COST 526

"Automatic Process Optimization in Materials Technology"

(APOMAT)

Half-Yearly Report

To be sent to **V.Tesch@access.rwth-aachen.de** until **February 29, 2004**

1. Reporting Period	**1.7.2003 – 31.12.2003**
Project title	Optimization of Fatigue Resistance of Cold Forging Tools by Considering Damage Mechanisms at Micro Scale
Project leader	Dr. Igor Gresovnik
Organization	C3M
Main collaborators involved	Faculty of Natural Sciences and Technology, University of Ljubljana, Slovenia.

2. Funding Situation	
Amount of money received specifically for COST	2.9 kEuros
Other resources partially used for the project	kEuros

3. International Collaboration (mention group and type of work done in collaboration during the reporting period)
Participation in the Working Group Meeting in Krakow + project progress report ❒ YES ➨ ❒ NO
Rockfield Software, Swansea, discussion and improvement of the direct interface between the optimisation shell and the simulation programme "Elfen" (final definition of the core part, clean-up). NTF, Ljubljana – discussion of the test multi-scale optimization example LMT Cachan, preparation of the software background for the multi-scale example (definition of solution procedure, implementation in FEAP, parallelization of micro-level simulations, setting up computational environment).

4. Industry participation (mention name of companies and work done in collaboration during the whole project)
Iskra-Avtoelektrika. Analysis of service life of tooling systems

5. Meetings, visits, exchange of scientists, short-term scientific missions	**Location, date**
Working Group Meeting	Krakow, November 2003

6. Progress within the reporting period

(Not exceeding 3 pages, including tables and figures)

Work was concentrated around the definition of a test case for shape optimization. In accordance with project objectives, it was decided that the test case would incorporate multi-scale simulation approach. This would enable to identify potential difficulties in combining optimization with multi-scale simulation approaches.

A substantial amount of work has been done previously in collaboration between NTF and LMT on multi-scale modeling with strong coupling between scales. Since this work was implemented in the finite element program FEAP, this software was chosen as a basic simulation platform. This implied the necessity of implementing an interface between FEAP and the optimization shell. Some ideas adopted previously at constructing an interface with the commercial code Elfen were used, however a simpler version of interface was implemented providing only the minimal means of interaction between the two programs necessary to solve the example.

Multi-scale finite element analysis usually require long computational times – a possibly prohibitive factor for combining this type of analysis with optimization. The idea was formed to overcome this problem by parallelization of the computation at the microscopic level, for which the utilized multi-scale simulation model is well suited. Parallelization was implemented on top of the parallel interface adopted in PLATON – a Problem Solving Environment (PSE) for distributed numerical optimisation, developed at Technical University Braunschweig. The hardware basis chosen for solution of the problem was a cluster of Linux machines connected in LAN. The optimization shell had to be ported to the Linux platform for this reason, as it has been mainly developed on MS Windows. The task of porting turned straightforward except for some external libraries used by the shell for tasks such as visualization of results. In order to avoid unnecessary deadlocks, the critical parts were excluded from Linux version of the shell and the loss of functionality has been bridged by using the Windows version in parallel.

The test case under consideration includes optimization of shape of inclusions in a periodic microstructure with respect to given criteria concerning the overall response of a specimen under a prescribed loading. The problem is schematically depicted in figure 1. A specimen supported at its bottom ends is loaded by a vertical force acting in the middle of its topper surface. The specimen consists of periodic inclusions of a harder material incorporated in a softer matrix. The task will be to optimize the shape and orientation of these inclusions by taking into account criteria that incorporate the deflection of the specimen, work of external forces and energy dissipation.

In the figure, the discretization of the macroscopic model is indicated by thicker lines. For each macroscopic element, a complete microscopic finite element model is built, consisting of sufficient number of periodic cells to achieve satisfactory accuracy of the computed response. Periodic cells are delimited by thinner lines in the figure. The finite element discretization of the microscopic model is not shown. The right-hand side of the figure shows a single periodic cell of the microscopic model, with a harder inclusion inserted in a matrix material. The intended parameterization of the inclusion capable of producing inclined elliptical shapes is indicated.

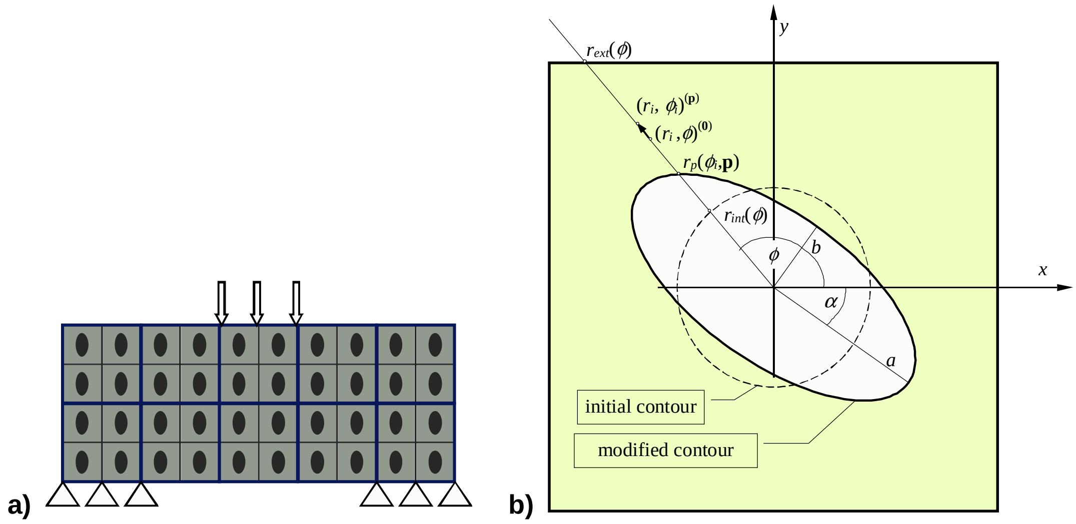

Figure 1: a) Studied structure under loading and b) example of three parametric description of the shape of inclusions within microscopic periodic cells.

Computation of the response of the structure is performed by a two scale finite element model with strongly coupled scales. This allows for more accurate computation in the case where the scale effect is expressed, i.e. where the size of the structure would affect the macroscopic response. On the other hand, the computation of the microscopic response can be replaced by a homogenized phenomenological model in those macroscopic elements where the contribution to the total response is less significant, or the internal state is such that a homogeneous model is accurate enough. We hope that significant improvement of the speed of computation can be achieved in practical applications on this account.

Some conceptual details of parameterization are still under discussion. Two possible approaches envisaged include “exact shape representation”, where the finite element mesh matches the boundary of inclusions, and the “fixed mesh” approach as its alternative. In the latter approach material properties could vary within elements dependent on the actual position of the inclusion-matrix interface. Only the exact mesh representation has been implemented and tested so far. The direct shape parameterization approach was adopted where new shapes are generated by applying parameter dependent maps to a reference configuration. This approach is essentially similar as the approach that has been considered previously for definition of outer object boundaries for the purpose of optimization. However, some additional difficulties arise because the boundary between two materials is considered rather than the external boundary of an object. Actual realization of the shape transform is therefore different and some auxiliary procedures must be applied to enable robust and efficient computation. These procedures will be further improved in the course of the project and more detailed treatment of the test case will be performed.

7. List of publications

a) Published

b) Submitted for publications
c) In preparation