Paper 103

©Civil-Comp Press, 2012
Proceedings of the Eighth International Conference
on Engineering Computational Technology,
B.H.V. Topping, (Editor),
Civil-Comp Press, Stirlingshire, Scotland

Modelling of Multi-Material Assemblies using an Equivalent Finite Element

O. Omram[1], V.D. Nguyen[1], H. Jaffal[2], P. Marchand[2] and P. Coorevits[1]
[1] Université de Picardie Jules Verne
Eco-PRocédés, Optimisation et Aide à la Décision, (EPROAD EA-4669)
IUT de l'Aisne, Saint-Quentin, France
[2] CETIM, Pôle ICS, Senlis, France

Abstract

The aim of the present work is to develop a methodology which allow us to simplify an assembly point, and keeping a realist physical behaviour, in addition to reduce the time of calculation of numerical simulations. Indeed, the study of the structure by a three-dimensional finite element, taking into account the problem of non-linearities (behaviour and contact) increase the time of calculation and require large size of memory. Likewise, the optimization of the assembly (number and position of the points) leads to an important number of calculations. Therefore it is absolutely necessary to reduce the time of each numerical simulation.

The classical method of simplification consisted to replace the assembly point by a simple rigid connector, in many cases this method don't present the behaviour of point and can cause significant errors in the global response, or local behaviour.
Several studies exist in the literature but they do not apply to multi-material assemblies or they require specific elements whose it is not feasible to apply it in an industrial code.

In this paper, we propose a simple and nonlinear finite element model that responds to industrial requirement. This model is an equivalent element (connector) that creates a connection between two nodes; this type of connector is available in many industrial codes. So the difficulty is to determine its mechanical behaviour, taking into account the geometrical and materials parameters, which is based on experimental tests. We will show the feasibility of this approach.

Another important problem need to study is to modeling the location of plastic strain and damage in the contact area, where this location leads to embrittlement of structures and pilots their mechanical ruin. So we will study the local stress in the critical points of assembly by a post-treatment of finite element solution.

Keywords: finite element, assembly, homogenization, multi-materials, equivalent element, optimization.

1 Introduction

The bolting and the riveting are parts of technique of assembly, which their applications are used in several fields industrial: aviation, railway, automotive … in order to optimize the design of structure. The global model of great structures (eg: airplane wing, train set…) may be very complex and very expensive to design because of the important number of degrees of freedom and the non-linearity of behaviour of materials and the contact. To control the size of global model and the simulation time, a simplified model has been used.

The experimental works of Langrand [1] and Dang-Hoang [2] have studied the influence of edge effects, clearances and number of fixings on the global mechanical behaviour of the assembly of a mono-material. In [1], the author has showed that a simple model of friction as Coulomb's model is not well adapted for modeling this kind of assembly. Furthermore, we must take into account the influence of deformation, load and the state of the contact area on the friction to improve the results. In numerical standpoint, the study [2] showed that we can describe the behaviour of bolted assembly but by introducing the preload to obtain the tightening torque. In the study [3], Berot has considered a simplification of a rivet by a third cylindrical body placed between two plates; this approach shows disadvantages because it does not represent the actual geometry of the assembly system (introduction of the distance between the plates, no consideration of the rivet head,...). In a second study, the rivet is replaced by a virtual model, that is to say an area joint to the two plates with a specific behaviour. The results are quite acceptable for the overall behaviour, but this method cannot describe the local behaviour of the assembly point; the author also noted that it is difficult to automate this method.

Another simple method consists of modeling the plates by finite element and the assembly point by an equivalent element. In the literature; these studies [1, 3-4] have shown that we can use kinematic constraints (relationship between the displacements of points or surfaces), non-linear elements (beam, spring or mixed elements) and hybrid formulations. Langrand showed that the use of kinematic constraints is not appropriate to describe the nonlinear behaviour of the assembly [1]. Moreover, this approach depends on the compliance of meshing. The hybrid approach formulations are quite expensive in computation time.

We propose here an approach based on nonlinear elements.

2 Assembly model by nonlinear elements

2.1 Problem

In order to be able to produce the behaviour of an assembly, it is necessary to define a reference which identifies this equivalent mechanical behaviour. Once all the requests have been successfully simulated, an equivalent element whose behaviour reproduces the best specifications of the complete model can be built. For this model, the equivalent behaviour will be identified by minimizing the difference between the curves charge/displacement (for traction and compression) or the curves moment/angle (for bending) of the equivalent model and those of the reference model, the Figure 1 shows the type of action used to identify the mechanical behaviour of a structure.

In our study, the idea is to simplify the three-dimensional model (3D) by a beam of one-dimensional model (1D); our approach consists of replacing the bolt by a nonlinear element (nonlinear spring). So we need to determine the parameters of this element: modulus and elastic limit, plasticity constants. They are identified from experimental results of Dang Hoang [2]. We will begin by studying the simplest case of traction.

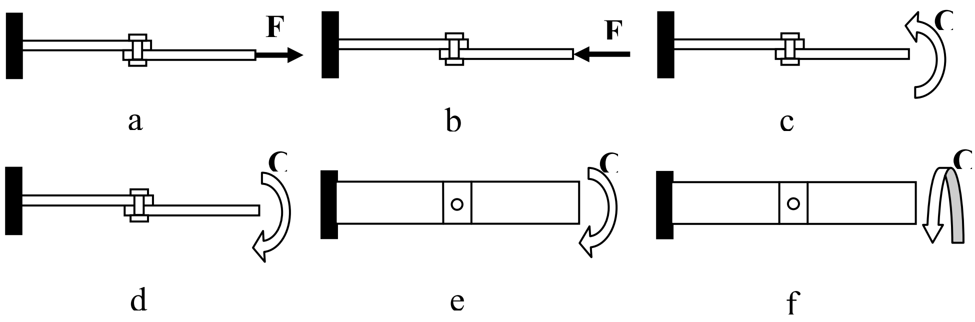

Figure 1: The six basic actions (a-traction; b-compression; c- peeling; d- anti peeling; e- torsion of bolt; f- torsion)

2.2 Experimental Results

Dang-Hoang has studied the bolted assembly of two beams (Figure 2); the beams are constituted of the same material (Aluminum 6082 T6). The mechanical properties of this material are: Young's modulus E=69GPa and elastic limit σ_e=266MPa; the geometric characteristics of the beams are: length l_o=195mm and section S=128mm^2.

Stainless steel bolts are used in the assembly, type ISO4762–M8x20–8x8. A tightening torque C=40Nm is applied to the bolt. The assembly is subjected to a uni-axial displacement until the rupture.

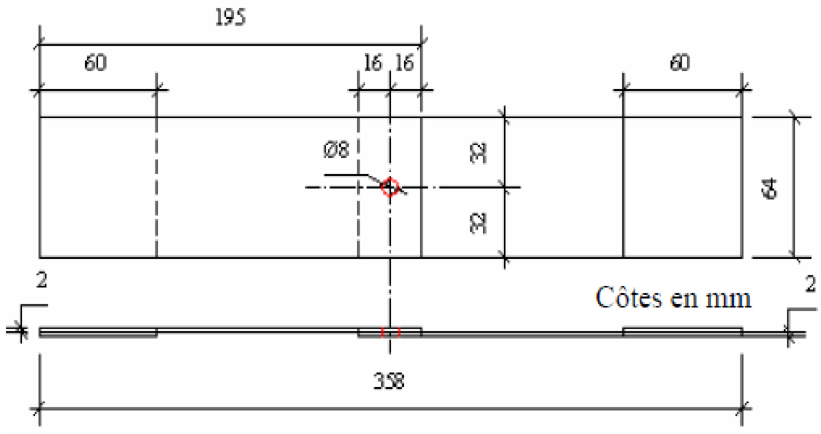

Figure 2: Dimensions of the tested assembly in millimeter

When the traction force reaches a certain value, the slip appears; the sliding of the plates begins with an adjusting clearance between the bore and the body bolt. When the plate 1 touches the bolt's body, the sliding continues due to the clearance between the plate 2 and the body.

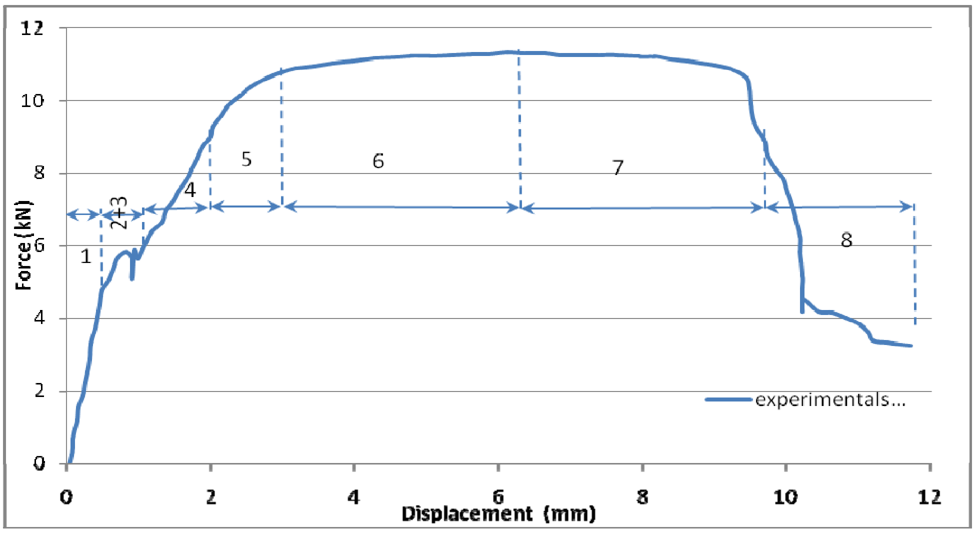

Figure 3: Traction force as a function of the displacement

The behaviour of the structure is identified experimentally at Figure 3; we note the existence of 7 phases:

- Phase 1: elastic behaviour,
- Phase 2: relative displacement due to sliding of the beams corresponding to the clearance bore/bolt,

- Phase 3: matting and adaptation occurs at level of contact of bolt's body and bores of the plates because of the difference between the dimension of the bolt and of the borings of plates,
- Phase 4: elastic behaviour of the structure (plates + bolt) which is subjected to a tensile with a secondary inflection,
- Phase 5: plasticization around the bore of the plates,
- Phase 6: stable propagation of the crack,
- Phase 7: unstable propagation of the crack,
- Phase 8: brutal rupture of the assembly.

2.3 One-dimension model

Our model consists of two beams connected by a nonlinear spring (Figure 4) subjected to a tensile force. To identify the behaviour law of this model, we find that the material of the beams remains in the elastic field because of the strain does not exceed the elasticity limit; therefore the inelastic deformation occurred is due to the non-linear spring.

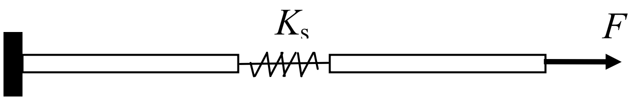

Figure 4. Model 1D

The spring extension is calculated from the overall displacement:

$$\Delta l_s = \Delta l - 2\Delta l_b = \Delta l - 2F/K_b \quad (1)$$

That Δl_b is the beam displacement, K_b elastic rigidity ($K_b=ES/l_o$). So we can deduce the experimental curve of the behaviour of the spring (Figure 5).

We will describe the behaviour of the spring in 4 phases:

Phase 1: elastic

$$\text{If } \Delta l_s \leq \Delta l_e,\ F = K_{s,e}\Delta l_s \quad (2)$$

From the experimental curve, we identify the values of Δl_e=0.373mm and $K_{s,e}$=15.528 kNmm^{-1}.

Phase 2: slipping

The force remains constant, the maximum value of the displacement of slipping is Δl_g = 0.773mm and F_e=5.6695kN:

$$\text{If } \Delta l_e \leq \Delta l_s \leq \Delta l_g,\ F = F_e \tag{3}$$

Phase 3: elastic

$$\text{If } \Delta l_g \leq \Delta l_s \leq \Delta l_{eg},\ F - F_e = K_{s,g} \Delta l_s \tag{4}$$

From the experimental curve, we can identify the values of Δl_{eg}=1.818mm and $K_{s,g}$=3.938 kNmm^{-1}.

Phase 4: plastic

This non-linear phase is presented by a simple model of plasticity:

$$\text{If } \Delta l_{eg} \leq \Delta l_s,\ F - F_{eg} = K_{s,p}(\Delta l_s - \Delta l_{eg})^n \tag{5}$$

By the identification from the experimental curve, we obtain $K_{s,p}$=7139.251 kN/mm^n and n=0.298.

In Figure 5, we observe that the behaviour of model matches the experimental curves.

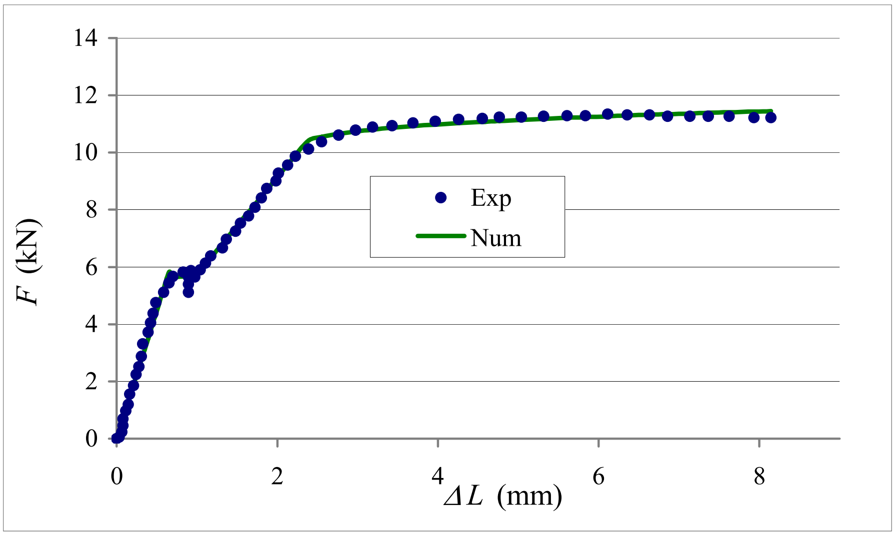

Figure 5. Comparison between the experimental and spring model results

2.4 Calculation by finite elements method in 1D

By using finite element code Cast3M, we have implanted a simple connector that produce the model described previously. Cast3M is a classic code of finite element in displacement; it is based on the principle of minimizing the potential energy. In the elastic field, the potential energy W is expressed by:

$$W = \frac{1}{2}\left[U_b^1\right]^T \times \left[K_b^1\right] \times \left[U_b^1\right] + \frac{1}{2}\left[U_b^2\right]^T \times \left[K_b^2\right] \times \left[U_b^2\right] - F.u_N^2 \tag{6}$$

where $[K_b^1]$, $[K_b^2]$, $[U_b^1]$, $[U_b^2]$ are respectively the rigidity and the displacement matrices of beams 1 and 2, u_N^2 is the displacement of the node where the tensile force F is applied.

We have to add the energy of deformation of the spring. In elastic domain, this energy W_s expresses classically by:

$$W_s = \frac{1}{2} K_{s,e} (u_1^2 - u_N^1)^2 \tag{7}$$

By derivation of the potential energy, we obtain the following system:

$$\begin{bmatrix} k_e & -k_e & & & & & & & \\ -k_e & 2k_e & \ddots & & & & & & \\ & \ddots & \ddots & -k_e & & & & & \\ & & -k_e & K_s + k_e & -K_s & & & & \\ & & & -K_s & K_s + k_e & -k_e & & & \\ & & & & -k_e & 2k_e & \ddots & & \\ & & & & & \ddots & \ddots & -k_e & \\ & & & & & & -k_e & k_e & \end{bmatrix} \times \begin{bmatrix} U_1^1 \\ \vdots \\ U_{n-1}^1 \\ U_n^1 \\ U_1^2 \\ U_2^2 \\ \vdots \\ U_n^2 \end{bmatrix} = \begin{bmatrix} 0 \\ \\ \vdots \\ \\ 0 \\ F \end{bmatrix} \tag{8}$$

Note that each beam is divided into n elements of the same length; its rigidity is expressed by:

$$K_{element} = \begin{bmatrix} k_e & -k_e \\ -k_e & k_e \end{bmatrix} \tag{9}$$

In the non-linear domain, the beams remains elastic, therefore, their matrices of rigidity do not change. For the spring, by using incremental algorithm, the elastic rigidity K_s is replaced by tangential rigidity $K_{s,t}$ as:

$$K_{s,t} = n\left(\Delta l_s - \Delta l_g\right)^{n-1} \tag{10}$$

The implementation of this model in Cast3M gives the following comparison (Figure 6). We find that our numerical curve is identical with the experimental curve of Dang Hoang [2], so the proposed model is justified.

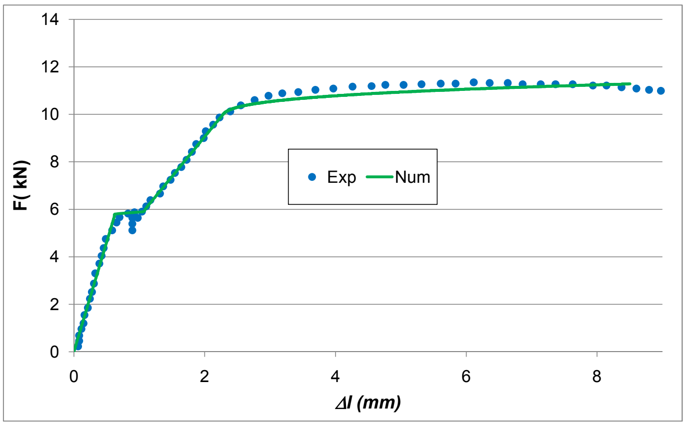

Figure 6. Comparison between the experimental and numerical results of the assembly

3 Conclusion

Initially, we simplified the three-dimensional model of assembly by one-dimensional model composed of two beams connected by spring. The characteristics of spring were identified from experimental results of Dong Hoang for a simple case of tension. The implementation and the validation of the connector model were verified by comparing our results with the experimental results.

In a second step, we will complete this model for different types of actions such as compression, torsion, peeling, etc. (Figure 1). Then we investigate to validate our model with an experimental study in collaboration with CETIM and ArcelorMittal, ours industrial partners.

References

[1] B. Langrand, “Contribution à la caractérisation numérique et expérimentale d’assemblages structuraux rivetés sous sollicitation dynamique ”, Université de Valenciennes, Thesis in French, 1998.

[2] T. Dang-Hoang, “ Rupture et endommagement d’un assemblage boulonné : approche expérimentale et simulation numérique”, Université de Lille 1, Thesis in French, 2009.

[3] M. Berot, “Modélisation simplifiée d'assemblages par éléments équivalents”, Mines ParisTech, Thesis in French, 2009.

[4] E. Deletombe "IMT crashworthiness for commercial aircraft- Modélisation par éléments finis des assemblages, rotules plastiques et rotules- Rapport final", ONERA-Lille, 96/25, 1996.

[5] G. Winkelmuller "IMT crashworthiness for commercial aircraft- 24 month progress report", Mecalog project, E422/92, 1995.