stress $\sigma^g_y$ is a function of the virgin yield stress $t^+ \Delta t \sigma_{yv}$ and the plastic modulus $t^+ \Delta t H'$ at temperature $t^+ \Delta \theta$ as well as the known effective plastic strain $t^* \varepsilon_p$. For a mixed isotropic-kinematic hardening rule, the weight factor $\beta$ can have values between 0 (purely kinematic hardening) and 1 (purely isotropic hardening).

Since the material studied undergoes only martensitic transformation during cooling, the decomposition of austenite into martensite can be described by Eq. (3). In the case of diffusional transformations such as austenite decomposition into ferrite, pearlite, and bainite, the microstructural evolution may be modeled using the method proposed by Kirkaldy et al. [15].

**Stress Calculations**

The thermo-elastic-plastic constitutive model uses the ESF algorithm proposed by Kojic et al. [16]. The ideal of the ESF algorithm is that the constitutive behavior is written in terms of one variable, namely, the effective stress. The solution of the unknown stress state is reduced to the solution of a single equation, the ESF equation. In this work, the ESF algorithm has been modified to include transformation plasticity as an additional strain component related to the stress and to the progress of transformation. This development is based on the ESF implementation of Oddy [17]. A description of the algorithm used in the current case is given below.

The total incremental strain tensor $\Delta \varepsilon_{ij}^{\text{tot}}$ is assumed to be composed of elastic $\Delta \varepsilon_{ij}^e$, plastic $\Delta \varepsilon_{ij}^p$, thermal $\Delta \varepsilon_{ij}^{\text{th}}$, and transformation plasticity $\Delta \varepsilon_{ij}^{\text{trp}}$ strain increments such that

$$\Delta \varepsilon_{ij}^{\text{tot}} = \Delta \varepsilon_{ij}^e + \Delta \varepsilon_{ij}^p + \Delta \varepsilon_{ij}^{\text{th}} + \Delta \varepsilon_{ij}^{\text{trp}}$$
The deviatoric and hydrostatic stress state, respectively, corresponding to time \( t + \Delta t \) can be written as

\[
^{t+\Delta t}S_{ij} = 2^{t+\Delta t}G \left( e_{ij}^e + \Delta e_{ij} - \Delta e_{ij}^p - \Delta e_{ij}^{trp} \right)
\]

and

\[
^{t+\Delta t}\sigma_m = 3^{t+\Delta t}K \left( \frac{\sigma_m}{3K} + \Delta e_m - \Delta e^h \right)
\]

where for time \( t \) and \( t + \Delta t \), \( ^{t+\Delta t}G \) is the shear modulus, \( ^{t+\Delta t}S_{ij} = 2^{t+\Delta t}G e_{ij}^e \) is the elastic deviatoric strain, \( \Delta e_{ij} = \Delta e_{ij} - \Delta e_m \delta_{ij} \) is the deviatoric strain increment, \( \Delta e_{ij}^p \) is the plastic strain increment, \( \Delta e_{ij}^{tr} \) is the transformation plasticity strain increment, \( ^{t+\Delta t}K \) and \( ^{t+\Delta t}K \) is the bulk modulus, \( ^t\sigma_m = ^t\sigma_{ii}/3 \) is the hydrostatic stress, \( \Delta e_m = \Delta e_{ii}/3 \) is the mean strain increment, and \( \Delta e^h \) is the thermal strain increment. The thermal strain increment is obtained using the relation

\[
\Delta e^h_{ij} = \Delta e^h S_{ij}
\]

where \( \Delta e^h \) is the increment in thermal dilatation, assumed to include both thermal expansion and volume change due to phase transformation.

Using von Mises’s yield condition and an associated flow rule, the incremental plastic strain tensor becomes

\[
\Delta e_{ij}^p = \Delta \lambda \left( ^{t+\Delta t}S_{ij} - ^{t+\Delta t}\alpha_{ij} \right)
\]

where \( \Delta \lambda \) is a scalar to be determined. Equation (13) can be written as

\[
\Delta e_{ij}^p = \Delta \lambda ^{t+\Delta t}A_{ij}
\]

where

\[
^{t+\Delta t}A_{ij} = ^{t+\Delta t}S_{ij} - ^{t+\Delta t}\alpha_{ij}
\]

is the radius of the von Mises yield surface. An expression for the effective plastic strain increment is obtained by taking the scalar product of both sides of Eq. (14)

\[
\Delta e^p = \left( \frac{2}{3} \Delta e_{ij}^p \Delta e_{ij}^p \right)^{1/2} = \sqrt{\frac{2}{3}} \Delta \lambda ^{t+\Delta t}A_{ij}
\]

For a mixed isotropic-kinematic hardening rule, the center of the yield surface is given by Prager’s rule

\[
^{t+\Delta t}\alpha_{ij} = ^{t+\Delta t}C ^{t+\Delta t}e_{ij}^p = ^{t+\Delta t}C \left( e_{ij}^p + \Delta e_{ij}^p \right) = ^t\alpha_{ij} + ^{t+\Delta t}C \Delta e_{ij}^p
\]

where

\[
^{t+\Delta t}C = \frac{2}{3} (1 - \beta) ^{t+\Delta t}H
\]
Using Eqs. (14), (15), and (17), the deviatoric stress tensor can be written as

\[ ^{t+\Delta t}S_{ij} = (1 + ^{t+\Delta t}C\Delta \lambda) ^{t+\Delta t}A_{ij} + ^{t+\Delta t}\alpha_{ij} \] (19)

Using Eqs. (6), (10), (14), and (19), \(^{t+\Delta t}A_{ij}\) can be written in the form

\[ ^{t+\Delta t}A_{ij} = g_{ij} \left/ \left( \frac{1}{2^{t+\Delta t}G} (1 + ^{t+\Delta t}C\Delta \lambda) + \Delta \lambda + ^{t+\Delta t}\phi_h \left( \frac{^{t+\Delta t}\sigma}{\sigma_y} \right) \right) \right. \] (20)

where

\[ g_{ij} = ^{t+\Delta t}e_{ij} + \Delta e_{ij} - \frac{1}{2^{t+\Delta t}G} ^{t+\Delta t}\alpha_{ij} \] (21)

The radius of the yield surface is

\[ R = \sqrt{\frac{2}{3}} \left( ^{t+\Delta t}\sigma_y + \beta ^{t+\Delta t}H^{t+\Delta t}\bar{\varepsilon}_p \right) \] (22)

which is a function of the virgin yield stress \(^{t+\Delta t}\sigma_y\) and the plastic modulus \(^{t+\Delta t}H\) at temperature \(^{t+\Delta t}\theta\) as well as the effective plastic strain at the end of the time step \(^{t+\Delta t}\bar{\varepsilon}_p\). Rewriting Eq. (22) with the use of Eq. (16) gives

\[ R = \sqrt{\frac{2}{3}} \left( ^{t+\Delta t}\sigma_y + \beta ^{t+\Delta t}H^{t+\Delta t}\bar{\varepsilon}_p + \beta ^{t+\Delta t}H^{t+\Delta t}\sqrt{\frac{2}{3}} \Delta \lambda^{|^{t+\Delta t}A_{ij}|} \right) \] (23)

Requiring that the final stress state must lie on the yield surface, the relation between the magnitude of \(^{t+\Delta t}A_{ij}\) and the radius of the yield surface \(R\) is

\[ R = |^{t+\Delta t}A_{ij}| = \left( ^{t+\Delta t}A_{ij} ^{t+\Delta t}A_{ij} \right)^{1/2} \] (24)

To determine \(\Delta \lambda\), we take the scalar product of both sides of Eq. (20), which together with Eqs. (23) and (24), gives

\[ \Delta \lambda = \left( \|g\| - R^* \left( \frac{1}{2^{t+\Delta t}G} + ^{t+\Delta t}\phi_h \left( \frac{^{t+\Delta t}\sigma}{\sigma_y} \right) \right) \right) \cdot \left( R^* \left( \frac{^{t+\Delta t}C}{2^{t+\Delta t}G} \right) + \frac{2}{3} \|g\| \beta ^{t+\Delta t}H^{t+\Delta t}\bar{\varepsilon}_p \right)^{-1} \] (25)

where

\[ \|g\| = (g_{ij} g_{ij})^{1/2} \quad \text{and} \quad R^* = \sqrt{\frac{2}{3}} \left( ^{t+\Delta t}\sigma_y + \beta ^{t+\Delta t}H^{t+\Delta t}\bar{\varepsilon}_p \right) \] (26)
The ESF is obtained by taking the scalar product of both sides of Eq. (10)

\[ f(t^+\Delta t \sigma) = \frac{1}{2^+\Delta t E} t^+\Delta t \sigma^2 - 3^+\Delta t p_{ij} = 0 \]  

where

\[ p_{ij} = ^e_e e_{ij} + \Delta e_{ij} - \Delta \lambda ^+\Delta t A_{ij} - ^+\Delta t \phi h \left( \frac{t^+\Delta t \sigma}{\sigma_y} \right)^t + \Delta t A_{ij} \]  

Because of the implicit form of Eq. (27), an iterative solution must be applied. A secant algorithm is used to solve the ESF. To get started, two initial guesses for the effective stress must be generated. For this purpose the initial stress and the elastic predictor stress is used, that is,

\[ ^t + \Delta t \sigma^{(1)} = \left( \frac{3}{2} S_{ij}^{tr} t^i S_{ij}^{tr} \right)^{1/2} \] and \[ ^t + \Delta t \sigma^{(2)} = \left( \frac{3}{2} S_{ij}^{tr} t^i S_{ij}^{tr} \right)^{1/2} \]  

where

\[ S_{ij}^{tr} = S_{ij} + \frac{t^+\Delta t G}{t G} + 2t^+\Delta t G \Delta e_{ij} \]  

The convergence criterion used is based on the virgin yield stress, that is,

\[ |f(t^+\Delta t \sigma)| \leq \left( \frac{0.001 t^+\Delta t \sigma_y}{t^+\Delta t E} \right)^2 \]  

where \( t^+\Delta t E \) is Young’s modulus.

The computational procedure consists of the following steps.

1. Initial guesses for the effective stress \( ^t + \Delta t \sigma^{(1)} \) and \( ^t + \Delta t \sigma^{(2)} \). Then for \( k = 1, 2, \ldots \)
2. Compute \( ^t + \Delta t \phi^{(k)} \) and \( h(^t + \Delta t \sigma^{(k)}/\sigma_y^{(k)}) \).
3. Compute \( ^t + \Delta t \lambda^{(k)} \).
4. Calculate the value of the ESF \( f(t^+\Delta t \sigma^{(k)}) \). If the convergence criterion is satisfied, go to step 6.
5. Compute \( ^t + \Delta t \sigma^{(k+1)} \) using the secant algorithm (when \( k > 1 \)). Go to step 2.
6. Compute \( ^t + \Delta t \sigma_{ij}, \Delta e_{ij}^p, \) and \( \Delta \varepsilon^p \).

**Mechanical Material Properties**

To perform the thermomechanical structural analyses, the mechanical properties of the steel as a function of temperature and structural state must be known. The properties concerned include Young’s modulus, Poisson’s ratio, thermal dilatation, yield stress, and hardening modulus. The thermal dilatation is assumed to include
both thermal expansion and volume change due to phase transformation. Material properties shown in Figures 5 and 6 are obtained from Denis et al. [2]. The stress-strain description shown in Figure 5 is in the form of bilinear curves. Thus, a linear strain hardening law is used. However, a model with variable hardening is preferred if the corresponding material data are available. Young’s modulus and Poisson’s ratio are considered to be constant, equal to 145.5 GPa and 0.3, respectively. The density is 7800 kg/m³.

Besides the properties mentioned previously, the specific volume change $\Delta V/V$ and the yield stress of austenite $\sigma_y$ are required as inputs to the thermomechanical material model. In [2], the effects of transformation plasticity have been measured using a tensile test. A linear relationship between transformation plasticity and applied stress was observed. The experiment also shows a rise in the $M_s$ temperature when the applied tensile stress increases. A criterion for the variation of the $M_s$ temperature was established by Patel et al. [18]. They found that tensile and compressive stresses tend to raise $M_s$, although less by compressive stresses, while hydrostatic stresses lower $M_s$. The variation of the $M_s$ temperature is not taken into consideration in this investigation. Figure 7 shows the measured and calculated axial strain as functions of temperature for different applied tensile stresses. The calculations were carried out in order to match the experimentally obtained axial strain for an applied stress of 285 MPa. The value of $\left(\Delta V/V\right)\left(1/\sigma_y\right)$ was found to be $7.3 \times 10^{-11}$ Pa⁻¹. With this value the axial strain obtained for lower applied stresses agrees satisfactorily with the experimental values. In [3], values for $\Delta V/V$ and $\sigma_y$ are given as 0.03 and 285 MPa, respectively. Using these

![Figure 5](image)

**Figure 5.** Yield stress and hardening modulus as functions of temperature.
values and a linear relationship between transformation plasticity and applied stress, that is, not using Eq. (5), gives good agreement for the 285-MPa case. However, it leads to an overestimation of the axial strain for lower applied stresses. The value \( (\Delta V/V)(1/\sigma_y) = 7.3 \times 10^{-11} \text{ Pa}^{-1} \) is used in the present calculations.

Figure 6. Thermal expansion as a function of temperature.

Figure 7. Measured and calculated axial strain as functions of temperature for different applied tensile stresses.
RESULTS

The temperature histories from experiment and calculations for the 10-mm plate are shown in Figure 8. The maximal deviation between measured and calculated midpoint temperatures is approximately 75°C during cooling from 850°C to 300°C and approximately 40°C at lower temperatures. One source of the observed deviations may be the thermal material data used in the calculations. As mentioned earlier, the thermal material data are constructed from a material with a different chemical composition. However, it is believed that an uneven distribution of the water spray over the plate and an increased cooling of the back side of the plate at lower temperatures are the major causes of the observed deviations. The deformations of the 10-mm plate obtained from the experiment and from analysis with the calculated temperature field as thermal loading are shown in Figure 9. The differences observed can indicate that the measured temperatures are not representative for the whole plate, that is, assuming that the thermal material data are correct. When the water-cooled side undergoes martensitic transformation, at 55 sec, the deflection of the plate changes direction. Nonuniform cooling conditions and an increased cooling of the back side is believed to be the reason the large peak observed in the calculations is not present in the experiment. The calculated final state agrees well with the experiment when transformation plasticity is included. The difference in cooling conditions does not affect the calculated results at the end of the treatment since the martensitic transformation is time independent. No classical plastic strain is developed during the process. Thus, excluding transformation plasticity leads to a prediction of a final state with no permanent deformation.

Figure 8. Measured and calculated temperatures for the 10-mm plate.
Figure 9. Measured and calculated displacement for the 10-mm plate (trp = transformation plasticity).

Figure 10 shows experimental and calculated temperature histories for the 4.5-mm plate. Also in this case, there are differences between measured and calculated midpoint temperatures. The maximal deviation is approximately 80°C during cooling from 850°C to 400°C. The experimental midpoint temperature curve intersects the surface temperature curve at lower temperatures, probably due to an
uneven distribution of the water spray and the fact that the surface temperature and the midpoint temperature are measured at different points relative to the cooled surface. The deformation histories from experiment and calculations for the 4.5-mm plate are shown in Figure 11. The 4.5-mm plate shows a behavior similar to that of the 10-mm plate. An overestimation of the deformation in the elastic range is followed by a large peak when the plate undergoes martensitic transformation. Also in this case is the permanent deformation of the plate exclusively due to transformation plasticity.

CONCLUSIONS

A computational model for quenching simulations of thin plates and analyses compared with experiments are presented. A modification of the ESF algorithm is presented. The implementation includes a mixed isotropic-kinematic hardening rule and transformation plasticity. The ESF algorithm has proved to be an efficient method of solving the implicit equations of the thermo-elastic-plastic and transformation plasticity constitutive behavior of the material during the quenching process.

In the present configuration of the experiment, uniform cooling conditions were difficult to obtain. The measured temperature history is assumed to be correct at the point where it is measured but not representative for the whole plate at the same time. The results from the analyses of the quenching process suggest that, due to an uneven distribution of water spray, there is a phase difference in
the cooling process from point to point in the plane of the plate. Since the martensitic transformation is time independent, these problems do not affect the numerical results at the end of the quenching process. For the conditions examined, transformation plasticity must be taken into account in the calculation of internal stresses. The analyses of the present experiment show that the permanent deformation of the plates is exclusively due to transformation plasticity.

With some enhancements, the experiment with one-sided spray cooling of plates can be used as a relatively simple method of verifying material data and material models both for the thermal and the deformation process of quenching. The deformation history can be measured more easily than the history of stresses during the cooling process. Using deformations instead of stresses as measures of the material response, the verification of the material model is not restricted to the final state of the experiment. The measured deformations are also expected to be more accurate than measurements of residual stresses. The experiment can also be used to study diffusional transformations if the cooling time is long enough for the selected material. Different cooling histories can be obtained by changing the water flux or the thickness of the plates used in the experiment.

REFERENCES

17. A. S. Oddy, Research Associate, Carleton University, Ottawa, Canada, private communication, 1996.
A FINITE ELEMENT MODEL FOR THERMOMECHANICAL ANALYSIS
OF SHEET METAL FORMING

G. Bergman and M. Oldenburg

Department of Mechanical Engineering, Luleå University of Technology
SE-971 87 Luleå, Sweden

Abstract

A thermal model based on explicit time integration is developed and implemented into the explicit
finite element code DYNA3D to model simultaneous forming and quenching of thin-walled struc­
tures. A staggered approach is used for couphng the thermal and mechanical analysis, wherein each
analysis is performed with different time step sizes. The implementation includes a thermal “shell”
element with linear temperature approximation in the plane and quadratic in the thickness direction,
and contact heat transfer. The material behaviour is described by a temperature dependent elastic-
plastic model with a nonlinear isotropic hardening law. Transformation plasticity is included in the
model. Examples are presented to validate and evaluate the proposed model. The model is evaluated
by comparison with a one-sided forming and quenching experiment.

1 Introduction

The present study is initiated by the demand from the industrial application of
simultaneous sheet metal forming and quenching, also referred to as hot-stamping.
This process is used for manufacturing of low weight and high strength automotive
components. The components are in general used for functions related to the safety
of car passengers, e.g. side impact protection beams. The sheet metal is a steel alloy
with very high yield strength after quenching. If the forming takes place before the
quenching with standard sheet forming methods, there will be considerable
distortion due to residual stresses from the forming stage combined with the
dilatation during quenching. With this type of high strength steel, forming can not
take place after quenching since the formability is too limited in the normal case.
On the other hand, if forming can be done, there will be considerable springback
that is difficult to predict and to account for in the production process due to the high
strength material. In addition, subsequent heat treatment may cause further
distortion.
The aim of this work is to establish the methods needed to efficiently analyse the combined forming and quenching process with use of numerical methods. The resulting methods can be used early in the product development process to study tool design and process parameters. With early analyses, the manufacturing and testing of tool prototypes can be minimised and the lead time and cost for product development can be significantly reduced.

2 Requirements for analysis of hot-stamping

The hot-stamping manufacturing process is a thermomechanical forming operation. The modelling of the process requires a material description for the steel that includes plastic deformation with a nonlinear temperature dependent hardening law. The yield stress and elastic modulus also vary with temperature. The material model should describe the dilatation and also transformation plasticity during the hardening process. The process modelling involves transient heat transfer analysis with thermal contact. Before forming, the material is heated to the austenite region, approximately $850^\circ\text{C}$. During and after the forming stage, it is assumed that the cooling rate is high enough to obtain only martensitic transformation. Thus, the phase transformation can be modelled as time independent. The nonlinear thermal material behaviour includes heat capacity and thermal conductivity. The process also includes latent heat due to phase transformation. The modelling of the process must include a description of the temperature distribution in the thickness direction of the blank and consider mechanical and thermal contact between the tools and the blank.

In contrast to the extensive amount of published work concerning analyses of traditional sheet metal forming and quenching processes with the finite element method, see e.g. [1, 2], there is no work reported in the literature up to this date about the modelling and simulation of the combined forming and quenching of steel plates. Early sheet forming simulations using explicit time integration of the equations of motion are described in [3]. Comparisons with traditional implicit methods are made by several authors, see e.g. [4]. Examples of coupled thermomechanical analyses of different types of bulk forming operations can be found in e.g. [5-7]. Non-isothermal analyses of sheet metal forming where the temperature rise due to the plastic work is accounted for are described by several researchers, see e.g. [8, 9].

Related to this study, the quenching of steel plates using one-sided spray cooling is studied in [10] and the development and evaluation of the thermomechanical
material model used in the quenching simulations is described in [11].

3 Thermal model

The intended application requires a thermal model applicable to heat transfer analysis of thin-walled structures subjected to thermal contact. Particular attention is focused herein on time integration, element formulation, stability and thermal contact heat transfer. Fundamentals not covered can be found in e.g. [12].

The thermal model is based on the solution of the semi-discrete heat equation

$$CT + HT = f$$

where $C$ is the capacity matrix, $H$ is the conductivity matrix, $T$ is the temperature vector, $\dot{T}$ is the time derivative of $T$, and $f$ is the heat load vector.

3.1 Time integration

Large temperature differences between the workpiece and tools in combination with an evolving thermomechanical contact will impose a restriction on the time step size. We consider an explicit scheme a viable alternative to the more commonly used implicit midpoint rule since the time step size must be relatively small in the actual application to obtain an accurate solution. The time discretisation of Eq. (1) is achieved by the forward difference method where the time derivative $\dot{T}$ is approximated as

$$\dot{T}_n = \frac{T_{n+1} - T_n}{\Delta t}$$

where $\Delta t$ is the time step size. Substituting Eq. (2) into Eq. (1) gives

$$T_{n+1} = T_n - \Delta t C^{-1} (H_n T_n - f_n).$$

The advantage of explicit time integration can be seen in Eq. (3). If the capacity matrix is diagonal (lumped), the solution can be advanced in time without the necessity of equation solving. Furthermore, the complexity of the method is independent on whether the system is linear or nonlinear. The disadvantage is that the method is only conditionally stable. Thus, the time step size must be smaller than a critical value which can be determined from the geometry and material properties of the elements in the model.
3.2 Thermal shell element

Shell elements for heat conduction are normally derived from three-dimensional isoparametric solid elements where two faces of arbitrary order are connected by linear edges [13], resulting in a linear temperature approximation in the thickness direction. In applications where the shell surfaces are subjected to one-sided or double-sided thermal contact, the linear temperature approximation in the thickness direction is not adequate. For accurate heat transfer analysis of such applications, an element with linear temperature approximation in the plane and quadratic in the thickness direction is proposed. For the present application, a typical configuration of tools modelled with three-dimensional solid elements and a workpiece modelled with shell elements is illustrated in Figure 1.

![Cut-through view of tools and workpiece configuration and corresponding finite element discretisation.](image)

Figure 1. Cut-through view of tools and workpiece configuration and corresponding finite element discretisation.

The geometry of the shell element is defined as in the DYNA3D [14] implementation of the linear Hughes-Liu shell element [15]. The Cartesian coordinates of a point in the element are defined by the following equations

\[
x(\xi,\eta,\zeta) = \sum_{a=1}^{4} N_a(\xi,\eta)\bar{x}_a + \sum_{a=1}^{4} N_a(\xi,\eta)z_a(\zeta)\hat{x}_a
\]

(4)

\[
z_a(\zeta) = \frac{h_a}{2}(\zeta - \bar{\zeta})
\]

(5)

where \(\xi\) and \(\eta\) are the natural coordinates in the plane of the element and \(\zeta\) is the
natural coordinate in the thickness direction. In Eqs. (4) and (5), $x$ is a point in the element, $N_a$ denotes a two-dimensional shape function associated with node $a$, $\bar{x}_a$ is the position of node $a$, $z_a$ is a thickness function associated with node $a$, $\hat{X}_a$ is a unit vector which defines the fiber direction at node $a$, $h_a$ is the shell thickness in the fiber direction at node $a$, and $\zeta \in [-1,1]$ is the reference surface location. When $\zeta = -1, 0$ and $1$, the reference surface is located at the bottom, middle, and top surface of the shell, respectively. The same geometry description can be used for the Belytschko-Lin-Tsay shell element [16] where the geometry is defined by the reference surface ($\zeta = 0$) with coordinates $\bar{x}_a$, uniform thickness $h$, and fiber direction $\hat{X}$. The fiber direction is in this case computed as

$$\hat{X} = \frac{s_3}{\|s_3\|}$$

with

$$s_3 = r_{31} \times r_{42}$$

where $r_{ij}$ is a vector emanating from node $j$ to node $i$ and $\| . \|$ denotes the Euclidean norm.

The temperature approximation is derived by introducing additional temperature nodes in the thickness direction. This approach was used by Surana et al. [17] when deriving a curved shell element for steady state heat conduction with $p$-approximation in the thickness direction. For the linear-quadratic element 12 temperature nodes are required. The concept is illustrated in Figure 2. The temperature nodes are always located at $\zeta = -1, 0$ and $1$, independently of the reference surface location. The temperature approximation can be written as

$$T(\xi, \eta, \zeta) = \sum_{a=1}^{4} N_a(\xi, \eta)(\bar{N}_1(\zeta)T_{a1} + \bar{N}_2(\zeta)T_{a2} + \bar{N}_3(\zeta)T_{a3})$$

where

$$N_1(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 - \eta)$$
$$N_2(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 - \eta)$$
$$N_3(\xi, \eta) = \frac{1}{4}(1 + \xi)(1 + \eta)$$
\[ N_4(\xi, \eta) = \frac{1}{4}(1 - \xi)(1 + \eta) \]  
(8e)

\[ \overline{N}_1(\xi) = \frac{1}{2} \xi(\xi - 1) \]  
(8f)

\[ \overline{N}_2(\xi) = 1 - \xi^2 \]  
(8g)

\[ \overline{N}_3(\xi) = \frac{1}{2} \xi(\xi + 1) \]  
(8h)

The element integrals are calculated using Gaussian quadrature. The capacity matrix is computed using a \(2 \times 2 \times 3\) rule, and the conductivity matrix using a \(2 \times 2 \times 2\) rule. These quadrature rules are such that "full integration" of the element integrals are obtained.

![Figure 2](image)

Figure 2. Notations used for the quadratic temperature approximation in the thickness direction at node a.

### 3.3 Diagonal form of the capacity matrix

To reduce the computational expense of explicit time integration a diagonal form of the capacity matrix is used. The diagonal form is calculated using the row-sum method [18], where the consistent capacity matrix

\[ C_{ij} = \int_{\Omega_e} \rho c N_i N_j d\Omega \]  
(9)
is replaced by

\[(C_{ij})_L = \delta_{ij} \sum_k C_{ik}\]  \hspace{1cm} (10)

where \(N_i\) and \(N_j\) are the shape functions, \(\rho\) is the density, \(c\) is the heat capacity and \(\delta_{ij}\) is the Kronecker delta. Note, using Eq. (10) for quadratic elements gives lumped quantities that are negative. This is not the case for the linear-quadratic element where each lumped quantity is positive.

3.4 Stability

In return for the simplicity of the forward difference method it is only conditionally stable. Thus, stability must be investigated for each element type used in the finite element discretisation. To study the stability we consider the homogenous form of Eq. (1). Using Eq. (2) we can write the recurrence formula

\[T_{n+1} = AT_n\]  \hspace{1cm} (11)

where

\[A = I - \Delta t C^{-1} H\]  \hspace{1cm} (12)

is the amplification matrix and \(I\) is the identity matrix. The stability is dependent on the eigenvalues of the amplification matrix, which can be obtained by solving the characteristic equation

\[AT_i = \mu_i T_i\]  \hspace{1cm} (13)

where \(\mu_i\) are the eigenvalues and \(T_i\) are the corresponding eigenvectors. For stable schemes we must have

\[|\mu_i| \leq 1\]  \hspace{1cm} (14)

for any eigenvalue \(\mu_i\) of \(A\), since numerical errors must not be amplified from one step to the next.

Investigations of this type have been done by Comini et al. [19]. They obtained algebraic expressions of the maximum stable time step size for several commonly used low order elements with both consistent and lumped capacity matrices. A
result from their work is the stable time step size for the 8-node brick element with a lumped capacity matrix

\[ \Delta t_{\text{crit}} = \frac{\rho c L^2}{2k} \]  

(15)

where the characteristic length \( L \) is the smallest distance between any two nodes lying on an element edge, \( \rho \) is the density, \( c \) is the heat capacity and \( k \) is the thermal conductivity. It was also shown that consistent capacity matrices lead to more severe conditions for stability than lumped capacity matrices.

Solving Eq. (13) with global matrices replaced by the element matrices for the linear-quadratic element gives

\[ \Delta t_{\text{crit}} = \frac{\rho c t^2}{12k} \]  

(16)

where \( t \) is the element thickness. In Eq. (16) it is assumed that the smallest characteristic length is in the thickness direction of the element. If the smallest length is in the plane of the element, Eq. (16) still holds as long as the thickness is less than two times larger than the in-plane length.

3.5 Contact heat transfer

The resistance to heat transfer when rough surfaces are pressed together is mainly due to the low percentage of surface area really in contact. The heat transfer through the contact interface takes place by conduction through the contacting spots, conduction through the interstitial gas, and radiation across the gaps. A large number of contact conductance models are available, see [20, 21] for a review. These models indicate that the contact conductance depends on the material of the bodies, the microscopic shape of the surfaces, the contact pressure and other factors. A finite element formulation of a contact element based on microscopic thermomechanical laws including spot and gas conduction is presented by Zavarise et al. [22].

Since the contact is considered at discrete points, i.e. between a contact node and the corresponding contact segment, the heat transfer between these points need to be established. A single-surface contact search algorithm [23] is used to find all contacting nodes with their corresponding contact segments. We assume one-dimensional heat transfer where the heat flux \( q \) is given by
\[ q = h(p)(T_a - T_b) \]

where \( h \) is the contact conductance, \( T_a \) is the temperature at the contact node, and \( T_b \) is the temperature at the contact segment. The contact conductance is assumed to be a combination of additive components from spot and gas conduction, and radiation. In the present work a pressure dependent contact conductance is implemented. The contact pressure is computed from the contact force and the contact area associated with the contact node. Each segment sharing the contact node contribute to the contact area as shown in Figure 3. The contribution from a segment is simply computed as

\[ A_i = \frac{1}{4}|m_i \times n_i| \]

where \( m_i \) and \( n_i \) are vectors along the segment edges emanating from the contact node. Some care must be used in the computation of this area for the case in which one or both surfaces of the interface is sharply curved.

4 Mechanical model

The mechanical model is based on an explicit dynamic formulation. In the present work the public domain code DYNA3D from Lawrence Livermore National Laboratory is used.

Thermomechanical analyses in standard DYNA3D are based on an uncoupled approach in which the thermal problem is solved based on the initial geometry, and the resulting temperature history is used as thermal loads in the mechanical problem. In applications where the deformation changes the thermal boundary

Figure 3. Contact area associated with a contact node.
conditions a coupled analysis is required. The proposed thermal model is implemented into DYNA3D to solve coupled thermomechanical problems.

4.1 Material model

The material model employed in this work is a temperature dependent elastic-plastic model [11]. The strain rate dependence at higher temperatures is not considered in the model. The plastic behaviour of the material is described by von Mises isotropic yield condition, an associated flow rule and mixed linear isotropic-kinematic hardening. The effective-stress-function algorithm [24] is used to update the stresses. Transformation plasticity is included in the model as an additional strain component related to the stress state and the progress of transformation [25].

Some modifications of the model in [11] have been made. Firstly, the model has been extended for three-dimensional stress calculations. Secondly, secant iterations is used to account for the zero normal stress condition in shell analysis [26]. Thirdly, a nonlinear isotropic hardening law has been added. Experimental observations indicate a strongly nonlinear stress-strain relationship for quenched steels at temperatures below the martensite start temperature. The hardening law is defined as tabulated values of yield stress versus effective plastic strain.

It is well known that the blank, usually made of rolled material, exhibits transverse anisotropy. In finite element simulations of cold sheet metal forming processes the Hill [27] anisotropic yield condition is often used. In the hot-stamping process, where the blank is austenitized before forming, it is assumed that the transverse anisotropy vanish and that an isotropic yield condition can be used.

The order of the temperature field in thermal stress analyses should be such that consistent total and thermal strain fields are created. Inconsistent strain fields may create spurious stresses [28, 29]. In elements with linear displacement fields, thermal strain fields should be calculated from a constant temperature calculated at the element centroids. In the present implementation, thermal strains in shell elements are calculated using the full thermal field in the thickness direction. That is, at each integration point the temperature is computed using Eqs. (8a-8h) evaluated at \((0,0,\zeta)\). In 8-node brick elements, thermal strains are calculated from a constant temperature.
5 Thermomechanical coupling

A staggered approach is used for coupling the thermal and mechanical analysis. In this approach, separate analyses are performed with data exchange at the end of each time step. In particular, the thermal analysis is based on the geometry and contact data calculated at the end of the previous mechanical step. The resulting thermal field is then used in the mechanical analysis. The thermal and mechanical problem are both solved using an explicit formulation, where each time integration scheme have different stability limits. The stability limit for a square shell element with side lengths $L$ and thickness $t$ in the mechanical formulation is

$$\Delta t_m = \frac{L}{\sqrt{\frac{E(1-v)}{(1+v)(1-2v)}\rho}}$$

(19)

where $E$ is Young’s modulus, $v$ is Poisson’s ratio and $\rho$ is the density. The stability limit for the same element in the thermal formulation with $t < L$ is given by Eq. (16). The thermal stability limit is greater than the mechanical stability limit if

$$\frac{t^2}{L} > \frac{12k}{c} \sqrt{\frac{(1+v)(1-2v)}{\rho E(1-v)}}$$

(20)

The same comparison for a cubic 8-node brick element with side lengths $L$ restricts the smallest element length to

$$L > \frac{2k}{c} \sqrt{\frac{(1+v)(1-2v)}{\rho E(1-v)}}$$

(21)

For a typical steel at room-temperature this yields $t^2/L > 10^{-8}$ m for a shell element and $L > 10^{-9}$ m for an 8-node brick element, which will be satisfied for a normal element size. Since the stable time step size in the thermal formulation is approximately $10^4$ to $10^6$ times larger than in the mechanical formulation, each analysis is performed with different time step sizes. That is, between two consecutive thermal steps, many mechanical steps are performed. During these intermediate mechanical steps a linearly interpolated temperature field is used. In applications where the evolving thermomechanical contact affects the heating or cooling of the workpiece some care must be used when determining the time step size in the thermal analysis. A smaller time step size may be required than predicted by Eqs. (15) and (16).

Using an explicit formulation for solving quasi-static problems such as sheet forming requires a very large number of time steps. To increase the computational
efficiency of the mechanical formulation in such applications, two numerical
artifices are generally employed: i) the stability limit is increased by increasing the
material density; ii) the natural time of the process is artificially reduced by
increasing the velocity of the tool. The amounts by which the material density and
tool velocity can be increased is limited, as the inertia forces may become
unacceptably large. The first approach is useful when the mechanical material
response is rate dependent. In the present work, the second approach is employed.
In a thermomechanical analysis, thermal properties must be scaled accordingly. For
example, if the tool velocity is increased by a factor 10, the thermal conductivity,
contact conductance, radiation and convection coefficients are each increased by a
factor 10, while the time step size and time scale of the prescribed temperature
histories are each decreased by a factor 10. In a multiple process such as hot-
stamping, the duration of the forming stage is shorter than the total process time.
After the forming stage is completed, additional scaling of the thermal problem is
possible which further reduces the computational time.

6 Numerical examples

The proposed model and the implementation into DYNA3D are evaluated by three
examples. The first two examples focus on the validation of the model. The third
example is an application of the model to a simultaneous forming and quenching
problem.

6.1 Thermal contact

The first test is a one-dimensional steady state problem from [30] where an
analytical solution is available. This problem concerns the thermal contact between
two cubic blocks of dimension $L = 0.002$ m. The top and bottom blocks are given
initial temperatures of 800°C and 0°C, respectively. The temperature of the top
surface of the top block is prescribed to be $T_A = 800$ °C, and the temperature of
the bottom surface of the bottom block is prescribed to be $T_B = 0$ °C. The sides of
the blocks are insulated. The thermal conductivity $k$ is 20 W/m°C, the heat
capacity is 500 J/kg°C and the density is 7800 kg/m$^3$. The thermal contact
conductance $h$ is 6000 W/m$^2$ °C. The problem is solved with two models, one
with two thermal shell elements and one with ten 8-node brick elements. The
analytical solution for the contact surface temperature of the top block ($T_A$) and the
bottom block ($T_B$) is given by

$$T_A = \frac{(1 + \eta)T_A + \eta T_B}{1 + 2\eta} \quad \text{and} \quad T_B = \frac{(1 + \eta)T_B + \eta T_A}{1 + 2\eta}$$ (22)
where
\[ \eta = h \frac{L}{k}. \] (23)

The calculations converges exactly to the analytical solution of \( T_A = 6400/11 \) °C and \( T_B = 2400/11 \) °C.

In the second test, a one-dimensional transient problem is solved. This test is designed to evaluate the thermal shell element in an application similar to a hot blank in contact with a cold tool. Once again the contact between two blocks are considered. The dimension of the top block is 0.002 m, and the dimension of the bottom block is 0.075 m. The top and bottom blocks are given initial temperatures of 800°C and 20°C, respectively. The same material data as in the first test are used. The top block is modelled with a thermal shell element and the bottom block with 115 8-node brick elements. For comparison, the same problem is solved using an implicit formulation. The top and bottom blocks are in this case modelled with 20 and 115 8-node brick elements, respectively. The time step size is 0.001 s in both calculations. The temperature distribution in the top block at different times are shown in Figures 4 to 6.

![Figure 4](image-url). Temperature in the top block as a function of distance from contact interface at \( t = 0.005 \) s.

The difference between the two calculations shows that a quadratic function can not describe the temperature distribution exactly in the beginning, but at sequential times the difference almost disappears. Figure 7 shows the temperature at three
points in the top block as a function of time. From this it is evident that the thermal shell element can be used to model thin-walled thermal contact problems with good accuracy.

Figure 5. Temperature in the top block as a function of distance from contact interface at $t = 0.05\ s$.

Figure 6. Temperature in the top block as a function of distance from contact interface at $t = 0.5\ s$. 
6.2 Thermo-elastic bending

To test the validity of using thermal strains calculated from the full thermal field in the thickness direction of shell elements, a simply supported beam subjected to a quadratic temperature distribution in the transverse direction is studied. Ten Hughes-Liu shell elements are used to model the 10 mm long beam, which is 1x1 mm in cross section. Young’s modulus $E$ is 200 GPa and Poisson’s ratio is zero. The coefficient of thermal expansion $\alpha$ is $2 \times 10^{-5} \, ^\circ\text{C}^{-1}$. Initially the beam is at uniform temperature of 0°C. The bottom is kept at $T_b = 0$ °C, while the middle is heated to $T_m = 75$ °C and the top is heated to $T_t = 100$ °C. The analytical solution based on technical beam theory for the bending stress in a beam with thickness $h$ is

$$\sigma = 2E\alpha(T_t + T_b - 2T_m)\left(\frac{1}{12} - \left(\frac{z}{h}\right)^2\right)$$  \hspace{1cm} (24)$$

where

$$-\frac{h}{2} \leq z \leq \frac{h}{2}.$$  \hspace{1cm} (25)$$

Figure 8 shows the analytical solution for the bending stress and the finite element solution achieved at the Gauss points. In the finite element solution five Gauss
points are used through the thickness. As can be seen in Figure 8, there is very good agreement. Tests with finite element solutions of a beam subjected to a constant or linear temperature distribution predict stress free deformation fields, a result in agreement with Eq. (24).

![Bending stress distribution through thickness](image)

Figure 8. Bending stress distribution through thickness.

6.3 Simultaneous forming and quenching

The objective of this simulation is to evaluate the proposed model by comparison with a one-sided forming and quenching experiment. The experiment is designed to include most of the characteristics of the industrial application of hot-stamping under well defined boundary conditions.

The 2.9 mm thick plate is modelled with 132 quadrilateral Hughes-Liu shell elements in the mechanical part of the analysis. The thermal shell element is used for the heat transfer analysis in the plate. The tool is considered to be rigid in the mechanical analysis, but is modelled with 1600 8-node brick elements since the same mesh is used in both the mechanical and thermal analysis. Temperature and temperature history dependent thermal and mechanical material properties are used in the analysis. Because of symmetry conditions only one quarter of the workpiece and the tool is modelled. The finite element mesh can be seen in Figure 9. The tool motion is modelled with prescribed velocities and the forming depth is 26 mm. The initial temperature of the workpiece and the tool is 796°C and 20°C, respectively.
A constant thermal contact conductance equal to 6000 W/m²°C is used. The total experimental process time is approximately 60 s. The actual forming stage is completed after approximately 1 s, after which the tool and the workpiece remains in contact for further cooling. To decrease the required solution time, the velocity of the tool is increased by a factor 20. Additional scaling of the thermal problem by a factor 50 is employed after 1.1 s. The required problem time $t_d$ in DYNA3D for a process time $t_p$ can be expressed as

$$t_d = \frac{t_a}{s_i} + \frac{t_p - t_a}{s_i \cdot s_a}$$

(26)

where $s_i$ is the initial scale factor and $t_a$ is the time when the additional scale factor $s_a$ is employed. In the present simulation the total problem time is 0.1139 s.

The temperature distribution in the plate and the tool at time $= 9$ s is shown in Figure 9.

![Figure 9. Temperature distribution in the plate and the tool at $t = 9$ s.](image-url)
The measured and computed tool force are shown in Figure 10. The tool force is computed with and without transformation plasticity (trp), in both cases including the transformation volume change. After the forming stage is completed, the force increases due to thermal shrinkage. When the martensite start temperature \( (M_s = 247^\circ C) \) is reached in the plate, the force decreases. The simulations shows acceptable agreement during the initial forming stage, followed by an overestimation of the force for times between 2 s and 10 s. The computed force without trp shows acceptable agreement at sequential times. When trp is included the force decreases rapidly when \( M_s \) is reached.

![Figure 10. Measured and computed tool force.](image)

Full details of the experiment and the computational model are found in [31].

**7 Conclusions**

A coupled thermomechanical formulation applicable to simultaneous forming and quenching simulation is presented.

A staggered approach is used for coupling the thermal and mechanical problem which are both solved using an explicit formulation. The stability limit for the proposed linear-quadratic thermal shell element and an 8-node thermal brick element is compared with the mechanical counterpart. The comparisons indicate that the stability limit for the thermal problem is much larger than for the
mechanical problem. As a consequence, different time step sizes are used in the thermal and mechanical analysis.

The model is validated and evaluated by three examples. It is shown that the thermal shell element can model thin-walled thermal contact problems with good accuracy. By comparison with an analytical solution it is shown that thermal strains in shell elements can be calculated using the full thermal field in the thickness direction. The evaluation of the model by comparison with the forming and quenching experiment show qualitatively agreement, but also that there are remaining challenges for future research. The material response, mechanical and thermal, must be further evaluated and the influence from transformation plasticity as well as thermal contact should be further investigated for the present application.

Acknowledgement

The financial support from the Research Council of Norrbotten is gratefully acknowledged.

References


FINITE ELEMENT ANALYSIS OF SIMULTANEOUS FORMING AND QUENCHING OF THIN-WALLED STRUCTURES

G. Bergman, L. Sandberg* and M. Oldenburg

Department of Mechanical Engineering, Luleå University of Technology
SE-971 87 Luleå, Sweden

* SSAB HardTech AB
SE-971 88 Luleå, Sweden

Abstract

Finite element analyses of simultaneous forming and quenching are compared with corresponding experimental results. A coupled thermomechanical finite element code based on explicit time integration is used in the simulations. In the developed experiment, pre-heated steel plates are simultaneously formed and quenched by a cold tool. Temperatures and tool force are measured for comparison with the simulations. Thermal and mechanical material properties of the steel plate at elevated temperatures are presented. The analyses show good agreement during the initial forming stage, followed by an overestimation of the tool force at sequential times. It is shown that the computed tool force is very sensitive to the sequence of cooling in different parts of the plate.

1 Introduction

The industrial process of simultaneous forming and quenching of thin-walled automotive components has grown to become important in both economical and technological terms. The process, also referred to as hot-stamping, has been used mostly for safety related structural components such as side impact protection beams and bumper systems. An increased use of high strength hardened components in vehicle structures are foreseen due to the possibilities of weight reductions. The advantage of using simultaneous forming and quenching for high strength parts is the dimensional accuracy of the process. High strength parts are difficult or impossible to form after hardening. If quenched after forming, there will be considerable distortion due to residual stresses from the forming stage combined with the dilatation during quenching.

The research concerning modelling and simulation of sheet metal forming as well
as thermomechanical manufacturing processes is extensive, see e.g. [1] and [2] conference proceedings. Related to this study, methods for modelling and simulation of the combined forming and quenching process are developed, cf. [3]. The aim of the present work is to evaluate the implemented methods by comparison of finite element simulations with experimental results.

2 Simultaneous forming and quenching experiment

The experiment that is our subject for analysis is designed to include most of the characteristics of the industrial application of hot-stamping under well defined boundary conditions. The one-sided forming and quenching experiment shown in Figure 1 consists of a cylindrical tool and a plate support designed to avoid draw-in of material during the test. The initial distance between the tool and the workpiece is 3 mm in all tests. The initial geometry of the test specimens at room-temperature (20°C) is shown in Figure 2. The initial thickness of the test specimens used in the test are 1.9 mm and 2.9 mm. The tool and the plate support are made of SS 142242-02 steel, and the test specimens are made of SS 142550-02 steel.

The experimental procedure followed is austenitization of the workpiece at 850 °C for 30 minutes, insertion of the workpiece into the set-up, and forming to a depth of 26 mm. After the forming stage is completed, i.e. after approximately 1 s, the tool and the workpiece remains in contact for further cooling. The tool velocity is given in Table 1. The initial temperature of the 1.9 mm and 2.9 mm test specimen, when inserted into the set-up, is 748°C and 796°C, respectively. The initial temperature of the tool is 20°C in all tests. During the experiment, both temperatures and the tool force are measured. The temperatures are measured with thermocouples type K mounted in drilled holes in the tool or welded on the surface at the non-contact side of the test specimens. The location of the temperature measurement points, T_A to T_H, is shown in Figures 1 and 2. The tool force is measured by the Instron 1251 tensile test machine in which the set-up is installed.

3 Material properties

The dependence on the microstructure of the material is included in the properties of SS 142550-02 since the presented values are obtained during quenching from the austenite region. The properties of the tool material, SS 142242-02, are given for heating from room-temperature.
3.1 Thermal properties

The heat capacity of SS 142550-02 originates from measurements performed at the Thermophysical Properties Research Laboratory [4]. The temperature history during the measurements were austenitization at $850^\circ$C for 30 minutes followed by cooling at $20^\circ$C per minute. The cooling rate was high enough to obtain only austenite to martensite transformation. The martensite start temperature $M_s$ is $247^\circ$C. The thermal conductivity of SS 142550-02 is estimated using a linear mixture rule [5]. The single phase data for austenite and martensite are obtained from [6]. These data originate from measurements on British Standard Steel En 352. Measured heat capacity and calculated thermal conductivity of SS 142550-02 are shown in Figure 3. Heat capacity and thermal conductivity of SS 142242-02 are given in Table 2 [7]. The density is considered to be constant for both steels, equal to 7800 kg/m$^3$.

3.2 Mechanical properties

The mechanical properties of SS 142550-02 are obtained from in-house tests and from literature. Some mechanical properties of 60 NCD 11, equivalent to SS 142550-02, are available in [8]. A series of tensile tests, according to standard SS 112116, have been performed at elevated temperatures. The temperature history during the tests were austenitization at $850^\circ$C for 30 minutes followed by air cooling to the selected test temperature, fast enough to obtain only austenite to martensite transformation. The tensile tests were performed at a constant strain rate of 0.07 s$^{-1}$. Experimental true stress-strain curves are presented here as yield stress versus effective plastic strain curves for the inelastic behaviour. The elastic behaviour is described in terms of Young’s modulus and Poisson’s ratio. Yield stress versus effective plastic strain curves during quenching are shown in Figures 4 and 5. Young’s modulus and thermal dilatation are shown in Figure 6. The thermal dilatation includes both thermal expansion and volume change due to phase transformation. Poisson’s ratio is considered to be constant for both steels, equal to 0.3.

A comparison of the inelastic material behaviour shown in Figures 4 and 5 with the corresponding data presented in [8] is difficult since this data is in the form of bilinear curves. Furthermore, the tangent modulus presented in [8] is determined by considering only a deformation up to 0.5%. Somewhat higher values for the virgin yield stress and plastic modulus are reported in [8] at temperatures above $M_s$. A significantly different material behaviour is found in the present study at
temperatures below $M_s$. It is clear that this strongly nonlinear behaviour can not be represented by a linear hardening law.

Besides the properties mentioned above, additional data are required if transformation plasticity effects are to be included in the material modelling. In this work a transformation plasticity model proposed by Leblond et al. [9] is used. The required properties are the relative volume change associated with the austenite to martensite transformation $\Delta V/V$ and the yield stress of austenite $\sigma_y$. In [8], the effects of transformation plasticity have been measured using a tensile test. Following the same procedure as in [5], where FEM calculations were fitted to experimental data, $(\Delta V/V)/(1/\sigma_y)$ is estimated to be $9.9 \times 10^{-11} \text{ Pa}^{-1}$. In the present estimation, the FEM calculations are fitted to experimental axial strain curves for applied tensile stress levels of 18 MPa, 107 MPa, 209 MPa, and 285 MPa.

4 Computational model

Simulation of the simultaneous forming and quenching experiment requires a finite element code capable of solving coupled thermomechanical problems. In [3], a thermal model based on explicit time integration is developed and implemented into the explicit finite element code DYNA3D [10]. A staggered approach is used for coupling the thermal and mechanical analysis, wherein each analysis is performed with different time step sizes. The thermal model includes two element formulations, an 8-node brick element and a shell element with linear temperature approximation in the plane and quadratic in the thickness direction. The shell element uses the same geometry description as the Hughes-Liu shell element [11]. Thus, the same mesh can be used for both the thermal and mechanical analysis. Thermal contact, in which heat transfer depend on the mechanical deformation, is included in the thermal model. The material behaviour is described by a temperature dependent elastic-plastic model with a nonlinear isotropic hardening law. Transformation plasticity is included in the model. More details can be found in [3, 5].

The plates are modelled with 132 quadrilateral Hughes-Liu shell elements in the mechanical part of the analysis. Five through-thickness integration points are used in all analyses. The thermal shell element is used for the heat transfer analysis in the plates. The geometry of the plates is compensated for the dimensional change due to pre-heating in the furnace. A mean coefficient of thermal expansion equal to $1.27 \times 10^{-5} \degree \text{C}^{-1}$ is used. The corresponding temperature change is determined from the actual start temperature of the experiment and the surrounding air temperature.
of 20°C. The modelled length of the plates is 129 mm. The tool is considered to be rigid in the mechanical analyses, but is modelled with 1600 8-node brick elements since the same mesh is used in both the mechanical and thermal analysis. Because of symmetry conditions only one quarter of the plates and the tool is modelled. The finite element mesh used in the analysis of the 2.9-mm plate is shown in Figure 7.

The plate support is not modelled since it is assumed that no draw-in of material occurs. Instead, nodes at the end of the plate, along line A in Figure 2, are fully restrained. The tool motion is prescribed according to the velocities given in Table 1. Note, these velocities is scaled as described below. Frictional effects are not considered in the analyses. The boundary conditions considered in the thermal analyses are contact heat transfer between the plate and the tool, cooling from the support, and convection and radiation at the non-contact side of the plate. A constant thermal contact conductance equal to 6000 W/m²°C is used. The cooling from the support is modelled by prescribing the temperature of the nodes between line A and point T_E according to the measured temperature history at point T_E, see Figure 2. Values for free convection are obtained from standard formulas for horizontal plates. The surface emissivity is assumed to be 0.8.

In DYNA3D, the time step size is calculated from the geometry and material properties of the elements in the model. The same option is available in the thermal model. The stable time step size is approximately 0.05 s in the thermal analyses. This estimation is based on the geometry of the initial mesh and the thermal material data in the temperature range from 800°C to 20°C. This implies that about 20 thermal steps will be performed during the actual forming stage if the largest possible time step size is used. In practice, a smaller time step size is required for this type of problem where the evolving thermomechanical contact affects the cooling of the workpiece. As a consequence, a constant time step of 0.001 s is used in the thermal analyses.

The total experimental process time is approximately 50 to 60 seconds depending on plate thickness. The actual forming stage is completed after approximately 1 s, after which the tool and the workpiece remains in contact for further cooling. To decrease the required solution time, the velocity of the tool is increased by a factor 20. The thermal problem is scaled accordingly. That is, the thermal conductivity, contact conductance, radiation and convection coefficients are each increased by a factor 20, while the time step size and time scale of the prescribed temperature history are decreased by a factor 20. Additional scaling of the thermal problem by a factor 50 is employed after 1.1 s. The total problem time considered in the
DYNA3D solution is 0.1139 s, representing a real process time of 60 s [3].

The microstructure evolution is modelled as time independent since the cooling rate is high enough to obtain only martensitic transformation. The decomposition of austenite into martensite is described by the Koistinen-Marburger’s equation [12].

5 Results and discussion

5.1 The 2.9-mm plate

The measured and computed temperatures in the plate and the tool are shown in Figures 8 to 10. The computed temperature at point $T_A$ agrees well with the measured temperature until time $t = 20$ s, subsequently a slightly faster cooling is obtained in the simulation. The temperatures at point $T_B$ are very similar to the presented temperatures at point $T_A$. Computed temperatures at point $T_C$ and $T_D$ differs significantly compared to measurements. The maximum deviation at point $T_C$ and $T_D$ is approximately 200°C and 250°C, respectively. The measured and computed temperatures at point $T_F$ and $T_G$ in the tool show good agreement. The temperatures at point $T_H$ are very similar to the presented values at point $T_F$. The measured and computed tool force are shown in Figure 11. The tool force is computed with and without transformation plasticity (trp), in both cases including the transformation volume change. The measured force is 13.5 kN at $t = 1$ s. The abrupt decrease in tool force after $t = 1$ s could indicate some slip at the plate support. After the forming stage is completed, the force increases due to thermal shrinkage to a maximum of 16.3 kN at $t = 7.5$ s. When the martensite start temperature is reached at point $T_A$ and $T_B$, the force decreases. The computed forces are 11 kN at $t = 1$ s. The force without trp shows acceptable agreement until $t = 14$ s, when the force increases until $t = 32$ s. When trp is included the force decreases rapidly when $M_s$ is reached.

It is believed that the increase in computed force without trp, from $t = 14$ s to $t = 32$ s, is a result of the much slower cooling obtained in the part of the plate where point $T_C$ and $T_D$ is located. Since the temperature in this part is above $M_s$ at $t = 14$ s, further cooling causes thermal shrinkage which results in the observed increase in force. It should be mentioned that the computed temperatures when including trp were similar to the temperatures shown in Figures 8 to 10. One source of the observed difference in temperatures may be the assumption of a constant thermal contact conductance equal to 6000 W/m²°C. However, the discrepancy in temperatures is not believed to be caused solely by this assumption since acceptable
agreement in temperatures are found at other points in the plate and the tool. The most likely explanation to the observed discrepancies is that, in the simulation, this part of the plate is not in contact with the tool. The absence of contact is probably due to the geometric discretisation of the contact surfaces.

In order to investigate the effect of faster cooling close to the support, the distance between surfaces for considering contact is increased from $1 \times 10^{-4}$ m to $5 \times 10^{-4}$ m for contact segments in the part that is outside the contact region. The re-computed temperatures at point $T_C$ and $T_D$ are shown in Figure 12. Compared to Figure 9 it can be seen that the difference between computed and measured temperatures has decreased. The maximum deviation at point $T_C$ and $T_D$ is now approximately $65^\circ C$ and $80^\circ C$, respectively. There is still a shift in time when $M_s$ is reached. The martensite start temperature at point $T_C$ is reached at $t = 9$ s in the experiment and at $t = 11$ s in the simulation. The corresponding times at point $T_D$ are $t = 16$ s and $t = 21$ s, respectively. The temperatures at the other points were very similar to the to the temperatures shown in Figures 8 and 10. The re-computed tool forces are shown in Figure 13. The computed force is 11 kN at $t = 1$ s and 22 kN at $t = 8$ s. As can be seen, faster cooling leads to an overestimation of the force for times between 2 s and 10 s, but also to better agreement at sequential times. Once again, when trp is included the force decreases rapidly when $M_s$ is reached. This rapid stress relaxation indicates that the applicability of the transformation plasticity model under the circumstances of very high stress levels and previous large plastic deformation is a subject for further investigations.

5.2 The 1.9-mm plate

The computed results presented for the 1.9-mm plate is with the same adjustments of the contact distance as for the 2.9-mm plate. The measured and computed temperatures are shown in Figures 14 to 16. Fewer temperature measurement points were used in the 1.9-mm plate. The computed temperature at point $T_A$ agrees well with the measured temperature during the first 4 s, subsequently a faster cooling is obtained in the simulation. The maximum discrepancy between measured and computed temperatures at point $T_C$ is approximately $50^\circ C$. The measured and computed temperatures at point $T_F$ and $T_G$ in the tool show acceptable agreement. The measured and computed tool force are shown in Figure 17. As can be seen, the simulation of the 1.9-mm plate shows a behaviour similar to that of the 2.9-mm plate.
6 Conclusions

In this work we have evaluated methods for simulation of simultaneous forming and quenching by comparison with corresponding experimental results. It is shown that an explicit formulation can be used for simulation of hot-stamping. The process time of the present experiment is, however, quite long. On the other hand, the process time is much shorter in the industrial application of hot-stamping due to active cooling of the tools. It should be mentioned that it is not investigated how much further the tool velocity and the additional scaling of the thermal problem can be increased until inertia forces affect the results too much. In this study, it is also shown that the linear-quadratic thermal shell element is capable of accurate modelling of heat transfer in thin-walled structures subjected to thermal contact.

Comparison of the finite element simulations with the experimental results showed at first large discrepancies with respect to the tool forces. The most likely explanation to the observed deviations is that, in the analysis, there is no contact between the plate and the tool near the support. Exactly how much of the plate that is in contact with the tool can not be determined from the experimental data. When measures are taken in order to expand the contact area towards the plate support, the force increases until martensite starts to form and decreases at sequential times, resulting in better agreement with measured values. There may be several possible causes to the remaining agreement between measured and computed tool force seen in Figures 13 and 17. Neglecting the strain rate dependence at high temperatures may be one source of error. The possible removal of the effects of previously accumulated plastic strains during phase transformations is not taken into account in the material model. Due to the accumulated measure of plastic strains, very large estimates of the yield strength at temperatures below $M_s$ are obtained in the present analyses. There is also a degree of uncertainty concerning the obtained values from the material testing. Another source may be frictional effects, which are not accounted for in the present study. However, it is believed that a minor slip at the plate support is the most plausible cause to the lower force obtained in the experiment after the forming stage is completed.

Future work may include additional material testing. The material response should be further evaluated by additional tensile tests at different strain rates. Experiments for determining the thermal contact conductance as a function of contact pressure, surface properties and other factors should also be performed. Experimental measurements are also needed to investigate the memory of previously accumulated plastic strains during martensite transformation and how large plastic deformation of the austenite phase affects the phase transformation and the transformation
plasticity.

Acknowledgement

The financial support from the Research Council of Norrbotten is gratefully acknowledged.

References


Figure 1. Experimental configuration and location of temperature measurement points in the tool (mm).

Figure 2. Initial geometry of test specimen and location of temperature measurement points (mm).
Figure 3. Heat capacity (c) and thermal conductivity (k) of SS 142550-02.

Figure 4. Yield stress versus effective plastic strain of SS 142550-02 at elevated temperatures.
Figure 5. Yield stress versus effective plastic strain of SS 142550-02 at elevated temperatures.

Figure 6. Young’s modulus (E) and thermal dilatation ($\varepsilon^T$) of SS 142550-02.
Figure 7. Finite element mesh used in the analysis of the 2.9-mm plate.

Figure 8. Temperatures in the plate at point $T_A$ and $T_E$. 
Figure 9. Temperatures in the plate at point $T_C$ and $T_D$.

Figure 10. Temperatures in the tool at point $T_F$ and $T_G$. 
Figure 11. Measured and computed tool force.

Figure 12. Temperatures in the plate at point $T_C$ and $T_D$. 
Figure 13. Measured and computed tool force.

Figure 14. Temperatures in the plate at point $T_A$ and $T_E$. 
Figure 15. Temperatures in the plate at point $T_C$.

Figure 16. Temperatures in the tool at point $T_F$ and $T_G$. 
Figure 17. Measured and computed tool force.
Table 1: Tool velocity.

<table>
<thead>
<tr>
<th>Time (s)</th>
<th>Velocity (m/s)</th>
</tr>
</thead>
<tbody>
<tr>
<td>0.0</td>
<td>0.0</td>
</tr>
<tr>
<td>0.072</td>
<td>0.0</td>
</tr>
<tr>
<td>0.222</td>
<td>0.04</td>
</tr>
<tr>
<td>0.688</td>
<td>0.04</td>
</tr>
<tr>
<td>0.89</td>
<td>0.02</td>
</tr>
<tr>
<td>1.02</td>
<td>0.0</td>
</tr>
</tbody>
</table>

Table 2: Heat capacity and thermal conductivity of SS 142242-02.

<table>
<thead>
<tr>
<th>Temperature (°C)</th>
<th>Heat capacity (J/kg°C)</th>
<th>Thermal conductivity (W/m°C)</th>
</tr>
</thead>
<tbody>
<tr>
<td>20</td>
<td>460</td>
<td>24.6</td>
</tr>
<tr>
<td>400</td>
<td>-</td>
<td>26.2</td>
</tr>
<tr>
<td>800</td>
<td>-</td>
<td>27.6</td>
</tr>
</tbody>
</table>
The objective of this thesis is to develop and evaluate numerical methods for modelling and simulation of simultaneous forming and quenching within an integrated product development environment. Simultaneous forming and quenching, also referred to as hot-stamping, is a manufacturing process for high strength automotive components such as side impact protection beams.

A concept for integrated product and process development is proposed. The prototype system consists of a CAD system, a relational database management system, program interfaces, and nonlinear finite element programs.

A thermal model based on explicit time integration is developed and implemented into the explicit finite element code DYNA3D to solve coupled thermomechanical problems. The implementation includes a linear-quadratic thermal shell element, and contact heat transfer. The material behaviour is described by a thermo-elastic-plastic material model. The effective-stress-function algorithm is used to update the stresses.

The implemented methods are evaluated by comparison with corresponding experimental results. In one of the developed experiments, pre-heated steel plates are simultaneously formed and quenched by a cold tool.

Typ: Doctoral thesis

Fakultet: Teknisk

Nyckelord:
finite element method, explicit time integration, shellelement, effective-stress-function algorithm, simultaneous forming and quenching, integration, relational database