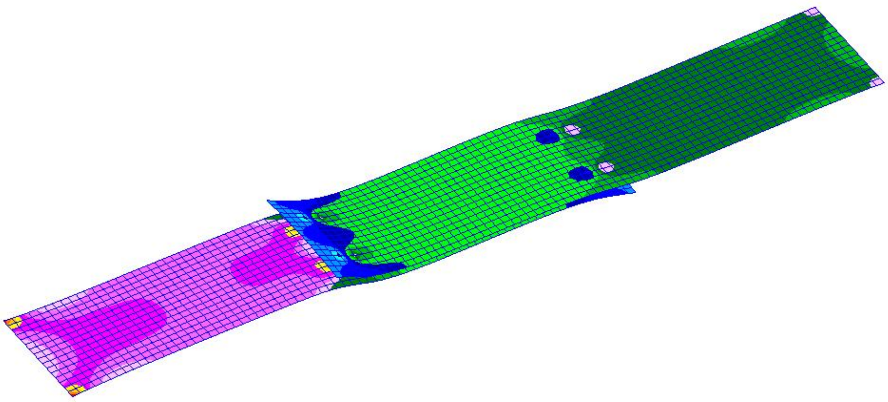
Nationaal Lucht- en Ruimtevaartlaboratorium
National Aerospace Laboratory NLR

Executive summary

Semi-analytical methodology to determine model parameters for simple finite element bolt model



Problem area
Experimentally determined bolt stiffnesses are available from various sources. A finite element model of a bolted joint should provide the same bolt stiffness; otherwise the distribution of bolt loads will be erroneous especially in a multiple load path situation. The bolt stiffness can be changed by adaptation of the beam area or the moments of inertia. Although it is necessary to account for the deformation of the adherent mesh in order to obtain the right bolt stiffness, this is not done in the current methodologies.

Description of work
The novel semi-analytical methodology presented in this paper to determine the model parameters for a simple finite element bolt model provides for a better correspondence between computed and experimentally determined behaviour of a bolted joint, as it explicitly accounts for the local deformation in the finite element representation of the adherents. A mathematical foundation is provided for the simple finite element bolt model that also enables a sensitivity analysis.
The influence of variation in mesh density and boundary conditions for the shell representation of the adherents is established through a simple finite element analysis. The influence of variations in material stiffness and adherent thickness for both metal and composite adherents

Report no.
NLR-TP-2006-136

Author(s)
J.C.F.N. van Rijn,

Classification report
Unclassified

Date
May 2006

Knowledge area(s)
Computational Mechanics & Simulation Technology
Aerospace Collaborative Engineering & Design

Descriptor(s)
FINITE ELEMENT METHOD
BOLTED JOINTS
PARAMETER IDENTIFICATION
MATHEMATICAL MODEL
MESH DEPENDENT BEHAVIOR

This report is based on a presentation to be held at ECCM 2006, Lisbon, 5-9 June 2006

is obtained with the presented methodology.

Results and conclusions
Relations between the displacement and the load and the rotation and the moment were established for the central node of a finite element model of a plate.
A finer mesh gives a larger displacement and rotation of the central node, all other parameters being equal.
An analytical methodology for the single bolt, single lap joint was developed. The model was developed for adherent models with and without an offset.
It is recommended to properly account for the mesh flexibility when beams are used to model the bolts in a joint.

Applicability
The method is relatively easy to use. To take into account the influence of mesh flexibility is a novel feature. In all instances where beams are used to model bolts the methodology should be applied, particularly in a situation where multiple load paths are present.
The use of the methodology was demonstrated to determine the bolt parameters for a total of 40 different configurations within a detailed model of a flap section of an airplane.

Nationaal Lucht- en Ruimtevaartlaboratorium, National Aerospace Laboratory NLR

Anthony Fokkerweg 2, 1059 CM Amsterdam,
P.O. Box 90502, 1006 BM Amsterdam, The Netherlands
Telephone +31 20 511 31 13, Fax +31 20 511 32 10, Web site: www.nlr.nl

Nationaal Lucht- en Ruimtevaartlaboratorium
National Aerospace Laboratory NLR

NLR-TP-2006-136

Semi-analytical methodology to determine model parameters for simple finite element bolt model

J.C.F.N. van Rijn

This report is based on a presentation held at ECCM 2006, Lisbon, 5-9 June 2006

Customer	National Aerospace Laboratory NLR
Contract number	---
Owner	National Aerospace Laboratory NLR
Division	Aerospace Vehicles
Distribution	Unlimited
Classification of title	Unclassified

Approved by:

Author	Reviewer	Managing department
18/5	18/5	18/05 2006.

Contents

10 Figures

(14 pages in total)

III European Conference on Computational Mechanics
Solids, Structures and Coupled Problems in Engineering
C.A. Mota Soares et.al. (eds.)
Lisbon, Portugal, 5–8 June 2006

SEMI-ANALYTICAL METHODOLOGY TO DETERMINE MODEL PARAMETERS FOR A SIMPLE FINITE ELEMENT BOLT MODEL

Joost C.F.N. van Rijn

National Aerospace Laboratory NLR
Collaborative Engineering Systems Department, Aerospace Vehicles Division
P.O. Box 153, 8300 AD Emmeloord, The Netherlands
rijn@nlr.nl

Keywords: Finite Element Analysis, bolted joint, properties, beam element, adherent flexibility.

Abstract. *Bolted joints are commonly used in aircraft structures to join the various parts. These joints are an important factor in the sizing of the construction as a larger thickness is normally required in the joint area. Consequently joints have an important bearing on the weight of the structure and the material costs.*

Aircraft structures are mostly modeled using a shell representation. Bolted joints are incorporated in these models using a simple model that represents the bolt by a beam element that connects to one node in each adherent finite element model. At these particular nodes a point force is exerted on the adherent models that will cause a certain, mesh dependent, deformation.

Experimentally determined bolt stiffnesses are available from various sources. A finite element model of a bolted joint should provide the same bolt stiffness; otherwise the distribution of bolt loads will be erroneous especially in a multiple load path situation. The bolt stiffness can be changed by adaptation of the beam area or the moments of inertia. Although it is necessary to account for the deformation of the adherent mesh in order to obtain the right bolt stiffness, this is not done in the current methodologies. The novel semi-analytical methodology presented in this paper to determine the model parameters for a simple finite element bolt model provides for a better correspondence between computed and experimentally determined behavior of a bolted joint, as it explicitly accounts for the local deformation in the finite element representation of the adherents. A mathematical foundation is provided for the simple finite element bolt model that also enables a sensitivity analysis. The influence of variation in mesh density and boundary conditions for the shell representation of the adherents is established through a simple finite element analysis. The influence of variations in material stiffness and adherent thickness for both metal and composite adherents is obtained with the presented methodology. The use of the methodology is demonstrated to determine the bolt parameters for a total of 40 different configurations within a detailed model of a flap section of an airplane.

1 Introduction

Bolted joints are commonly used in aircraft structures to join the various parts. These joints are an important factor in the sizing of the construction, for composite to composite as well as for composite to metal joints, as a larger thickness is normally required in the joint area. Consequently joints have an important bearing on the weight of the structure and the material costs. Moreover the production costs of joining are significant.

Aircraft structures are often modeled using a shell representation. Bolted joints are incorporated in these models using a simple model that represents the bolt by a beam element that connects to one node in each adherent finite element model. At these particular nodes a point force is exerted on the adherent models that will cause a certain, mesh dependent, deformation.

Experimentally determined bolt stiffnesses are available from various sources. A finite element model of a bolted joint should of course provide the same bolt stiffness; otherwise the distribution of bolt loads will be erroneous especially in a multiple load path situation. The bolt stiffness can be changed by adaptation of the beam area or the moments of inertia. It is necessary to account for the deformation of the adherent mesh in order to obtain the right bolt stiffness.

The novel semi-analytical methodology, presented in this paper, to determine the model parameters for a simple finite element bolt model provides for a good correspondence between computed and experimentally determined behavior of a bolted joint. It enhances the finite element representation of a bolt as it explicitly accounts for the local deformation in the finite element representation of the adherents.

Variation in mesh density, boundary conditions, material stiffness and adherent thickness are parameters in the presented methodology.

The use of the methodology is demonstrated by determining the bolt parameters for a total of 40 different configurations within a detailed model of a flap section of an aircraft

2 Single bolt model

A simple single bolt, single lap model is used to assess the influence of adherent stiffness parameters, mesh density and boundary conditions on the overall model stiffness.

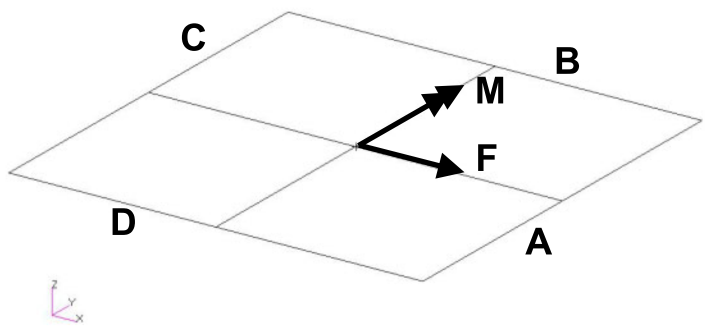


Figure 1. Adherent finite element model.

Adherent finite element model

The model including boundary conditions and loads is shown in figure 1.

The adherent dimensions are 20 by 20 mm2. The adherent is modeled with 4 4-node shell elements.

The adherent material is aluminum. The Young's modulus is 72000 MPa and Poisson's contraction ratio is 0.33. The thickness of the sheet is varied. The elements are positioned at the neutral plane of the sheet (no offset) or at the sheet surface (offset equal to half the thickness).

The boundary conditions are clamped or simply supported for all edges.

The load is a unit load in the x-direction and a unit moment around the y-axis acting on the central node.

The displacement in the x-direction and the rotation around the y-axis of the central node are the results of these analyses. The results for the model without the offset are as follows.

Influence of adherent thickness

The displacement in the x-direction is inversely proportional to the thickness:

$$u_x = \frac{a_x}{t} \cdot F$$

where the parameter a_x accounts for the mesh dependency which includes the effect of the materials Young's modulus and Poisson's contraction ratio.

The same equation and the same constant ($7.58\ 10^{-6}$ mm^2/N) were found for all boundary conditions.

The rotation around the y-axis can be written as a function of the thickness t and the moment M:

$$r_y = \frac{b_y(t)}{t^3} \cdot M$$

where the parameter b_y accounts for the mesh dependency which includes the effect of the materials Young's modulus and the Poisson's contraction ratio.

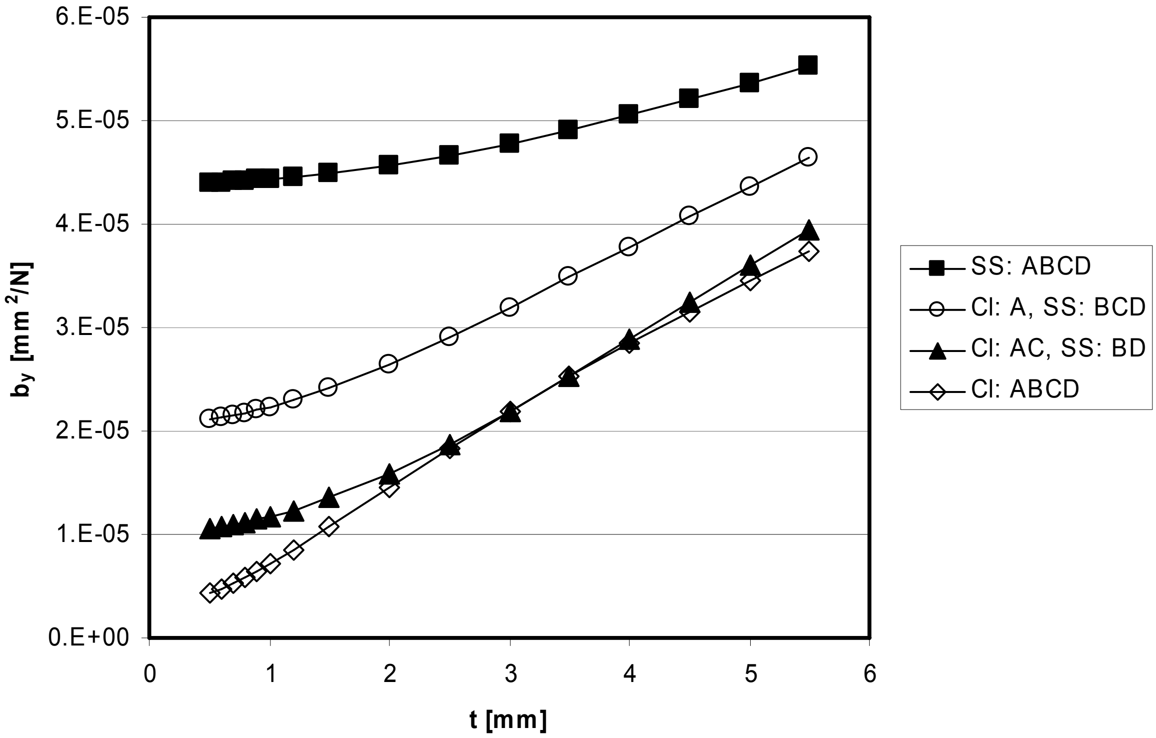


Figure 2. Parameter b_y as function of adherent thickness t for various boundary conditions for coarse mesh.

The parameter b_y is a function of the thickness and also varies with the applied boundary conditions, as depicted in figure 2. For thin plates the value b_y tends to a constant value except for the all clamped boundary condition. For thick plates the value b_y is almost proportional with the thickness, which means that the rotation is inversely proportional to the thickness squared.

Influence of mesh density

The adherent was modeled with various element sizes. The element size varied from 10 by 10 mm^2 (with 4 elements to model the adherent) to 1 by 1 mm^2 (with 400 elements to model the adherent).

The parameter b_y is a function of the thickness for various applied boundary conditions obtained with a mesh size of 1 by 1 mm^2, is shown in figure 3. The value of b_y for the fine mesh is (significantly) larger than for the coarse mesh. For the fine mesh the difference between the various boundary conditions is very small.

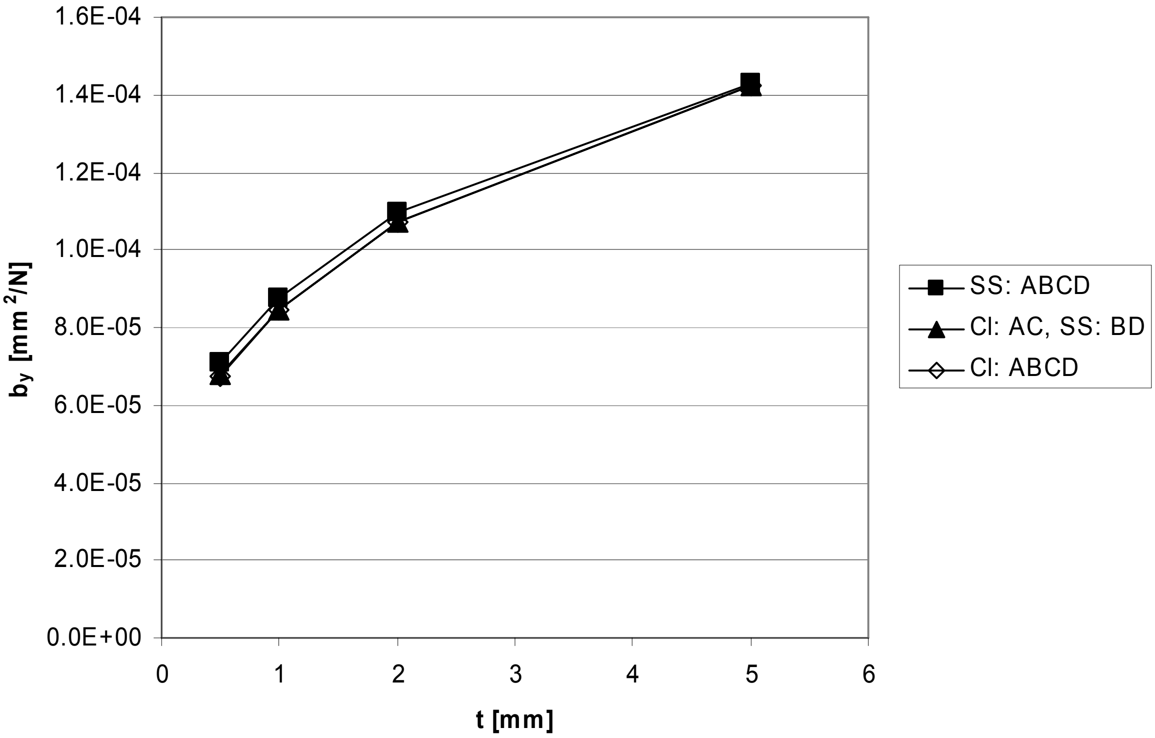


Figure 3. Parameter b_y as function of adherent thickness t for various boundary conditions for fine mesh.

The influence of element size on the parameters a_x and b_y is shown in figures 4, 5 and 6.

As can be seen, smaller elements give progressively increasing displacements and rotations for a given load. This behaviour is atypical for finite element analyses, where mesh refinement normally results in converging displacements. This finding can be attributed to the behavior of a plate loaded by a point load (or moment), see figure 7.

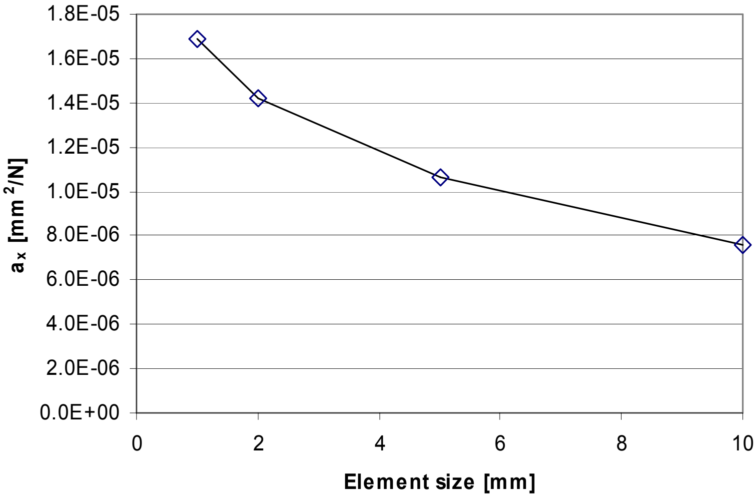


Figure 4. Parameter a_x as function of element size.

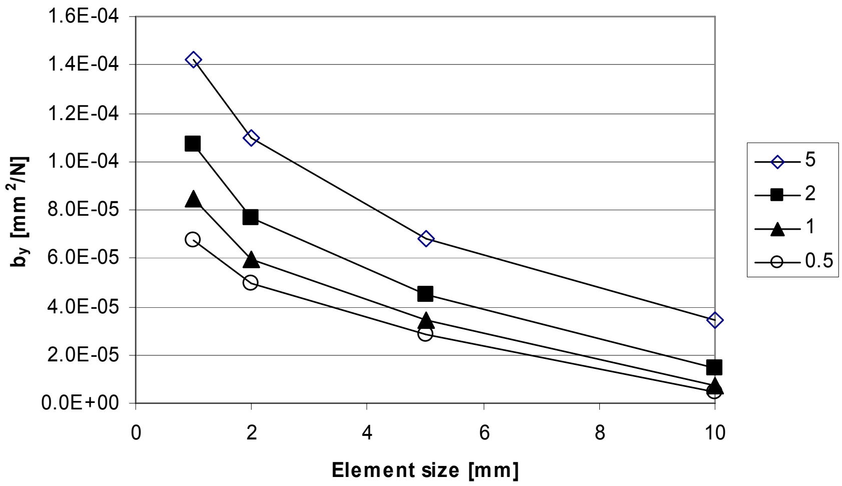


Figure 5. Parameter b_y as function of element size for various adherent thicknesses for all clamped boundary condition.

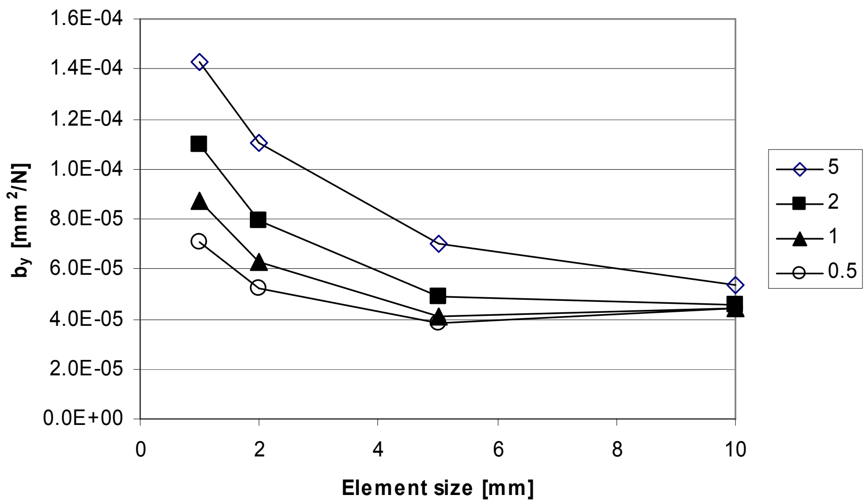


Figure 6. Parameter b_y as function of adherent thickness t for various adherent thicknesses for all simply supported boundary condition.

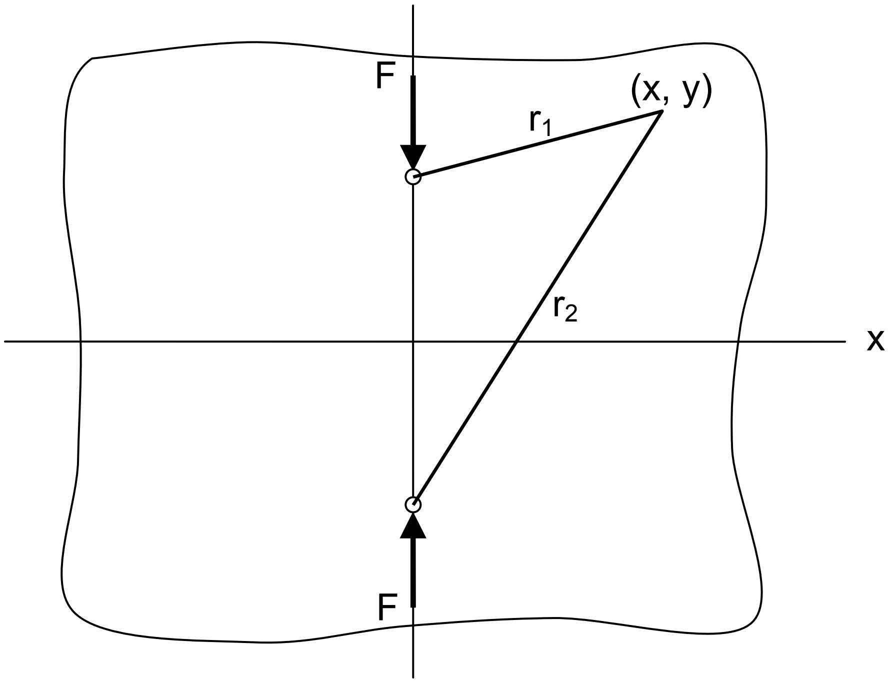


Figure 7. Parameters for a plate loaded by a point load.

The displacement in y-direction of a point (x, y) due to one pair of rivet forces acting in the y-direction is given by Love [Ref. 1]

$$v(x,y) = \frac{F(1+\nu)(3-\nu)}{4\pi Et}\left(\ln\left(\frac{r_1}{r_2}\right) + \frac{(1+\nu)}{(3-\nu)}x^2\left(\frac{1}{r_1^2} - \frac{1}{r_2^2}\right)\right)$$

where E is the Young's modulus and t is the sheet thickness.

This gives a logarithmic singularity of the displacement in the load introduction points ($r_1 = 0$ or $r_2 = 0$). The results obtained in the finite element analyses can be interpreted as a better representation of the logarithmic singularity of the displacement for smaller element sizes.

3 Analytical model for a single bolt, single lap joint

An analytical model for the single bolt, single lap joint was developed to create a better understanding of the influence of the various relevant parameters, and to facilitate the determination of the correct area for the beam that models the bolt.

A number of the model parameters is shown in figure 8.

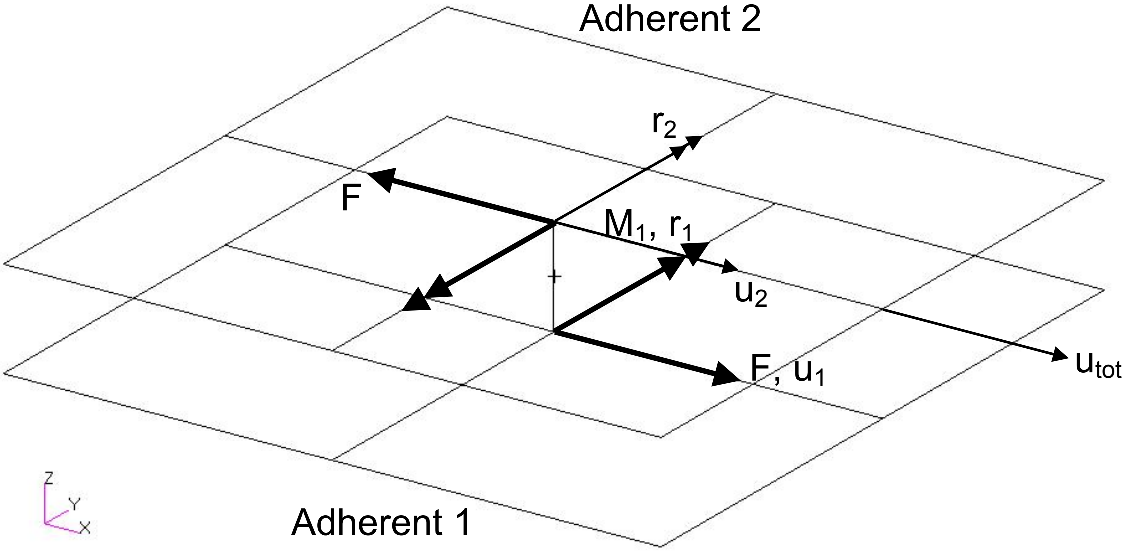


Figure 8. Parameters of analytical model for single bolt, single lap joint.

Analytical model for finite element model without offsets

For the adherents the following equations hold:

$$u_1 = \frac{a_1}{t_1} \cdot F$$

$$r_1 = \frac{b_1}{t_1^3} \cdot M_1$$

$$u_2 = u_{tot} - \frac{a_2}{t_2} \cdot F$$

$$r_2 = -\frac{b_2}{t_2^3} \cdot M_2$$

where t_1 is the thickness of adherent 1 and t_2 is the thickness of adherent 2, u_{tot} is the total displacement of adherent 1 that is forced upon all edge nodes.

For the beam element that models the bolt the following equations hold:

$$u_2 = u_1 + r_1 \cdot l + M_2 \cdot \frac{l^2}{2EI} - F \cdot \frac{l^3}{3EI} + F \cdot l \cdot \frac{2 \cdot (1+\nu)}{EAC_A}$$

$$r_2 = r_1 + M_2 \cdot \frac{l}{EI} + F \cdot \frac{l^2}{2EI}$$

where L is the length, A the area and I the moment of inertia of the beam element, E is the Young's modulus and ν the Poisson's contraction ratio of the bolt material.

The parameter C_A (C_A = 0.9836) is not used in the standard beam theory, but was found to be necessary to render the same results as obtained with the finite element analysis for the coarse mesh.

Moment equilibrium gives the seventh equation:

$$M_1 = M_2 + F \cdot l$$

This system can be solved for the unknown variables. The following relation is found between the total displacement u_{tot} and the load F:

$$u_{tot} = F \cdot \left[\frac{l^3}{12 \cdot E \cdot I} + \frac{2 \cdot l \cdot (1+\nu)}{E \cdot A \cdot C_A} + \frac{a_1}{t_1} + \frac{a_2}{t_2} + \left(B_3^+ - \frac{\left(B_3^-\right)^2}{l + E \cdot I \cdot B_3^+} \cdot E \cdot I \right) \cdot \frac{l^2}{4} \right]$$

where

$$B_3^+ = \frac{b_1}{t_1^3} + \frac{b_2}{t_2^3} \qquad B_3^- = \frac{b_1}{t_1^3} - \frac{b_2}{t_2^3}$$

Analytical model for finite element model with offsets

Sometimes the model uses offsets to position the finite element mesh on the surface of the adherents, for instance when a bolt is combined with adhesive and the adhesive thickness should be modelled properly. The lower adherent will be modeled with an offset in upward direction of $t_1/2$. The upper adherent will have an offset in downward direction of $t_2/2$. The displacements and rotations in the adherents are then as follows:

$$u_1 = \frac{a_1}{t_1} \cdot F + \frac{b_1}{2 \cdot t_1^2} \cdot \left(M_1 + F \cdot \frac{t_1}{2} \right)$$

$$r_1 = \frac{b_1}{t_1^3} \cdot \left(M_1 + F \cdot \frac{t_1}{2} \right)$$

$$u_2 = u_{tot} - \frac{a_2}{t_2} \cdot F + \frac{b_2}{2 \cdot t_2^2} \cdot \left(M_2 - F \cdot \frac{t_2}{2} \right)$$

$$r_2 = -\frac{b_2}{t_2^3} \cdot \left(M_2 - F \cdot \frac{t_2}{2} \right)$$

The equations for the beam element are the same.

Solving the system gives the following relation between the total displacement u_{tot} and the load F:

$$u_{tot} = F \cdot \left[\frac{l^3}{12 \cdot E \cdot I} + \frac{2 \cdot l \cdot (1+\nu)}{E \cdot A \cdot C_A} + \frac{a_1}{t_1} + \frac{a_2}{t_2} + \frac{B_1^+}{4} + \frac{B_2^+ \cdot l}{2} + \frac{B_3^+ \cdot l^2}{4} - \frac{\left(B_2^- + l \cdot B_3^-\right)^2}{l + E \cdot I \cdot B_3^+} \cdot \frac{E \cdot I}{4} \right]$$

where

$$B_1^+ = \frac{b_1}{t_1} + \frac{b_2}{t_2} \quad B_2^+ = \frac{b_1}{t_1^2} + \frac{b_2}{t_2^2} \quad B_2^- = \frac{b_1}{t_1^2} - \frac{b_2}{t_2^2} \quad .$$

4 Determination of beam properties

In the context of research on bond assisted single step assembly a finite element model was generated of a part of a wing flap. The model contains 205 beam elements that modeled a 4 mm bolt. A total of 40 different combinations of skin and doubler thicknesses were present in the model. The methodology presented in the previous section is used to determine the beam area that will give the desired bolt spring constants for a beam length of 0.2 mm, modeled using offsets.

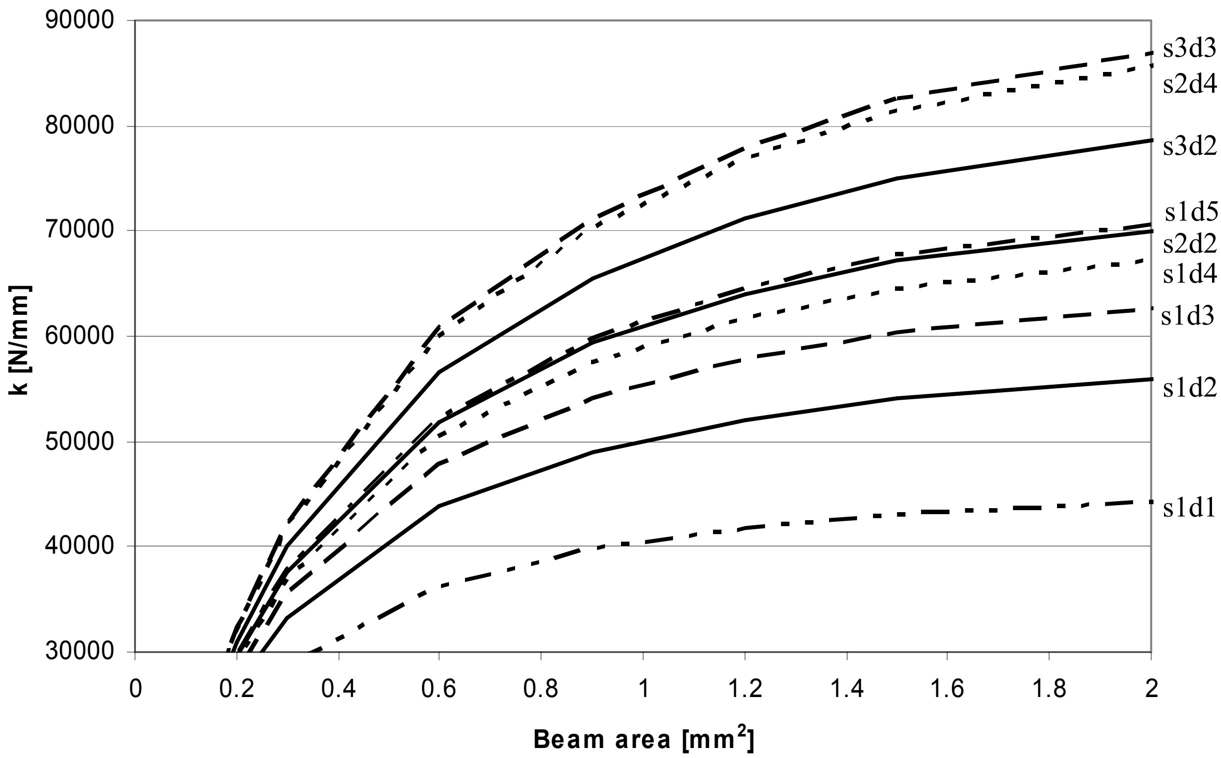


Figure 9. Bolt spring contant k as function of the beam area for various combinations of skin and doubler thickness.

The fastener spring constant as function of beam area for various combinations of the skin and doubler thickness (s1d1 indicates skin thickness 1 mm, doubler thickness 1 mm) is given in figure 9. The skin and doubler thickness can be interchanged.

The following equation (Ref. 2) was used for the fastener spring constants:

$$k = \frac{ED}{C_1 + C_2\left(\dfrac{D}{t_d} + \dfrac{D}{t_s}\right)}$$

where k is the fastener spring constant, E is the Young's modulus of skin and doubler, D is the fastener diameter, t_s is the skin thickness, t_d is the doubler thickness, C_1 is a constant equal to 1.667 for steel fasteners and C_2 is a constant equal to 0.86 for steel fasteners.

For the relevant combinations of skin and doubler thicknesses in the flap section the fastener spring constant was determined, as shown in figure 10. The beam area was determined using the final equation from section 3.1. The value for the parameter b was interpolated between the table values as shown in figure 2. The value of A is presented as a relative value normalized by the area of a 4 mm bolt (12.6 mm^2).

The bolts were modeled with very short beams; therefore the relative area of the beams is also small.

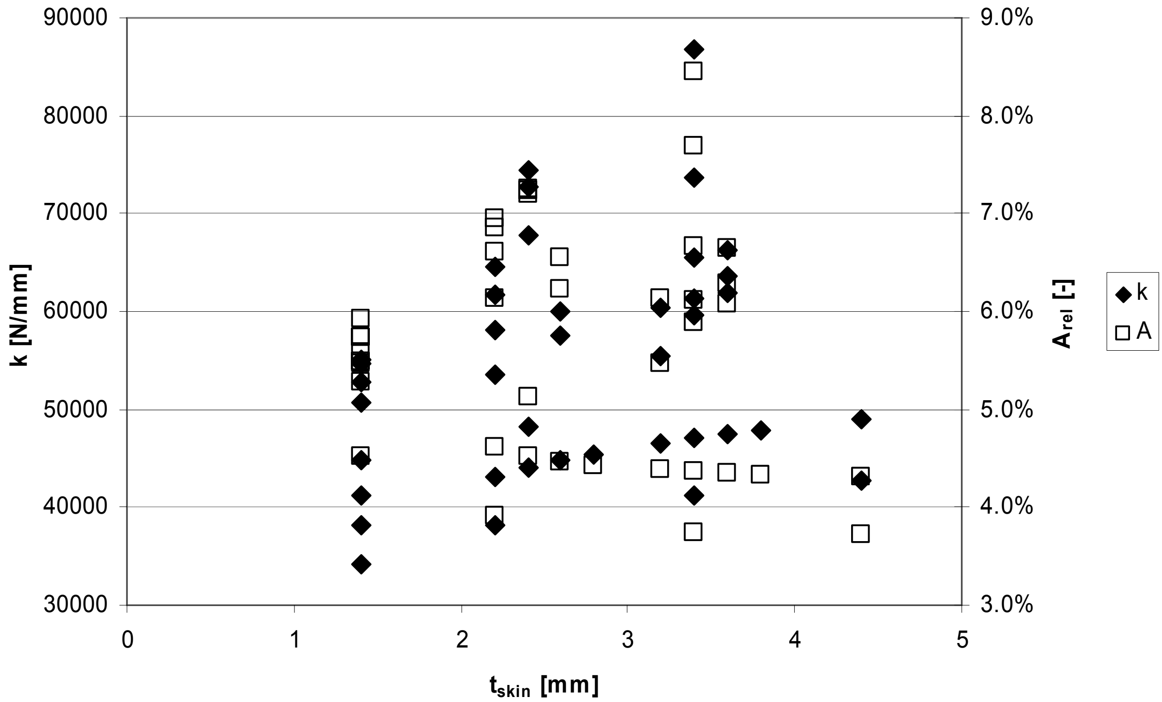


Figure 10. Bolt spring contant k and relative area of beam as function of the skin thickness for various doubler thicknesses.

5 Conclusions and recommendations

Relations between the displacement and the load and the rotation and the moment acting were established for the central node of a finite element model of a plate.

A finer mesh gives a larger displacement and rotation of the central node, all other parameters being equal.

An analytical methodology for the single bolt, single lap joint was developed. The model was developed for adherent models with and without an offset.

The methodology was used to determine beam areas for a series of bolt configurations in a finite element model of a flap.

It is recommended to properly account for the mesh flexibility when beams are used to model the bolts in a joint.

6 Acknowledgement

The presented methodology was developed within the context of the Sixth Framework EU project Bond Assisted Single Step Assembly (BASSA).

References

[1] A.E.H. Love, *A treatise on the mathematical theory of elasticity*. New York, Dover, 4th Ed., 1944.

[2] Swift, T., Fracture Analysis of Stiffened Structure, *Damage Tolerance of Metallic Structure: Analysis Methods and Application*, ASTM STP 842, J.B. Chang and J.L. Rudd, Eds., American Society for Testing and Materials, 1984.