Computational Materials Science 95 (2014) 288–301

Contents lists available at ScienceDirect

Computational Materials Science

journal homepage: www.elsevier.com/locate/commatsci

A practical flow diagram for the solution of complex non-linear thermo-mechanical numerical models

D.G. Karalis [a,*], N.G. Tsouvalis [b], V.J. Papazoglou [b], D.I. Pantelis [b]

[a] Hellenic Navy, Hellenic Naval Academy, Mechanics & Materials Division, Marine Materials Laboratory, Hazjikyriakou Avenue, Piraeus 185 39, Greece
[b] Shipbuilding Technology Laboratory, School of Naval Architecture and Marine Engineering, National Technical University of Athens, 9 Heroon Polytechniou Avenue, Zografou, Athens 157 73, Greece

ARTICLE INFO

Article history:
Received 13 November 2013
Received in revised form 30 May 2014
Accepted 27 July 2014
Available online 20 August 2014

Keywords:
Thermo-mechanical modelling
Finite element analysis
Convergence flow diagram
Repair welding

ABSTRACT

In this paper the authors propose a practical flow diagram for the systematic development and solution of complex thermo-mechanical finite element analysis models. The proposed diagram consists of three different phases and provides a step-by-step guide for the development of the final thermo-mechanical model, taking into account convergence issues, mesh density and estimation of time step magnitude. In phase I, a preliminary thermo-mechanical analysis is carried out in order to get an idea of the model behaviour, the required resources and the feasibility of the overall analysis. In phase II the final thermal model is developed in full, taking into account the mechanical results obtained at the end of phase I, whereas in phase III the final mechanical model is generated on the basis of a continuously modified thermal model. The proposed procedure presented herein in the form of a flow diagram provides the capability for gradual output of the numerical results (preliminary results, thermal results, mechanical results), while paying attention to the time-consuming problem of results convergence required for a numerically accurate analysis. The former is an important issue for large-scale complex simulation projects, whereas the latter provides evidence that the development of the numerical model has been realized on the basis of the modelling laws. For better presentation and understanding, the proposed procedure is applied to the study of a finite element analysis thermo-mechanical model, where increased intricacy generally exists.

1. The problem of modelling consecutive phenomena

1.1. Introduction

The thermo-mechanical response of steel or aluminium plates during welding or plate forming by line heating has been investigated by several researchers during the last decades. Most of the research is focused on either or both the thermal and the mechanical part of the structural response through a combination of experimental and numerical simulations. The numerical part of the investigation still attracts high interest due to its extreme intricacy and the uncertainty in predicting the structural response prior to the treatment (welding or line heating) itself. An extensive review has been conducted in [1–4].

In a fully uncoupled thermo-mechanical finite element model, the analysis is usually carried out in a staggered approach: the thermal problem is solved first, followed by the solution of the mechanical problem. The latter mechanical analysis runs on the basis of the thermal results in order to account for the thermal stress and phase change effects on the structural response of the structure. This is performed by importing to the mechanical model the nodal temperatures at each time increment and calculating the thermal strain. From the aforementioned staggered approach it is deduced that both thermal and mechanical models must normally run with the same analysis parameters, namely time step magnitude and mesh density. If, for example, the material undergoes phase transformation accompanied by volume change during a specific short temperature range, a small constant time step and a fine mesh are required in the areas of transformation for both the thermal and the mechanical analysis. This allows for the accurate monitoring of the transient stress developed during the transformation temperature range [5]. Thus, for the entire analysis there should be an exact correspondence between the mesh density and the time step magnitude between the two models. This requirement renders the whole procedure of model development very complex and time-consuming.

1.2. The three major problems: mesh density, time step and convergence of results

The first problem arising during the thermo-mechanical modelling is that the thermal and the mechanical models are completely

* Corresponding author. Tel.: +30 210 45 81 656, +30 697 37 97 661.
E-mail address: KaralisDimitris@TEEmail.gr (D.G. Karalis).

http://dx.doi.org/10.1016/j.commatsci.2014.07.045
0927-0256/

different in nature, as they model different physical phenomena. Therefore the mesh density selected for the solution of the thermal problem is, in most cases, inappropriate for the solution of the mechanical problem.

Secondly, the time step required for the accurate solution of the mechanical analysis may be too large compared to the time step required for the accurate solution of the heat flow problem, where, for example, extreme temperature gradients are encountered. The latter is also valid in the opposite case as, at high temperatures, the structure may exhibit extreme material non-linearities.

A third problem pertains to the results' convergence criteria. The development of a numerical model by means of the finite element method is generally terminated when the analysis has reached (a level of) results' convergence. For example, classical convergence criteria are based on the stabilization of nodal results, such as temperatures or displacements with regard to mesh density and time step. It is actually not worthy remeshing the model or reducing the time step if the nodal results do not change values versus simulated time.

It should be emphasized at this point that in general there are four types of convergence in finite element analyses:

i. convergence of equilibrium iterations due to non-linearities (e.g. material, contact or geometrical non-linearities),
ii. convergence in the solutions of the linearized systems of algebraic equations in case of iterative solvers,
iii. convergence of the results due to mesh refinement and
iv. convergence of the results due to time step reduction.

In most commercial finite element software platforms, specific optimum values and tolerances are already pre-set in order to control best the convergence of the equilibrium iterations due to non-linearities and convergence of the equations in case of iterative solvers. In the present study, emphasis is given only to the last two convergence types, namely time step and mesh refinement, as they are the main user-dependent parameters that strongly influence the entire simulation and results' convergence. The procedure followed towards the convergence of results governs directly the overall simulated time, the numerical analysis cost and affects the accuracy of the results. For example, some of the complex simulations presented in [6,7] have lasted a few days, time that could have been strongly increased if a few more additional analyses have been required due to convergence issues. At this point, it should be mentioned that in most publications dealing with complex thermo-mechanical simulations the convergence criteria have not been described at all, as the authors provide only the model's setup and the numerical results. Hence, the end reader of the aforementioned papers comes to understand that the authors have somehow performed a convergence analysis prior to publishing the results obtained by means of the finite element method. This convergence analysis is of great interest as it is complicated, time-consuming and strongly user-dependent.

In sum, a common time step and mesh density are normally required for both the thermal and the mechanical analysis. These two common parameters must allow both physical problems to be modelled satisfactorily, but they must also provide an acceptable level of results convergence for both models.

1.3. The aim of this paper

The authors aim at proposing a practical flow diagram for the systematic development and solution of complex FEA thermo-mechanical models. In this flow diagram a progressive development of several thermal and mechanical models will be presented on the basis of different mesh densities and time steps, aiming at reaching the convergence of the thermal and mechanical results.

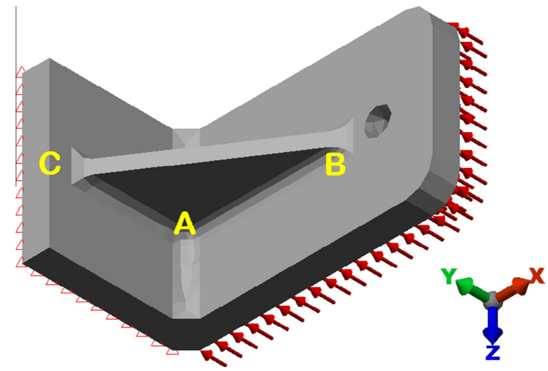

Fig. 1. The welded bracket used as an example to present the flow diagram.

It ought to be mentioned here that a flow diagram for the solution of such staggered thermo-mechanical models is missing from the international literature and that the whole process is a real labyrinth for both experienced and inexperienced users dealing with thermo-mechanical modelling. Please note that the aim of the authors is to discuss the proposed flow diagram and present the steps followed for creating the final thermo-mechanical model with regard to mesh density and selection of time step and not to provide the mathematically-based analysis for its development. The latter has already been discussed in the literature [8–35]. The implementation of the proposed flow diagram requires a commercial thermal and mechanical or multi-physics FEA software package for which code verification has been already performed.

2. A typical example to explain the flow diagram

2.1. The physical model

In order to discuss the proposed flow diagram, a thermo-mechanical simulation will be employed. The latter concerns the weld treatment of a welded bracket under load, by means of tungsten inert gas (TIG) welding. The whole configuration of the simulation is presented in Fig. 1.

The bracket shown in Fig. 1 is made of typical carbon structural steel (containing 0.45% w/w carbon) and consists of a bent flange and a triangular reinforcing web welded on the flange. The welds A–C and A–B exist along both sides of the web. Treatment is performed along the A–B weld on the side towards the +z semi-axis (the one that is visible in Fig. 1) using a TIG torch without filler metal and aims at treating the existing weld close to the melting temperature. Such treatments are applied to repair in-situ cracked or defected welds (repair welding). The welded bracket is fixed at its smaller side (see red[1] triangles in Fig. 1 that refer to the fixations). After the material has cooled to ambient temperature, uniform pressure is applied on the other side of the bracket (see red arrows in Fig. 1) tending to buckle the triangular reinforcing web. The latter pressure simulates the operational load present on the bracket after the completion of the treatment. The treated length l_{AB} is equal to 128 mm, whereas the flange and web thicknesses are equal to 25 mm and 12.5 mm respectively. The power of the welding torch was set equal to $Q = 3770$ W whereas the speed was set equal to $v = 6$ mm/s. This simulation is quite complex involving the existence of extreme non-linearities as temperatures are raised to the steel melting point. It represents a difficult-to-solve numerical analysis, as thermal, mechanical and thermally-induced mechanical

[1] For interpretation of color in Fig. 1, the reader is referred to the web version of this article.