University of Nottingham
School of Mechanical, Materials, Manufacturing Engineering & Management

Project:	**FENET EU Thematic Network** (Contract G1RT-CT-2001-05034)
Report Title:	**Post-Workshop Report on Durability & Life Extension** **FENET Workshop - DLE-04** **Finite Element Simulation of Welds and Joints** **27-28 February 2003, Barcelona, Spain**
Author:	**Prof. Adib Becker** **University of Nottingham, UK** **FENET RTD Coordinator (Durability & Life Extension)**
Date:	**15 April 2003**
Report No.	**FENET-UNOTT-DLE-04**

1. Introduction

The first three workshops on the FENET technology theme of Durability and Life Extension have dealt with contact analysis (Copenhagen, February 2002), structural integrity (Zurich, June 2002) and fracture (Trieste, September 2002). Discussions in these workshops have identified the analysis of welds and joints as challenging across many industry sectors, and have highlighted several difficulties in modelling the real-life behaviour of welds and joints.

2. Workshop Objectives

The main objectives of this workshop were:

- To provide an overview of current practice in modelling welds and joints using FE analysis
- To identify current limitations and difficulties in modelling welds and joints
- To provide a forum for discussion on "Guidelines" and "Best Practice" on using FE in weld and joints
- To present current techniques for modelling residual stresses and cracks in welds and joints
- To discuss the need for benchmarks for welds and joints

3. List of Presentations

A total of 8 presentations were delivered in this workshop by speakers from 4 European countries (Czech Republic, Netherlands, Italy, United Kingdom). Presenters represented university research groups, industry, a general purpose FE software vendor, a specialist weld simulation software vendor. Abstracts of the presentations are shown in Appendix A.

The workshop was chaired by Professor Adib Becker (University of Nottingham, UK). Discussions were invited after each presentation, with a final discussion form held after the last presentation. The topics covered in this workshop included the following:

- Ideas for benchmarks on joints (round-robin exercise)
- Use of general-purpose FE software (ABAQUS) in simulating welds
- Use of Specialist software (SYSWELD) and phase transformation data to simulate the weld process
- Modelling 3D thermal and residual stress models using simplified 2D models
- Analysis of structural integrity of pipeline welds
- Creep and damage in welds and weld repairs
- FE analysis of joints in aircraft structures
- FE analysis and experimental analysis of Electron beam welding

Day 1 – Thursday 27 February 2003

Title of Presentation	Speaker
FENET-DLE Workshops: Introduction and Objectives	Prof. Adib Becker University of Nottingham, UK
Finite Element Modelling of Features Common in Fabricated Plate/Shell Construction	Dr. Jim Wood Strathclyde University, UK
Simulation of welding processes and modeling of welded joints using ABAQUS	Frans Peeters ABAQUS Europe, The Netherlands
SYSWELD – Complete Finite Element Solutions for Simulation of Welding Processes	Josef Tejc MECAS ESI Plzen, Czech Republic
A numerical simplified model to predict the thermal load and residual stresses of welded joints	P. Ferro, F. Bonollo, and A. Tiziani University of Padova, Italy

Day 2 - Friday 28 February 2003

Title of Presentation	Speaker
Comparison of Predicted and Measured Residual Stresses In a Pipeline Girth Weld	Keith Wright and Vinod Chauhan Structural Integrity Assessments Ltd, / Advantica Technologies Ltd, UK
FE Analysis of Creep of Welds and Weld Repairs - Strategies and Difficulties	A.A. Becker, T.H. Hyde and W. Sun University of Nottingham, UK
Joints modelling by means of FEA for evaluation of influence of surrounding structure	Radek Doubrava Aeronautical Research and Test Institute, Prague, Czech Republic
Numerical and experimental analysis of the electron beam welding of a Nickel based superalloy	P. Ferro, A. Zambon and F. Bonollo University of Padova, Italy
FENET-DLE Workshop: Concluding Remarks/General Discussion	Prof. Adib Becker University of Nottingham, UK

4. Analysis of Workshop

- There are many difficulties/challenges in modelling weld processes and analysing weld behaviour.
- Obtaining accurate material parameters for use in FE models of welds is often difficult. The material properties of narrow Heat-Affected Zones are particularly difficult to obtain in laboratory tests.

- FE models of large 3D welded structures require very fine meshes and can be impractical for structures containing welds.
- Residual stresses are difficult to calculate and difficult to implement in FE analysis.
- General purpose FE codes are capable of analysing heat transfer and thermal stresses caused by the weld process, but require specialist subroutines and user interaction to simulate the welding process.
- Specialist FE weld simulation software incorporate built-in complex phase transformation data which enable users to simulate the welding process.

5. Concluding Remarks

This workshop has stimulated many discussions on the latest advances in the simulation of the welding process and the structural integrity of welds. There is a need for further research to incorporate the effects of residual stresses and crack propagation. Guidelines, best practice and benchmarks for modelling structures containing welds are also needed.

The next DLE workshop (Modelling Fatigue of Metals) will be held in Noordwijk, The Netherlands, on 9 - 10 Oct. 2003. The workshop will address the use of FE technology in the analysis of fatigue of metals, current limitations and difficulties and will provide a forum for discussion on “Best Practice” and benchmarks.

.

Appendix A
Summaries of Workshop Presentations

1. Finite Element Modelling of Features Common in Fabricated Plate/Shell Construction
Dr. Jim Wood
Strathclyde University, UK

The aim of this presentation is to examine the procedures used in the various FENET industry sectors, for the "routine" modelling and assessment of common fabrication details in plate/shell construction. In the first instance, attention will be given to procedures appropriate to a design environment, as opposed to assessment as a result of in-service failure. In the latter case it is not uncommon for a higher degree of complexity and sophistication to be employed in the modelling process.

The unique forum provided by FENET is an ideal basis for the dissemination of "best practice", in this area of wide applicability. Several relatively simple fabrication details will be selected for examination. The details will be simple enough to allow ready examination of the problem characteristics and variables, which present the inherent challenges in modelling and assessment. It is proposed that this general topic area forms the basis of the following FENET workshops, with a view to identifying "best practice".

2. Simulation of welding processes and modeling of welded joints using ABAQUS
Frans Peeters
ABAQUS Europe, The Netherlands

General purpose FE codes, such as ABAQUS, can be used to simulate welded structures. This requires the use of full or sequential coupled thermo-mechanical analysis, complex non-linear material behaviour and careful input of the transient heat transfer process, solidification. For weld simulation, users should use accurate temperarture- dependent material data.

3. SYSWELD – Complete Finite Element Solution for Simulation of Welding Processes
Josef Tejc
MECAS ESI s.r.o., Plzen, Czech Republic

This contribution deals with presentation of SYSWELD – commercial FEM software produced by ESI GROUP devoted to simulation of welding and heat treatment processes. To fit well with the scope of the workshop, only the features concerning welding simulation capabilities will be presented.

Field of application and structure of the code will be explained briefly. Further, basic skills of the software concerning the welding process simulation will be described and as well its specific demands on FE modelling (mesh creation, material data, etc.) will be explained.
For better understanding of the subject, an example of Sysweld simulation of a steel plate welding will be shown, with consequent discussion of obtained results (temperature field, metallurgical phases distribution, post-welding strains and residual stresses). Next a more complex application of Sysweld on a real industrial part will be presented.

4. A numerical simplified model to predict the thermal load and residual stresses of welded joints
P. Ferro, F. Bonollo, A. Tiziani
University of Padova, Italy

The finite element method is one of the most attractive approaches for computing residual stresses in welded joints but the enormous computational size of any practical 3D model makes it very inefficient. In this work the development of a 'one dimensional problem' was carried out, in order to obtain a more efficient FE model of the welding process. The temperature distributions were calculated taking into account the heat transfer balance and welding speed. In order to validate this method, temperature history and residual stresses were compared with a 3D FE model and experimental and analytical results. The analyses in this investigation are performed with the finite element codes ANSYS 5.7® and SYSWELD 2000®.

5. Comparison of Predicted and Measured Residual Stresses in a Pipeline Girth Weld

Keith Wright and Vinod Chauhan
Structural Integrity Assessments Ltd / Advantica Technologies Ltd, UK

The purpose of this presentation is to discuss the use of Finite Element Analysis in the prediction of pipeline girth weld residual stresses and to compare the predictions with some experimental measurements. The presentation will conclude by identifying some possible future workshop activities for the Durability and Life Extension technology area of FENET. In particular, the effects of hydrotesting on the residual stresses both predicted and experimental are compared. The initial use of some simplified modelling techniques will be described and the shortcomings highlighted.

6. FE Analysis of Creep of Welds and Weld Repairs - Strategies and Difficulties

A.A. Becker, T.H. Hyde and W. Sun
University of Nottingham, UK

Repair welding is often necessary on operating components to ensure their long term integrity, particularly at high temperatures. In-service inspections of the main steam piping systems in power plant can often reveal cavitation and cracking in or in the vicinity of the welds. If the damage is severe, the components may be replaced. However, in many circumstances, damage is local and therefore weld repairs can be made in order to reduce the cost whilst maintaining safety. Due to the complex nature of the problem, Finite Element (FE) analysis is often used to assess the weld repair and to predict the failure lives. Two main types of material behaviour models are often used in the FE modelling; simple power law creep constitutive laws and more complex continuum damage mechanics constitutive equations. However, before performing an accurate FE analysis, the material properties for each material zone must be accurately determined.

In this presentation, weld repair models, creep and continuum damage constitutive equations used for determining the material properties are briefly described. Typical results of stress analysis and life prediction for the repaired welds are presented. The potential uses and limitations of the FE modelling of creep of weld repairs are discussed.

7. Joints modelling by means of FEA for evaluation of influence of surrounding structure

Radek Doubrava
Aeronautical Research and Test Institute, Prague, Czech Republic

Most structures contain a considerable amount of various joint elements. These joints are typical sources of stress concentration and origins of fatigue failure during the operation of the structure. Typical joints in aircraft structures are rivets. A large number of publications involving analytical descriptions of this type of joint were published in the past. The results from these analyses form a good basis for application in modern numerical codes for stress analyses. The technique based on the spring element of the FEA code MSC/NASTRAN has been developed in VZLU (Aeronautical Research and Test Institute). It arises from the requirement to include the influence of the surrounding structure (such as the skin of aircraft) on the fracture characteristic calculation on the crack tip in the primary part of structure (such as flange plate, stringer etc.). From this point of view, it is not possible to perform detailed modelling of the joint with contact elements. The technique for detailed modelling of joint element loading and the influence of load transfer based on using contact elements of MSC/NASTRAN has been developed in VZLU, and it has been used on detailed static models of the root part of wing main span of small sport aircraft. The joint stiffness and geometry were optimised on the basis of these results. This optimisation leads to enhancement in life and safety.

8. Numerical and experimental analysis of the electron beam welding of a Nickel based superalloy

P. Ferro, A. Zambon and F. Bonollo
University of Padova, Italy

Electron-beam welding is commonly employed in the aeroengine industry for the welding of high integrity components, fabricated from high-strength superalloys. For such applications it is important to predict distortions and residual stresses induced by this process. In this work, a finite element process model has been designed using an uncoupled thermal-mechanical analysis. The heat source was modelled using a superposition of a spherical and conical shape heat source with a Gaussian power density distribution in order to model the nail shape of the fusion zone. The parameters of the source were chosen so that the model gives good agreement with experimentally determined weld pool shapes and temperature thermocouple measurements. Subsequently, the thermal analysis was used to drive the non-linear mechanical analysis. The predicted residual stresses were then compared with X-ray diffraction measurements. The analyses in this investigation are performed with the finite element code SYSWELD 2000®.