3D Modelisation of thermo-mechano-metallurgical coupling during welding process assisted by MSC.Marc

C. Darcourt, Z. Benadla, M. Rachik, J.M. Roelandt, D. Deloison* and J. Delmotte*

UTC Laboratoire Roberval, UMR UTC-CNRS 6066
BP20529 60205 Compiègne Cedex, France
Phone: 33 (0) 3 44 23 45 52
E-mail: claudie.darcourt@utc.fr

* EADS CCR, DCR/SE/LM, 12 rue Pasteur
BP76 92152 Suresnes Cedex, France
Phone: 33 (0) 1 46 97 32 62
E-mail: dominique.deloison@eads.net

ABSTRACT

A 3D finite element model, including thermomechanical coupling, is proposed in order to model the laser welding process. The simulation breaks up into the heat transfer calculation that allows determining the space-time temperature evolution in the workpiece according to the heat source position, thermal boundary conditions... and the mechanical calculation that evaluates stresses and strains in the structure from the previously computed heat load. Our work mainly aims to the heat source (laser) description and the mechanical constitutive model. The moving heat source is simulated by a Gaussian distribution of heat density and the MSC.Marc user subroutine FLUX is used for this purpose. Concerning the mechanical constitutive model, we quantitatively compare a standard elastic perfect plastic model with a viscoplastic model. The latter one is implemented by mean of the MSC.Marc user subroutine HYPELA. The viscoplastic constitutive model implementation is validated on a simple relaxation test and the proposed 3D finite element model is then used to investigate a T welding test.

INTRODUCTION

In recent years high power density welding technologies, like Laser welding, have been increasingly used in industrial manufacturing with respect to traditional methodologies due to lower dimension, shape distortion and greater processing velocity. Within the framework of a programme on
the lightening of aeronautical structures, we investigate the numerical simulation of Laser welding. The
goal of this work is the development of a numerical procedure for the whole process investigation.
Such a procedure can be helpful for the assessment of the influence of welding parameters on the
involved material transformations and the residual displacement (final workpiece shape).

Welding process is founded on several key aspects and many disciplines: the complete coupling
diagram of Figure. 1 shows the mutual influence of the fields like temperature, stress-strain and
microstructural state.

**Figure. 1: Couplings in welding process**

The above schematised interdisciplinar two-ways interactions have the following meaning:

1. Dilatation due to the temperature history produces thermal stresses and strains. Material
parameters like the yield stress are strongly affected by the temperature.

2. The temperature history is the main factor that induces the metallurgical phase transformations.
Metallurgists use the TTT and CCT diagrams to describe these phenomena.

   **TTT diagram = Time, Temperature, Transformation**
   **CCT diagram = Continuous, Cooling, Transformation**

3. Each phase transformation is a source of deformation within the material. This is mainly due to the
density variations. Another phenomenon is also noticed: “the transformation plasticity”. This new
term of plastic strain appears when a transformation occurs in a stressed piece even if the stress
level is very low (2).

4. As a consequence of an elevated strain rate, the stress field evolution can involve a thermal
dissipation within the material.
5. When a phase transformation takes place under an applied stress kinetic modification of the transformation occurs. It increases or decreases the incubation time, i.e. the position of transformation lines and the transformation rates.

6. During the phase transformations the latent heat, which can influence the actual temperature transient field, is released.

**PROBLEM DEFINITION**

A first investigation is attempted on the thermomechanical simulation of the laser welding process. For the analysis of interest, a reasonable assumption has been adopted by neglecting the mechanical dissipation. Indeed the released strain energy is insignificant in comparison with the energy supplied by the heat source (1): a strain of 1% under 400 MPa brings an overheating of about 1°C. This allows performing a semi coupled thermomechanical analysis, in which the heat generated by inelastic deformation is neglected, but the effects of temperature on thermal and mechanical material properties are included.

In this work, a particular attention is paid to the material mechanical constitutive model. As it is difficult to represent and simulate all the involved phenomena during the welding operation, we focus upon the determination of a simple constitutive model that includes the most relevant phenomena (high temperature, strain rate effect, metallurgical transformations...).

**Thermal considerations**

The unsteady state heat transfer during welding operation can be described by the general heat equation (6) as follows:

\[ \rho C_p \dot{T} - w + \text{div} \ q = 0 \]  \[ \text{[1]} \]

\( \rho \): the material density
\( T \): the temperature and \( \dot{T} \) is time derivative
\( C_p \): the specific heat
\( w \): the heat flux created by the heat source
\( q \) is the heat flux that is related to the temperature \( T \) by the Fourier constitutive law:

\[ q = -\lambda \ \text{grad}T \]  \[ \text{[2]} \]

\( \lambda \): the thermal conductivity.

During the welding operation, several heat transfer modes can be observed:

- Convective flux described by:

\[ q_{\text{conv}} = -\lambda \frac{\partial T}{\partial n} = H(T - T_o)n \]  \[ \text{[3]} \]

\( n \): the outward normal to the surface
\( T_o \): the surrounding temperature
\( H \): the transfer coefficient

- Radiative flux given by the expression:
\[ q_{\text{ray}} = -\lambda \frac{\partial T}{\partial n} = \sigma E_m (T^4 - T_0^4) n \]  

\( \sigma \): the Stefan-Boltzman constant  
\( E_m \): relative emissivity coefficient

\[ q_{\text{cond}} = -\lambda \frac{\partial T}{\partial n} = -\frac{\lambda_c}{e} (T - T_c) n \]

\( \lambda_c \): the thermal conductivity of the support  
\( T_c \): the temperature of the contact element  
\( e \): thickness the contact element

In laser welding simulation, the heat source is chosen to be a Gaussian distribution (see Figure. 2) which is formulated as follows (5):

\[ I = I_0 \exp\left( -\frac{r^2}{r_0^2} \right) \]

with \[ r_0 = f(z) = r_e - (r_e - r_i) \frac{(z_e - z)}{(z_e - z_i)} \]  

where \( I \) is the density of heat quantity and \( I_0 \) is the maximal density of heat quantity

Hence the total power is given by:

\[ P = \frac{1}{3} \pi I_0 (z_e - z_i) (r_e^2 + r_i^2 + r_e r_i) \]

**Mechanical considerations**

For the mechanical constitutive model, we compare a classical elastoplastic model (without hardening) and a rate dependent viscoplastic model. The set of equations associated with these models is briefly recalled in the following section (3):

- The elastic part of the response is governed by the Hook stress-strain relationship given in a rate form:

\[ \sigma = D(\dot{e} - \dot{\varepsilon}^\text{an}) \]
\( \sigma \): the rate of the Cauchy stress

\( D \): the elastic constitutive tensor

\( \dot{\varepsilon} \): the total strain rate

\( \dot{\varepsilon}^{an} \): the rate of the inelastic strain part

- In the case of associated plasticity with a von Mises yielding criteria, the inelastic part of the deformation is defined by the following flow rule:

\[
\dot{\varepsilon}^{an} = \frac{3}{2} \dot{p} \frac{s}{J} \tag{10}
\]

\( \dot{p} \): the rate of the equivalent inelastic strain

\( s \): the deviatoric stress

\( J \): the von Mises equivalent stress

\[ J = \sqrt{\frac{3}{2} : s} \]

- The previously given equations are valid for both elastoplastic and elastoviscoplastic models. The difference between the two models results from the determination of \( \dot{p} \) (7).

**Elastoplastic model:** for the rate independent plastic model, \( \dot{p} \) is obtained by mean of the consistency condition:

\[ J - k = 0 \tag{11} \]

\( k \): the yielding stress

**Viscoplastic model** for the rate dependent viscoplastic model, the consistency condition is replaced by the following constitutive relation:

\[
\dot{p} = \left( \frac{J - R(p) - k}{K} \right)^N \tag{12}
\]

\( R(p) \): the yield surface expansion according to an isotropic hardening model

\[
R(p) = Q_1 p + Q_2 \left( 1 - e^{-bp} \right) \tag{13}
\]

\( p \): the equivalent viscoplastic strain

\( Q_1, Q_2, b \) and \( N \) are material parameters

**ANALYSIS AND DISCUSSION**

**Viscoplastic model validation**

While the elastoplastic constitutive model is available in MSC.Marc software, the viscoplastic one is implemented using The MSC.Marc user subroutine HYPELA (4). To validate this implementation, a relaxation test is investigated and the obtained numerical results are compared with a reference solution. The relaxation test consists in a specimen uniaxial extension hold under constant temperature (300°). Concerning the finite element model used for this test, the specimen is descritized
by mean of 40 solid brick elements (element 7). A full description of the finite element model is given in Figure. 3.

![FE model for relaxation test](image1.png)

Figure. 3: FE model for relaxation test

Figure. 4 gives a comparison between the equivalent von Mises stress from our implementation and a reference solution. It clearly shows good agreement between the two results.

![Comparison of equivalent von Mises stress during relaxation test](image2.png)

Figure. 4: Comparison of equivalent von Mises stress during relaxation test

**T-welding simulation**

The T-welding test investigated in this work is described in Figure. 5. As the problem is symmetric, only one half of the workpiece is discretized by mean of 2055 3D solid elements. The welding simulation is semi coupled since it consists in a heat transfer analysis followed by a stress analysis.
The heat transfer analysis consists in two separate load cases. The welding where the heat source moves from point A to point B and the cooling. The two load cases are associated with convection heat transfer on the external faces of the workpiece.

Figure 6 presents a contour plot of the temperature during welding. It shows that the heat source is assumed to be a normal Gaussian distribution that spreads over several elements.
Figure. 7 shows the temperature distribution along the seam weld at different times.

![Temperature Distribution Graph](image1.png)

Figure. 7: Temperature evolution along the seam weld at different times

The stress analysis uses the previously computed heat data. It consists in three load cases namely welding, cooling and unbridling. The mechanical boundary conditions associated with welding and cooling consist in simple clamping of the vertical and horizontal plate extremities (all nodal degrees of freedom are restrained). For the unbridling, the boundary conditions just consist in rigid body motion removal. Concerning the constitutive model, the stress analysis is carried out using the elastoplastic model and the viscoplastic model. The numerical results from the two constitutive models are compared. It should be pointed out that the material parameter evolution with temperature is taken into account. In addition, the elastoplastic constitutive model and the viscoplastic one use the same Young modulus and the same yielding stress.

In figure. 8, we present the time evolution of the temperature and in figure. 9 we compare the equivalent von Mises stress, at point C, obtained from the two constitutive models. The comparison between these results shows that the strain rate influence is well represented by the viscoplastic model.

![Time Temperature Graph](image2.png)

Figure. 8: Time temperature evolution at point C
Figure. 9: Equivalent von Mises stress

Figure. 10 gives a comparison between the two constitutive models in terms of residual displacements. It can be noticed that the two models give the same final shape but the residual displacements from the elastoplastic model are more important.

Figure. 10: Y displacement distribution by using an elastoplastic law (a) and a viscoplastic law (b)

CONCLUSIONS

A 3D finite element model is proposed for the numerical simulation of laser welding. The semi coupled thermomechanical analysis breaks up into two different phases:

- The heat transfer analysis where the moving heat source (laser) is described by a Gaussian distribution that is integrated in the MSC.Marc software by mean of the user subroutine FLUX.
- The stress analysis that use the heat loading (from heat transfer analysis) and temperature effect on material parameters.

Concerning the mechanical constitutive model, a rate dependent viscoplastic model is implemented in MSC.Marc software by mean of the user subroutine HYPELA. The proposed finite element model is used to investigate a T-welding test where the calculation is carried out using both the viscoplastic model and a simple elastoplastic one. The numerical results from the two models are
compared in terms of stress and residual displacements. It should be pointed out that the two constitutive models use the same Young modulus and the same yielding stress but the comparison between them is only qualitative since the other material parameters, for the viscoplastic model, are not identified from experimental data.

This first work has to be continued in the future and several improvements can be achieved:

- Improving the thermal analysis by integrating the conduction interaction with the support
- Improvement of the mechanical constitutive model and identification of the material parameters
- Integration of the metallurgical transformations in the model
- Experimental validation of the whole numerical procedure

ACKNOWLEDGEMENTS

This work is collaboration with the Common Research Centre of EADS (Suresnes, France).

REFERENCES


