FINITE ELEMENT MODELLING AND ANALYSIS OF RESIDUAL STRESSES IN AL−SIC METAL MATRIX COMPOSITES WITH GID®

A. Andrade−Campos[1,*#], J.A.M. Pinho−da−Cruz[1,#] and F. Teixeira−Dias[1]

[1]*Dept. Mechanical Engineering, University of Aveiro, 3810−193 Aveiro, Portugal*

SUMMARY: Residual stresses are often induced in metal matrix composite materials (MMC) when these are cooled down from fabrication to room temperature. This is mainly due to the difference in the coefficients of thermal expansion of the matrix and reinforcement materials. In the present paper, these stress fields are studied with a fully three−dimensional thermomechanical model. The strength differential in subsequent uniaxial tension from the thermal residual stress state is also analysed. The thermomechanical model implemented to describe these processes considers that the reinforcement component has a thermoelastic behaviour and that the metallic matrix exhibits a thermoelastic−viscoplastic behaviour. The Finite Element analyses presented are based on three−dimensional geometrical composite unit cell models built with the GiD® preprocessor. Results obtained with long cylindrical SiC fibres embedded in an aluminium matrix are presented. All the process data are introduced in a dedicated problem type with a graphical user−friendly input data interface (GUI) built inside GiD®. Postprocessing tasks are also carried out with GiD®, graphically presenting, for example, the triaxial stress state, the equivalent stress, the equivalent plastic deformation rate and the resulting displacement fields in the composite unit cell. The results are then bounded with an analytical model and compared with numerical results from other authors.

KEYWORDS: Preprocessing, postprocessing, metal matrix composites, residual stresses.

INTRODUCTION

Metal matrix composite materials (MMC) have become increasingly attractive in recent years for their high strength and creep resistance properties. Thermally induced residual stresses are developed in MMCs during cooling from the fabrication temperature to room temperature. For discontinuously reinforced MMCs this has often been attributed to the mismatch between the thermal expansion coefficients of the fibre and matrix materials. The subsequent tensile flow stress of the composite is influenced by the thermal residual stresses generated during cooling down from fabrication temperature[1−4].

In the present work, three−dimensional (3D) unit cell Finite Element (FE) models are developed for composite materials with cylindrical reinforcements in order to analyse the magnitude and distribution of the residual stresses that result from the cooling process and their effects on subsequent deformation in tension. 3D FE models were chosen for their advantages over axisymmetric and two−dimensional models in that they consider a more realistic fibre geometry at fibre ends and do not ignore a portion of the matrix material. In this kind of research it is crucial to have an adequate graphical support to built the model,

---

[*] Corresponding author: Tel: +351.234.370830; Fax: +351.234.370953; <gilac@mec.ua.pt>

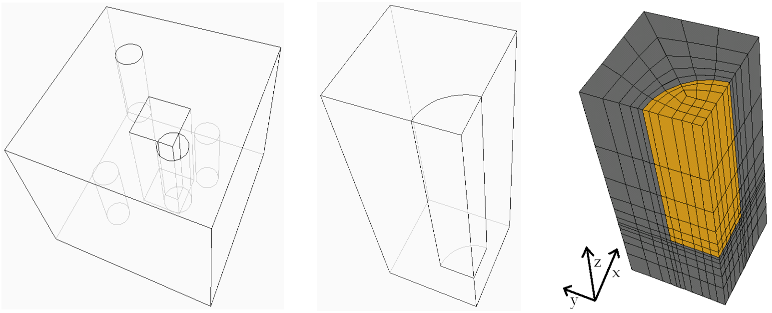
[#] These authors acknowledge financial support given by Portuguese Foundation for Science and Technology.

introduce the process data, execute the analysis modules and analyse results. The authors used the pre/postprocessor GiD$^{®}$ to perform all the graphical tasks. The model was created using GiD$^{®}$'s 3D mesh generator. A dedicated problem type was built in order to accept process input data and, along with the coordinates of the nodes and element conectivities, write into a file to be read by an analysis module. This module calculates residual stress fields and strength differentials in dual phase materials. Post−processing was also performed with GiD$^{®}$.

MODELLING CONSIDERATIONS

It is assumed that all the reinforcement fibres have the same dimensions and orientation and are uniformly distributed. The matrix and fiber materials are isotropic in stiffness and thermal expansion. A perfect bonding between the aluminium matrix and the SiC reinforcement is assumed. The behaviour of the reinforcement is considered to be thermoelastic and the matrix to exhibit a thermoelastic−viscoplastic constitutive behaviour. The temperature field among the composite is assumed homogeneous at all times.

The representative unit cell, shown in figure 1, is hexahedral and includes the cylindrical reinforcement fibre. An orthogonal cartesian coordinate system was used as reference with *Ox*, *Oy* and *Oz* axes aligned with the main dimensions of the unit cell. The longitudinal axis of the reinforcement fibre is aligned with the uniaxial loading direction[2,3].



(a) *(b)* *(c)*

Fig. 1: Representative unit cell (RUC) for the cylindrical reinforcement MMC. (a) Geometrical material model, (b) RUC and (c) finite element mesh.

During the first stage of the analysis, the unit cell is subjected to a thermal load simulating the cooling down from fabrication (493K) to room temperature (293K). On the second step of the analysis, the composite unit cell is submitted to an uniaxial longitudinal tensile load. Symmetry and restraint conditions are the same in both steps. A uniform displacement is applied to all the nodes on the top surface of the unit cell, along direction *Oz*. The model consists of 689 hexahedral trilinear eigth−noded finite elements (fig.1c). The problem type TriCreator is defined by the (i) boundary conditions; (ii) material properties; (iii) material assignment; (iv) time and numerical optimisation data and (v) problem general data.

GRAPHICAL AND NUMERICAL IMPLEMENTATION

The conditions that can be assigned to an entity in the problem type TriCreator are: (i) *Point/Line/*S*urface−Constraints*, restraining the displacements along the *Ox*, *Oy* and/or *Oz*; (ii) *Point/*Surface−*Load*, loading entities as boundary forces; (iii) *Boundary−Displacements*, assigning nodes to move a predefined displacement. These conditions can be assigned either on the geometry and/or on the mesh. It is also possible to change material properties in the *Data Materials* menu box. The *Interval data* window contains the numerical optimisation parameters that control the iterative scheme in the analysis module. Problem data are the initial and final temperatures, cooling rate and total process time.

Once the conditions, materials, problem and interval data are properly assigned, GiD$^{®}$ creates an input file for the analysis module (fig. 2). The solver – Nostradamus − is dedicated to the numerical calculation of both thermally and mechanically induced residual stresses in dual−phase materials. Nostradamus produces two output files. These include gauss points lists, stresses, strains, etc. Another module, designated Sirius, was developed to interface the output files into postprocess files.

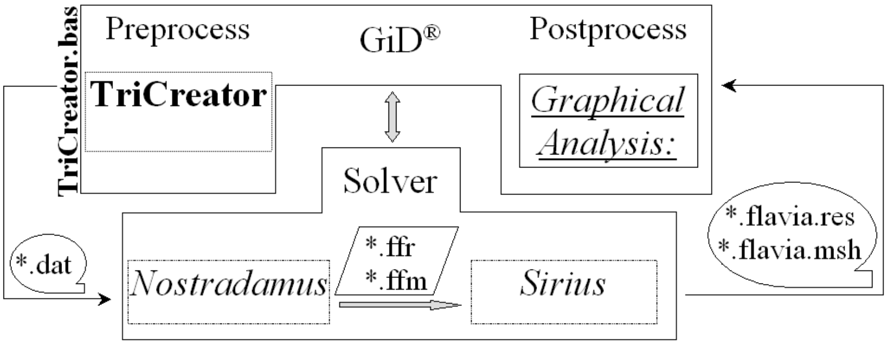


Fig. 2: Schematic representation of the Nostradamus−GiD® interactions.

RESULTS AND POST−PROCESSING

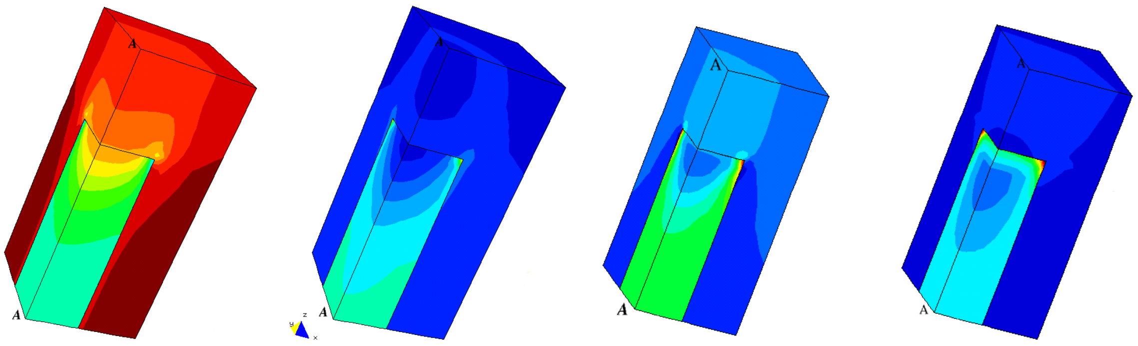
Numerical results were analysed using the post−processor GiD®. After the cooling stage, large stress gradients occur close to the fibre−matrix interface (figs. 3(a) and 3(b)), especially near the top corner of the fibre. The magnitude of the compressive stress is maximum at the fibre−matrix interface and progressively decreases as one moves away from the interface into the matrix. At 1% tensile strain the longitudinal stress, σ_{zz}, evidences high stress gradients near the fibre ends. These stresses are tensile throughout the unit cell, with its the maximum value near the fibre top corners. The von Mises equivalent stress, σ_{eq}, has strong gradients localised in the corner regions and in the fibre−matrix interface. The evolution of the stress fields during the far field loading, up to 1% strain in tension, are shown in figures 3(c) and 3(d).

DISCUSSION

In order to validate the numerical results obtained with the Nostradamus−GiD® system, these were compared with results obtained by another author (Jain et al.[1]), using different mechanical models and bounded with analytical calculations. The chart in figure 4 represents the thermal residual stress and the longitudinal stress profiles along the AA direction (fig. 3). The differences displayed in figure 3 can be attributed to the differences in the constitutive model which Jain et al.[1] consider to be elastic−plastic.

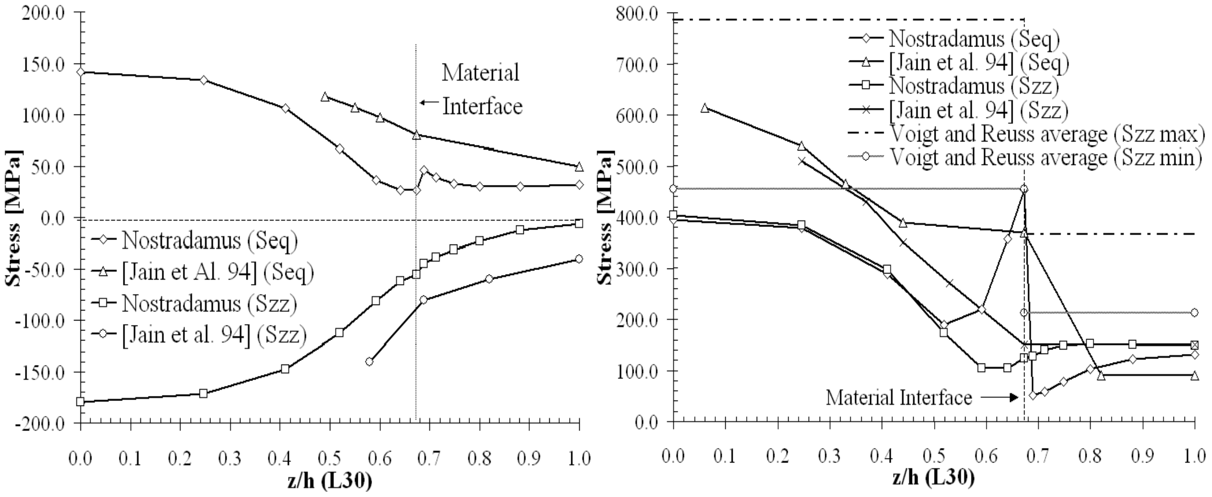
The thermomechanical loading of a composite material unit cell leads to non−uniform local (microscale) stress and strain fields, whose analytical descriptions correspond to complex problems. However, many simplified and useful results can be obtained in terms of average stress and strain values[5]. In this context, the elastic constants of composite materials can be evaluated with the aid of mean field methods. These techniques assume that averaged values of the stress and strain are representative of the behaviour of each material phase.

The earliest applications of mean field methods to compute the elastic constants of composite materials are associated to the works of Voigt and Reuss. These investigators derived simple expressions for the elastic constants of the composite that depend only on the elastic constants and volume fraction of the phases, and were independent of the phase geometry and spatial distribution. These expressions represent, respectively, upper and lower bounds for the composite's stiffness. Based in the Voigt and Reuss average stiffness matrices[6], the values of "local" average elastic stresses at the reinforcement and at matrix materials were obtained considering the total corresponding thermomechanical strains. These average stresses correspond to the Voigt (upper) and Reuss (lower) bounds (figs. 4(b)).



(a) *(b)* *(c)* *(d)*

Fig. 3: Residual Stress fields at room temperature: (a) σ_{zz} and (b) σ_{eq}. Uniaxial stress fields at 1% strain: (c) σ_{zz} and (d) σ_{eq}.



(a) *(b)*

Fig. 4: Stress profiles along direction AA. (a) Thermal residual stresses and (b) strength differential after the initial residual stress state.

CONCLUDING REMARKS

An example of application of GiD® in science and engineering was presented, confirming the importance of the pre and postprocessor in the analysis of the overall behaviour of multiphase materials. Stress gradients were determined in the surrounding area of the fibre−matrix interface and at the top corner of the fibre. The results of the simulations of thermal residual stresses and prediction of the strength differential are in qualitative agreement with other authors work. It is worth to notice that both Voigt and Reuss bounds define stress ranges whose values are superior than the numerical ones. This can be explained by the fact that analytical bounds are inherently elastic, while numerical results are predominantly elastoplastic.

REFERENCES

1. M. Jain, S.R. MacEwen and L. Wu, *Mater. Sci. Eng.* **A183**:111−120, 1994.
2. F. Teixeira−Dias and L.F. Menezes, *Int. J. Numer. Meth. Engng* **50**(3):629−644, 2001.
3. F. Teixeira−Dias and L.F. Menezes, *Comput. Mater. Sci.* **21**(1):26−36, 2001.
4. G. Meijer, F. Ellyin and Z. Xia, *Composites Part B* **31**(1):29−37, 2000.
5. R. Hill, *J. Mech. Phys. Solids* **11**:357−372, 1963.
6. S. Nemat−Nasser and M. Hori, *Micromechanics: Overall Properties of Heterogeneous* Materials, 2nd Rev. Edition, Elsevier Science B. V., 1999.