SIMULATION OF STRUCTURAL BEHAVIOUR OF FIBRE METAL LAMINATE JOINTS

Peter Linde[1], Henk de Boer[2], Jürgen Pleitner[1]
[1]Airbus, Kreetslag 10, DE-21129 Hamburg
[2]ALE, Advanced Lightweight Engineering, Rotterdamseweg 145, NL-2628 Delft

Fibre Metal Laminates, Fastener Model, Joint, Numerical Modelling

Abstract

In recent years Fibre Metal Laminates (FML) have been developed as an alternative to monolithical aluminium for use in aircraft structures. FML consists of a built-up material, comprising thin layers of aluminium alternated by layers of glass fibre reinforced epoxy [1]. The built-up material displays advantageous properties in terms of fatigue, corrosion and fire resistance. This paper deals with the development of numerical models for simulating the damage behaviour of joints in fibre metal laminates. The focus is placed on joints fastened by rivets, being numerically challenging, and still highly relevant in several instances for materials such as FML and CFRP.

A detailed model for FML skin based on the finite element method has been developed and validated extensively [2, 3]. In this model each layer is discretized by solid elements and damage mechanisms, such as matrix cracking, fibre failure and interlaminar delamination, are treated separately.

A model for damage behaviour of rivets comprising plasticity and tensile failure has been additionally developed. In combination with the model for the skin simulations of the structural behaviour of riveted lap joints were carried out. Experimental tests have been performed to validate the model.

In this paper the failure models of the skin material and rivets are presented, followed by an introduction to experimental tests of lap joints. Numerical models of the joint tests are created and simulation of the static behaviour of the joints is introduced. Thereby both unloading and reloading behaviour is studied, and comparison with the experimental data is performed. Finally a summary and conclusions are provided and recommendations for future research are given.

1. Introduction to Modelling of Fastener Connections

Numerical modelling of fasteners for simulation of structural behaviour may be carried out at different levels of detail. From the literature three common levels of detail may be identified as follows:

Microscopic fastener models: these models are typically built up by solid elements out of which a considerable number are used to model one single fastener. The surrounding material will typically as well be modelled by solid elements. This kind of fastener model is used for the study of the detailed damage behaviour. All physical effects occurring in reality can be incorporated in these models.

A second level is represented by meso models, which may consist of e.g. beam elements with several degrees of freedom at each node. Some of the physical effects may be included, such as pre stress, fastener tolerance, etc.

Macroscopic models, commonly offered as „fasteners“ by commercial finite element programs, represent the last level. They typically consist of a bar element etc., and do commonly not display failure criteria other than simplified ones.

The focus of this paper is placed on microscopic modelling of fasteners. With increasingly complex skin materials, such as hybrid composites like Fibre Metal Laminates, and carbon fibre composites, failure modes may be manifold.

This together with the fact that fasteners continue to play an important role in joining these new materials alongside alternative joining technologies, constitutes a reason for developing a fastener model capable of providing detailed information of its static damage behaviour until final failure.

This paper is organized as follows. At first the model of the fastener itself, based on solid elements, is presented, followed by description of damage and the modelling of the squeezing.

The modelling of the surrounding skin is then briefly summarized followed by numerical examples comprising validation of the model. Finally a summary and conclusions are presented.

2. Modelling of Fasteners

The fastener model treated here has been developed within the framework of a research program FIMELAS (Fiber Metal Laminate Simulation) carried out during 2003 through 2005 in cooperation between Airbus, ALE - Advanced Lightweight Engineering, Delft, and the Technical University of Delft, [4].

The purpose of the fastener model is to include the behaviour of spliced areas involving the fibre metal laminates and to study the complex damage mechanism that may occur in the skin, around the fastener and in the fastener itself.

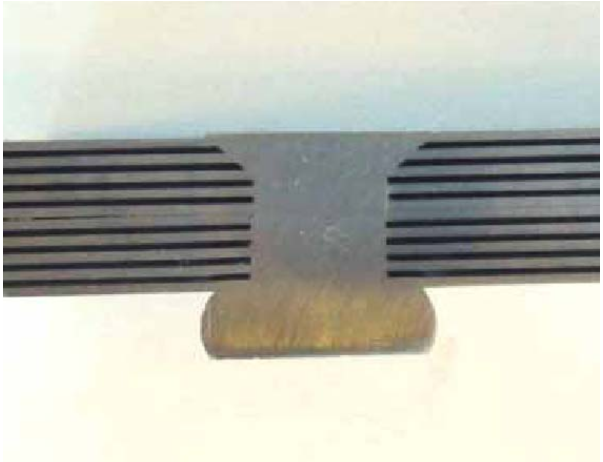


Fig. 1: Cross section of an aluminium rivet in fibre metal laminate

Amongst fasteners considered are: aluminium rivets, lock bolts, titanium bolts, etc. This paper will focus on aluminium rivets, which are squeezed into their final position by a squeezing tool. A typical cross section of such a fastener is shown in figure 1.

The fastener is modelled by solid elements. The development has been based on the finite element program ABAQUS, just like for the development of the skin models introduced later. Damage is considered to occur within the fastener and is described in the following section.

3. Damage Modelling for Fastener

The model allows for fastener damage mainly represented by principal stress tensile failure of the aluminium (tension or shear load on the fastener), or plastic damage in compression.

Two models for fastener damage were developed during the course of the research program mentioned above.

The first model treats tensile failure by tensile cracking, implemented in a user subroutine in ABAQUS, based on the smeared crack concept. This comprises a tensile crack in the material perpendicular to the principal stress direction. Upon crack opening, the crack may develop in its depth and its width. Over the integration point that is cracked, the crack opening is working in a “smeared” manner, thus the designation the “smeared crack model”.

The most common type of the smeared crack models is the so-called “fixed” smeared crack model, which indicates that the crack angle stays as it was at the crack opening. The crack can potentially carry some shear across it due to “teeth” action, and this is numerically represented by a “shear retention factor”, usually taken as some percent of the former full shear stiffness in the material stiffness matrix.

The second damage model employed is a plasticity model as provided in ABAQUS.

4. Simulation of Squeezing

Experimental tests have shown that the structural behaviour of a squeezed aluminium rivet is highly dependent on its squeezing force.

Its internal behaviour must include the stress fields actually present after the squeezing has been carried out. Thus without simulating this effect realistically, there exists no large hope of obtaining a useful model. In literature, attempts have been made to simulate the squeezing by applying a virtual temperature. However, it is needless to mention that these models will not function properly when actual conditions including different temperatures must be simulated.

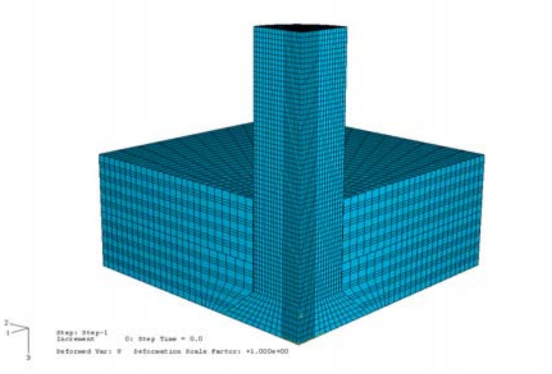
The high deformability of the model using ABAQUS/Explicit proved useful here and it was used simulating the squeezing sequence of the rivet. In the modelling, the unsqueezed rivet is at first modelled with its size and its geometry as is. The sheets to be riveted together are equally modelled with the initial rivet hole and thickness as is. Contact surfaces are defined on both the rivet and in the hole of the sheets.

The load step of the squeezing of the rivet into its final place using ABAQUS/Explicit is shown with deformed shapes in figure 2a through 2c.

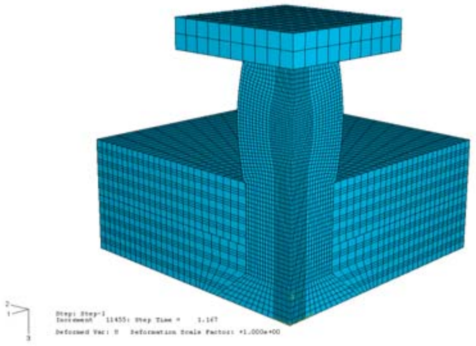
By this, a realistic shape of the squeezed rivet was obtained, and the subsequent structural behaviour also for the first time became realistic.

The pre-stress effect as well as the radial pressure were examined for different rivets applied to different sheets, and were found to be realistic [3].

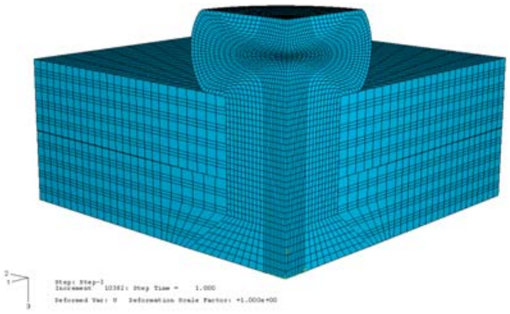
Recent versions of ABAQUS provide the “Contact Fit” option, letting surfaces enter into close contact with each other. This option was used, in combination with “Bolt Load” to obtain desired pre-stress level, as an alternative to the explicit simulation in obtaining the squeezing effect of the fastener.



a) undeformed shape of rivet



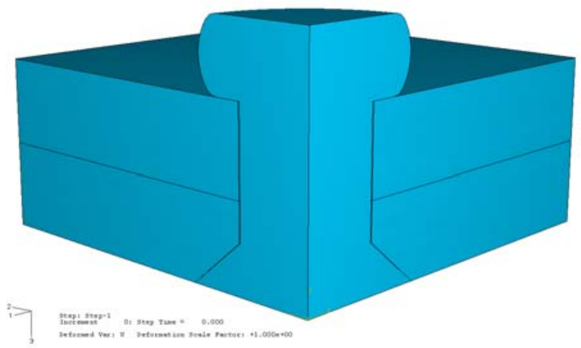
b) initial deformation



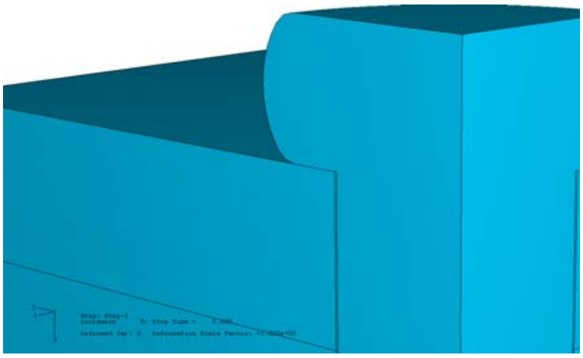
c) final deformed shape

Figure 2. Squeezing sequence by explicit numerical model

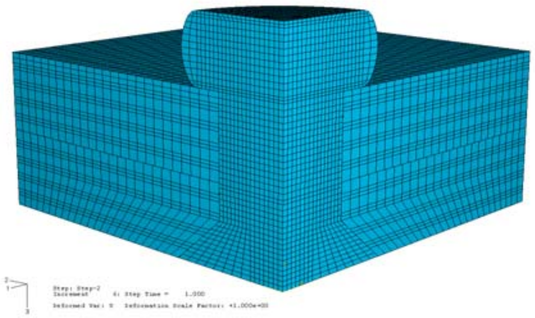
Figure 3 shows a sequence using the contact fit option. The contact fit model proved to deliver realistic solutions with a low computational effort compared to the explicit computation. A comparison between the final states obtained from the explicit computation and the contact fit



a) Underformed contact fit model



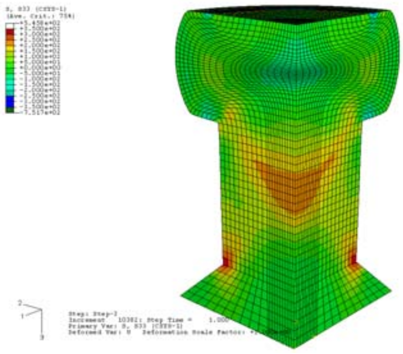
b) close-up of undeformed contact fit model



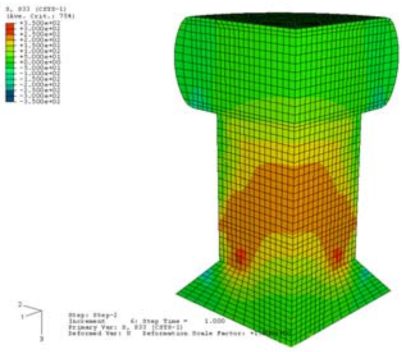
c) final deformed shape

Figure 3. Squeezing sequence by contact fit numerical model

model is shown in Figure 4, displaying residual stresses. A visible difference is the deformed mesh in the head, however of hardly any significance to the results here, since it lies outside the major stress paths. The stresses obtained at the beginning of the conical part and in the center of the rivet are of comparable magnitudes.

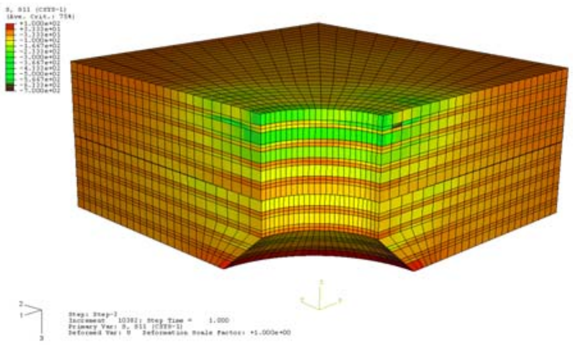


a) Residual stresses after explicit computation

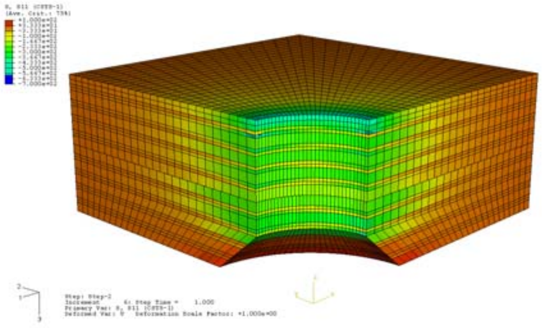


b) Residual stresses after contact fit

Figure 4. Comparison of residual stresses in rivet at final rivet state



a) Radial stresses after explicit computation



b) Radial stresses after contact fit

Figure 5. Comparison of radial stresses at final rivet state

The radial stresses are shown in figure 5 for both methods of simulating the squeezing, displaying comparable magnitudes.

5. Skin Modelling

The skin model used together with the detailed rivet model described here will be summarized only. For a detailed description, see [4].

For laminated materials, each layer is modelled by separate solid elements. The interface between the elements is modelled by a user interface, a development equally performed based on an ABAQUS user subroutine [5]. This allows for interlaminar delamination between the layers. The material in prepreg layers includes fibre failure and matrix cracking. For hybrid composites such as FML, the metal layers would include plasticity. The failure modes are summarized in figure 6.

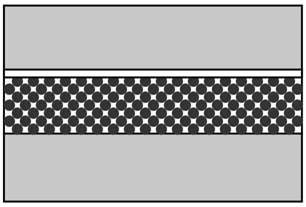
The delamination behaviour is described through a failure criterion based on relative displacements [6]. Failure in this model is related to the equivalent relative displacement, κ, defined by

$$\kappa = \sqrt{u_1^2 + \left(\frac{u_{ft}}{u_{fs}}\right)^2 u_2^2 + \left(\frac{u_{ft}}{u_{fs}}\right)^2 u_3^2}$$

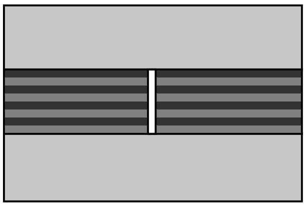
where u_{ft} is the gap opening displacement leading to failure and u_{fs} denotes the maximum shear displacement. Failure will occur when $\kappa > u_{ft}$. Notice that the failure function is based on relative displacements. The stiffness of the interface is taken equal to the stiffness of the matrix material. The strength of the interface is taken from experimental results. This leads to

$$u_{ft} = \frac{\sigma_{t,\max}}{E/t} \qquad u_{fs} = \frac{\tau_{\max}}{G/t}$$

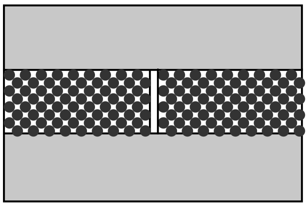
Here t denotes the thickness of the resin rich layer, E, G and maximum stresses relate to the matrix material. This relative displacement is introduced in damage expressions also involving the fracture energies in mode I (tension) and mode II (shear), as discussed in [4].



a) delamination



b) fibre failure



c) matrix failure

Figure 6. Failure modes of the skin model

The failure criterion for fibre failure is given by

$$f_f = \sqrt{\frac{\varepsilon_{11}^t}{\varepsilon_{11}^c}(\varepsilon_{11})^2 + \left(\varepsilon_{11}^t - \frac{(\varepsilon_{11}^t)^2}{\varepsilon_{11}^c}\right)\varepsilon_{11}}$$

Here ε_{11}^t and ε_{11}^c are the failure strains in fibre direction in tension and compression, respectively. Failure occurs when f_f exceeds its threshold value ε_{11}. This is also introduced into a damage expression [2].

Finally for matrix failure the following failure criterion is used

$$f_m = \sqrt{\frac{\varepsilon_{22}^t}{\varepsilon_{22}^c}(\varepsilon_{22})^2 + \left(\varepsilon_{22}^t - \frac{(\varepsilon_{22}^t)^2}{\varepsilon_{22}^c}\right)\varepsilon_{22} + \left(\frac{\varepsilon_{22}^t}{\varepsilon_{12}^s}\right)^2 (\varepsilon_{12})^2}$$

where ε_{22}^t and ε_{22}^c are the failure strains perpendicular to the fibre direction in tension and compression, respectively. The failure strain for shear is ε_{12}^s. Failure occurs when the quantity f_m exceeds its threshold value ε_{22}^t, and which is employed in the reduction by damage of the finite element material modulus matrix of the prepreg, consisting of contributions from both the fibres and the matrix, as discussed in [4].

6. Validation of Model

The model described above was validated using several tests carried out within the same research program. They comprised aluminium rivets as well as lock bolts. The first example shown here will be a lap joint test with aluminium rivets joining two glare sheets.

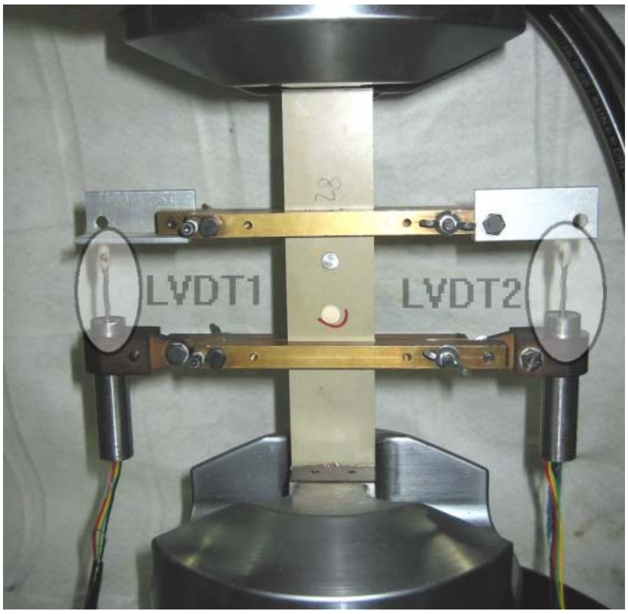
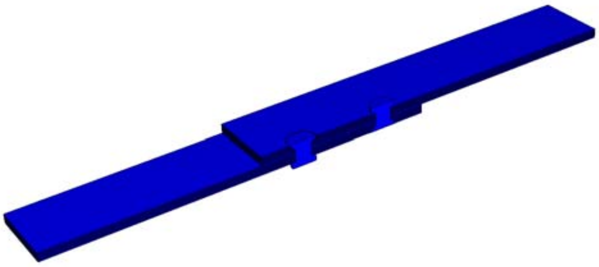


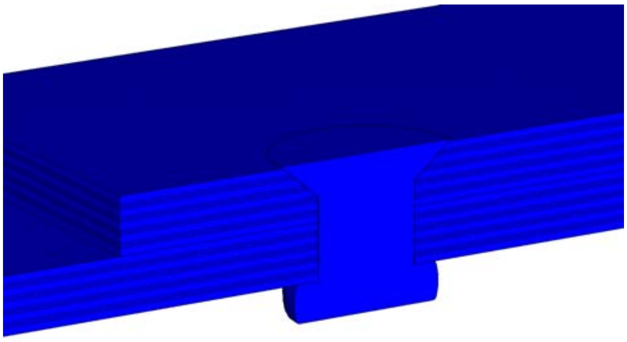
Figure 7. Test setup of two rivet row lap joint

Figure 7 shows the test setup of one such lap joint test. The corresponding numerical model employing the models for fastener and skin presented above is seen in figure 8.

A comparison of the force-deformation behaviour is shown in figure 9. The loading comprised initial loading, unloading and reloading in the same direction repeatedly.



a) model of riveted lap joint



b) Radial stresses after contact fit

Figure 8. Two rivet row lap joint

The tests of lap joints with aluminium rivets resulted in either rivet shear or tension failure alternatively rivet pull through, depending on relative skin and rivet dimensions. These failure modes could be reproduced by simulation.

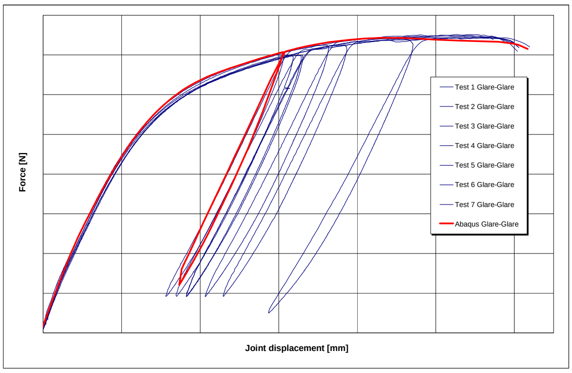


Figure 9. Load-displacement diagram of riveted lap joint test simulation

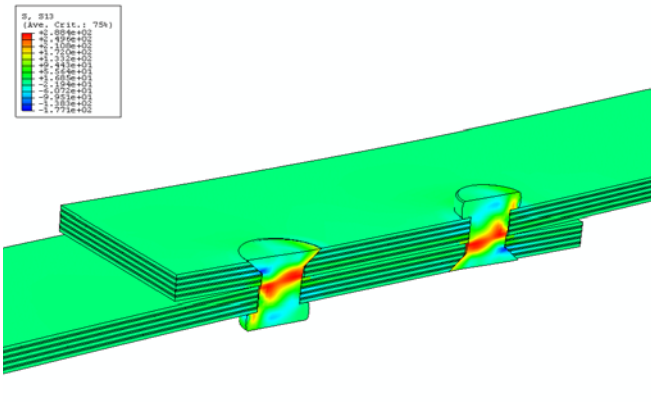


Figure 10. Stress contour plots of simulation of lap joint test

In figure 10 stress contour plots are shown for the lap joint simulation.

A second example comprises a similar lap joint with lock bolts. Figure 11 shows comparison force and deformation between test and simulation.

Figure 12 shows deformed shape of simulation compared with test. The general deformation pattern could be reproduced well. This also holds true for the damage types occurring during the test, including bearing damage comprising plasticity of the aluminium and delamination in the vicinity of the fastener.

The pull through failure mode occurring in the tests with the lock bolts could be reproduced in the simulation [3].

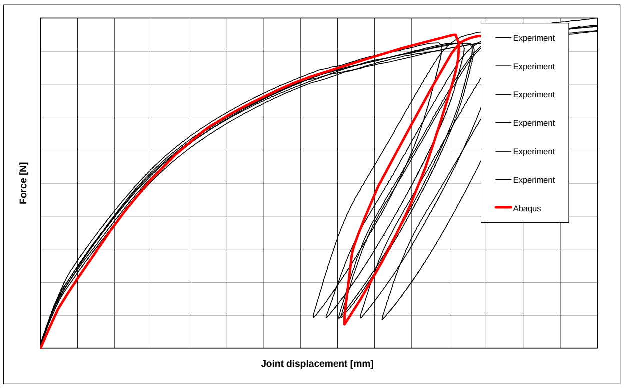


Figure 11. Load-displacement diagram of lock bolt lap joint test simulation

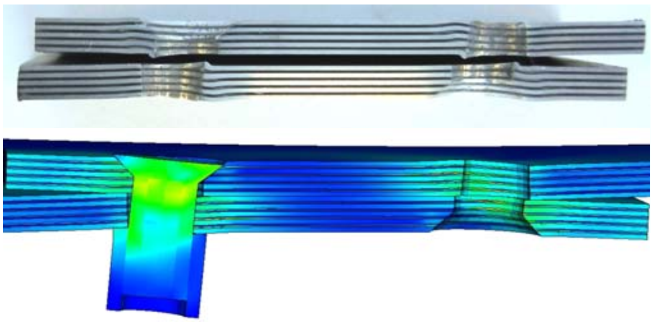


Figure 12. Deformed shape of lock bolt lap joint test; comparison between test and simulation

7. Summary, Conclusions and Recommendations

An overview of different models for the simulations of the static strength behaviour of fasteners has been given. The development of a detailed fastener model, based on solid finite elements, was introduced. The model of surrounding materials, in particular fibre metal laminates was also introduced.

The modelling of damage within the fastener as well as in the surrounding materials was described.

Focus was placed on aluminium rivet. The squeezing sequence for such a rivet was simulated numerically and two methods for this were presented.

Numerical simulation of lap joints using the above model was then presented introducing a physical test carried out in the same research program.

The numerical results allowed for the following conclusions:

- the most important physical aspects of a rivet connection can be readily accounted for in the model
- in particular the squeezing of an aluminium rivet can be realistically carried out

by the model using two methods both giving realistic results

- the static tests could be simulated realistically including the behaviour displayed at unloading and reloading
- the described models were capable of reproducing the different type of fastener failure occurring during tests
- the test conditions applied here are more severe than real aircraft conditions

Recommendations for further research can be made as follows. The detailed models here should be supplemented with computationally more efficient models, incorporating the most important characteristics of fastener behaviour, to be applied in engineering computations. The detailed models presented here should be further developed and be adapted to further laminated materials.

Acknowledgements

The studies presented here were part of the R&T project FIMELAS sponsored by Airbus 2003 through 2005 under a grant in the Advanced Numerical Methods cluster. The authors are grateful to Koen Vervenne and Attila Nagy of ALE for proofreading the manuscript.

8. References

[1] Vlot, A., Gunnink, J. W., “Fibre Metal Laminates - An introduction”, Kluwer Academic Publishers, 2001.

[2] P. Linde, FIMELAS, outline proposal for Research & Technology project, internal report, Airbus Deutschland GmbH, 2002.

[3] Linde, P., FIMELAS – WP 3 Modelling of Joints and Structural Details, Technical Report RP0412481, Airbus, 2005.

[4] Linde, P., FIMELAS – WP 1 FML Skin Material Models: Numerical Models, Technical Report RP0412473, Airbus, 2005.

[5] Writing User Subroutines with ABAQUS, Hibbitt, Karlsson and Sorensen Inc., Pawtucket, RI, 2001.

[6] Schipperen, J. H. A., “Computational Modelling of Failure in Fibre Rinforced Plastic”, Dissertation, Delft University of Technology, Delft, 2001.

[7] Linde, P., de Boer, H., Modelling of Inter Rivet Buckling of Hybrid Composites, Comp. Struct. 73, pp. 221-228, 2006.