

## **BOLJAT - A Tool for Designing Composite Bolted Joints using Three-Dimensional Finite Element Analysis**

G.S. Padhi, M.A. McCarthy\*, C.T. McCarthy

Composites Research Centre,  
Department of Mechanical and Aeronautical Engineering  
University of Limerick, Limerick, Ireland

### **Abstract**

This paper discusses the development of a software tool for design of composite bolted joints, using three-dimensional finite element analysis. The tool allows the user to create the joint geometry through a menu-driven interface and then generate a customised mesh according to the user's needs. Contact parameters are defined automatically, which shields the user from the most difficult part of the process. Boundary conditions, bolt pre-loads, and material properties can also be set. Only a few manual steps are necessary to complete the finite element code generation process. By automating the time-consuming model creation process, the tool facilitates the increased use of three-dimensional finite element analysis in the design of composite bolted joints. A case study is shown to demonstrate the usefulness of the tool.

---

Key words: E. Joints/joining; C. Finite element analysis; A. Carbon-fibre; A. Layered structures

### **1. Introduction**

Mechanical fastening remains the primary means of joining composite components in modern aircraft structures. Since the stress concentrations that develop around the holes severely reduce the strength of the structure, it is important that the best available tools are used for analysis and design of composite joints. Inefficiently designed joints can have a severely detrimental effect on the weight-saving advantage of composites over metals, while incomplete understanding of stresses and failure in joints could lead to catastrophic failure of the structure. As composites find increasing use on primary, mission-critical parts of the structure, this latter concern is intensified.

\* Corresponding Author, Fax: +353-61-202944, Email: michael.mccarthy@ul.ie

Current industry design methods are largely based on design charts and stress handbooks. Finite element modelling plays a limited role, analyses generally being two-dimensional. The EU research project “BOJCAS – Bolted Joints in Composite Aircraft Structures” [1], involves partners from the aircraft industry, national research laboratories and universities across Europe, and aims to produce improved analysis and design techniques of two types: global methods for fast preliminary design, and local methods for detailed design of critical joints. Detailed methods will involve three-dimensional finite element analysis, and the University of Limerick is involved in this aspect of the work.

The bolted-joint problem is clearly three-dimensional in nature. For example, single-lap joints obviously produce non-uniform stress-distributions through the thickness, while recent work by Starikov and Schön [2] has demonstrated that the same is true for double-lap joints where bolt bending is present. Effects such as bolt clamp-up are best accounted for by three-dimensional models, while the bearing mode of failure is a three-dimensional phenomenon, involving through-thickness cracks and delaminations [3, 4]. However, three-dimensional analyses have only recently begun to appear in the research literature. Ireman [5] developed a three-dimensional finite element model of a composite-to-metal, single-bolt, single-lap joint, and determined non-uniform stress distributions through the thickness of the composite laminate near the hole. Chen and Lee [6] conducted a three-dimensional analysis including progressive damage of a composite laminate with a single countersunk bolt. Camanho and Matthews [7, 8], using a pin-lap configuration with a rigid pin model, studied the progression of damage and delamination onset around the bolt-hole interface. Lin and Jen [9] investigated bolted joints with a bonded interface.

While three-dimensional analyses have been carried out by researchers, to date they have rarely been used in the aircraft industry for joint design, due to the length of time required for analyses. This problem can be broken down into time to create and post-process the models, and the time to run the models. The latter problem still exists, and even an elastic single-bolt model with sufficient mesh refinement to properly capture the stresses close to the hole, can take several hours to run on the most powerful PC processors on the market today. Modelling progressive damage and/or multi-bolt joints takes much longer. However, though it may take considerable time, the problem of run-time will eventually disappear as processor and memory capabilities continue to grow.

The other part of the problem is model creation and post-processing time. The generation of three dimensional finite element models of bolted joints is a complex, time-consuming task. As an analyst gains experience, model generation times can be greatly reduced, but in order for such analysis techniques to be accepted by industry, tools are needed to make model generation relatively simple for analysts without previous experience of bolted joint modelling. As well as making the process straightforward, it also needs to be made fast, so that several models can be created in a short period of time, for parameter studies.

The current method of creating such models involves use of finite element pre-processors such as MSC.Patran, MSC.Mentat, IDEAS etc. to generate input files for commercially available solvers such as MSC.Marc or ABAQUS. Though the procedures of generating models for bolted joints are quite general, the user needs to re-generate the models of some or all of the joint components if the joint parameters are changed. In addition, contact treatment forms an integral part of bolted joint analysis. Accurate modelling of contact interfaces between parts is crucial to the predictive capability of the simulations. In general, considerable human intervention is required to guarantee that contact conditions are accurately modelled.

This paper describes the development of a software tool (BOLJAT - Bolted Joint Analysis Tool), which is designed to simplify and speed up the process of generating three-dimensional finite element models of composite bolted joints. Once such tools are fully developed, the only barrier to using three-dimensional modelling routinely in design will be processor and memory capacity, which can be expected to be removed in a few years. The design philosophy behind the package is described along with a description of the major components within it. Then an example is given of a bolted joint study carried out using BOLJAT.

## **2. Development of BOLJAT**

### *2.1 Design philosophy*

The tool is aimed in the first instance at the aircraft industry, though use in other industries is clearly also a possibility. A recurring problem in the development of any modelling software is the wide range of finite element solvers in use. Since commercial providers of finite element pre-processing software already provide solutions to this problem, it was decided to develop the tool in the programming language of one of these

pre-processors, rather than using a general-purpose language such as C++ or FORTRAN. The pre-processor chosen was MSC.Patran, and BOLJAT has been developed using the Patran Command Language (PCL) [10, 11, 12]. MSC.Patran was chosen because it is in wide use in the aircraft industry, and appears likely to remain so for the foreseeable future. It also interfaces with most finite element solvers in use in the industry.

In general, the basic steps for performing a bolted joint analysis including contact are as follows:

- Create the model geometry of the parts to be joined.
- Create the model geometry of the bolt(s), nut(s) and washer(s).
- Mesh all parts in a way that provides refinement only where it is needed.
- Determine contacting surfaces, in a way that results in the most efficient analysis possible (contact is the chief factor in overall execution time). Depending on the solver, these surfaces may need to be separated into e.g. “master and slave”, or “contacting and contacted surfaces”.
- Define several parameters that determine contact behaviour.
- Apply necessary boundary conditions.
- Define element material properties.
- Define solution options and load steps.
- Submit the problem to a finite element solver.
- Post-process the results.

Modern software works through graphical user interfaces (GUIs), so BOLJAT provides GUIs for most of the above steps. All inputs are parameterised, so joints with different geometry and materials can be modelled easily. The crucial contact steps are almost fully automated, which shields the user from the complications of this process. The steps for geometry and mesh creation in BOLJAT provide a mesh that can be used in any finite element solver. However, there are many different methods for modelling contact and each solver tends to use its own method. Thus, to date, the contact steps in BOLJAT have been aimed at one solver only (MSC.Marc); in the future it is planned to extend this to other solvers.

BOLJAT was developed based on the following four requirements:

- (i) The intellectual overhead in learning to use the package should be low.

- (ii) The package should be flexible, i.e. it should be convenient to use in a wide variety of situations.
- (iii) It should serve users having different levels of expertise in bolted joint analysis, ranging from the casual user to the researcher.
- (iv) It should be designed so that it can be extended to add new functionalities easily.

Development has been performed on a Dell Precision 610 Workstation running under Windows NT operating system. However, no hardware or operating system dependent functions have been written into the BOLJAT code. Therefore, the BOLJAT source code can easily be compiled in any other computing environment. The MSC.Patran finite element pre-processor should be installed before installing BOLJAT. Normal PCL procedures are used for compiling the BOLJAT source code.

## *2.2 Current status*

BOLJAT has been brought to prototype status in the BOJCAS project. The current capability of BOLJAT is to create four different types of single-lap joint:

- (i) Joints with a single protruding-head bolt
- (ii) Joints with a single countersunk-head bolt
- (iii) Joints with three in-line protruding-head bolts
- (iv) Joints with three in-line countersunk-head bolts

These initial configurations were selected to coincide with those in an experimental test programme being performed by the University of Limerick in BOJCAS. In the following sections, the individual components of BOLJAT are described, with particular reference to the three-bolt, protruding-head case only.

### *2.3 BOLJAT main menu*

Fig. 1 shows the MSC.Patran main menu with BOLJAT installed. The four types of joints available are shown when the BOLJAT main menu is selected: Single Bolt (PH), Single Bolt (CS), Three Bolt (PH), and Three Bolt (CS). Upon selection of e.g. the menu item “Three Bolt (PH)”, the “Three Bolt (PH)” pull-down menu appears. Clicking on the “Three Bolt (PH)” menu item reveals another pull-down menu with twelve items, as shown in Fig. 2. Clicking on each menu item results in display of a graphical user interface (GUI). These GUIs are explained in the next sections.

### *2.4 GUI for creation of solid model of the joint plate*

When the first menu item, “Plate Solid Model”, is selected (Fig. 2), the GUI for creating the solid model of the joint plates appears, as shown in Fig. 3. As can be seen, the user has control over the dimensions of the plates, including length, width, thickness, hole radii, edge distance, and hole pitch. Default values are displayed initially, but these can be changed by the user – floating point or integer input is accepted. Presently, both plates are required to be identical, but this will be changed in future versions. Allowing separate radii for each hole, allows for the possibility of different clearances at each hole. In order to facilitate control of the finite element mesh later, the plate is divided into three zones as shown in Fig. 4. The “washer contact zone” is the zone where the washers will be in contact with the plate. Since this is a three-bolt joint case, there are three sub-zones in the washer contact zone. The “Washer Contact Zone Radius”, as shown in the GUI, is the outer radius of each of these three sub-zones. The washer contact zone comprises twenty-four solids. The “outer zone” consists of twenty-four solids, and the “non-overlap zone” comprises two solids. Upon selection of the Apply button, the solid model of the upper plate is created.

The sequence for creation of the plate solid model is as follows: A set of key points as shown in Fig. 5 is first generated. Next, the necessary edges are created. Straight lines are created from two points, while each hole consists of four arcs of a circle. The circular arcs are created by specifying the centre point, radius, start angle and end angle. The creation of the hole is crucial to the modelling – an earlier method utilising the parametric cubic interpolation function created what to the naked eye was a perfect circle, but led to

erroneous stresses at the hole. Closer examination revealed that the reason for this was that the curve produced was not a perfect circle. Surfaces are next created from the edges using the “2 curves” option in MSC.Patran. These surfaces are as shown in Fig. 5. A mirror image of these surfaces is then created to form the full plate surface. All these surfaces in the x-y plane are then swept along the z direction, to create solids.

### *2.5 GUI for creation of solid model of the bolt*

Fig. 6 shows the GUI for creating the solid model of a protruding head bolt. The bolt dimension definitions are shown in Fig. 7. The user can choose the shaft length, shaft diameter, bolt head thickness, nut thickness and bolt head diameter. At present, all bolts in the joint are assumed identical. The shaft is a cylindrical solid divided into eight sectors. For meshing purposes, the bolt head and nut consist of two zones: one is a continuation of the shaft and the other is an outer zone. Both zones in the bolt head and the nut are divided into eight solids. Clearly there are some approximations in modelling the bolt. The bolt threads are not modelled, since this would require a different order of magnitude of mesh refinement. The nut is assumed to be attached to the bolt, and pre-stress is applied in a separate step (see Section 2.9). The bolt head and nut are modelled as circular rather than hexagonal.

### *2.3 GUI for creation of solid model of the washer*

Fig. 8 shows the GUI for creating the solid model of a washer. The inner diameter, outer diameter and thickness can be set by the user. Fig. 9 shows the solid model corresponding to the default values given in Fig. 10. The solid model consists of eight solids, and all washers in the joint are presently identical.

### *2.4 GUI for creation of mesh seeds of the joint plate*

After solid models of the plate, bolt and the washer have been created, the user is ready to define finite element mesh seeds, which control the mesh density. Fig. 10 shows the GUI for defining the mesh seeds for the joint plate. Efficient control on the finite element mesh is obtained by separately defining the number of

seeds in the washer contact zone, the outer zone, and the non-overlap zone. In the washer contact zone, two seeds, radial and sector (circumferential), are defined with the first and second databoxes. The value in the third databox defines the thickness seed for all the solids in the joint plate. For the outer zone, only the radial seed needs to be defined, as the sector seed will be the same as that of the washer contact zone. However, to allow for a biased mesh in the outer zone, a bias factor also needs to be defined, so that elements closest to the hole have the shortest edge length. The non-uniform seed spacing is calculated through a geometric progression based on the given bias factor. For the non-overlap zone, only the axial mesh seed needs to be defined, since the transverse mesh seed is determined by the sector seed in the washer contact zone. With the values given in Fig. 10, the mesh seeds in Fig. 11 would be produced.

#### *2.5 GUI for creation of mesh seeds of the bolt*

Fig. 12 shows the GUI for creating a customised mesh for the bolt. The seeds in the first three databoxes control the shaft mesh. Radial and thickness seeds for the bolt head control the mesh in the bolt head, while similar seeds control the mesh in the nut. With the values given in Fig. 12, the mesh seed in Fig. 13 would be produced.

#### *2.6 GUI for creation of mesh seeds of the washer*

Fig. 14 shows the GUI for defining mesh seeds in the washer. Radial, sector and thickness seeds must be defined. With the values given in Fig. 14, the mesh seed in Fig. 15 would be produced.

#### *2.7 GUI for the assembly of the plates, bolts, washers*

Fig. 16 shows the GUI for creating and assembling the finite element joint model. Based on the above mesh seed definitions, meshes are produced for the two plates, three bolts (and nuts) and six washers, and then assembled into their correct positions in the joint. In addition, light springs are attached to the bolts, and the non-stationary plate, to avoid possible rigid body motion during the analysis process. For the bolts, springs

in the Cartesian  $x$ ,  $y$ , and  $z$  directions are attached to the bottom left and top right nodes (lying in the joint plane of symmetry). One spring in the  $y$ -direction is also attached to the bolts on the top left node (again in the joint plane of symmetry).

## 2.8 GUI for defining the contact bodies

Fig. 17 shows the GUI for defining the contact bodies for the three-bolt joint. The GUI is very simple (to the user) since the process of setting up contact has been automated. This is the only part of the process that is specific to one solver (MSC.Marc). MSC.Marc requires definition of “contact bodies” which are the bodies that can potentially be in contact. On selection of the Apply button, all the contact bodies for the three-bolt joint are found (see Fig. 18). The easiest way to define the contact bodies would be to simply indicate the eleven physical bodies in the joint (two plates, three bolts/nuts and six washers). However a more efficient solution (in terms of CPU time), is produced with smaller, localised contact bodies, but the bodies must be adequate to capture all possible contact. BOLJAT first identifies the potential contact surfaces, and then it extracts the nodes associated with the contact surfaces. The elements associated with these nodes are then specified as contact bodies. The contact tolerance is also set automatically. This value is used to decide if a node is “in contact”. If the trial position of the node is within the contact tolerance zone, it is considered to be in contact. In the “direct-constraint” method used in MSC.Marc, once a node is detected to be in contact with a segment, the node is placed on that segment by means of a multi-point constraint (a so-called “tying” constraint), so penetration does not occur. If the node lies beyond the contact zone, it is considered to have penetrated and the current solution increment is split and a new trial position found – this continues until the node is within the tolerance zone. Too small a tolerance leads to a lot of increment splitting (and hence high computational cost), but too large a tolerance leads to premature contact detection. Because the gap between bolts and holes can be very small, a contact tolerance smaller than the usual default value is used by BOLJAT for the bolt-hole contact pair. This enables correct modelling of joints with small clearances.

### *2.9 GUI for application of boundary conditions*

In BOLJAT, boundary conditions are applied to the solid model instead of the finite element model. This is because solid model loads are independent of the finite element mesh, i.e. it is possible to change the element mesh without affecting the applied boundary conditions. Solid model loads are automatically transferred to the finite element model when the analysis specific code is generated.

Fig. 19 is the GUI for applying the boundary conditions on the three-bolt joint. Upon selection of the Apply button, fixed-displacement boundary conditions are applied to the nodes of the left end face of the stationary plate. A default x-displacement of 1.012mm is applied to the nodes of the right end face of the lower (non-stationary) plate – this value can be changed by the user later by editing the analysis code generated by BOLJAT. Motion in the y- and z- directions is restrained for these nodes. In addition, nodes lying on the symmetry plane, belonging to the bolts and washers (but excluding the potential contact nodes) are restrained in the y-direction. This has been found necessary (to date) to get a convergent solution. Finally, in order to apply a pre-load to the bolt a temperature loading is applied to the upper washers. This causes the washers to expand in the axial direction of the bolt, thereby stretching the bolt and compressing the laminates, which is essentially what happens during torquing of the bolt. A graph of temperature change versus bolt pre-load for a particular joint configuration can be quickly generated by running a few models with temperature loading only. All bolts in the joint are currently assumed to be torqued to the same levels.

### *2.10 GUIs for defining the material properties of the joint plates, bolts and washers*

The GUI for generating the material properties of the joint plates is shown in Fig. 20. As can be seen the plate can be modelled with homogeneous, orthotropic properties. Isotropic properties (for a metal joint) can also be input of course. Non-homogeneous, layered properties are not yet available in BOLJAT, though the user can overwrite the homogeneous properties created by BOLJAT, if necessary. Presently, both plates are defined to have the same material properties. The GUIs for setting the material properties for the bolts and washers are similar to the one shown, except for the fact that they use isotropic mechanical properties.

### 3. Case study using BOLJAT to study of the effects of bolt-hole clearance

It is well known that joint geometry affects the stress state in a bolted joint, and there have been many studies on these effects (mostly with two-dimensional finite element or analytical models). A review of some studies concerning the effect of joint geometry on joint behaviour was recently performed by Camanho and Matthews [13].

To illustrate the use of BOLJAT in investigating geometry effects, this section describes three-bolt joint models with two different clearance configurations, which were generated with BOLJAT. The results are compared with experimental results.

#### 3.1 Problem definition

The geometric dimensions for the three-bolt joint are given in Fig. 21. The bolt numbering scheme and boundary conditions are shown in Fig. 22. Specifications for the clearance codes are listed in Table 1. In the C1\_C1\_C1 joint, all three clearances are essentially neat-fit, while the C4\_C1\_C1 joint has one relatively loose-fitting bolt. In the experiments, the clearances were obtained with specially made reamers with a tight tolerance. The carbon fibre/epoxy material for the joint plates was HTA/6376, a high-strength material currently used in the aircraft industry. The laminate stacking sequence was quasi-isotropic consisting of forty plies ( $[45/0/-45/90]_{5s}$ ), yielding a nominal laminate thickness of 5.2 mm. The orthotropic material properties for this lay-up are given in Table 2. The protruding head bolts were Titanium alloy 6Al-4V, while the washers and nuts were steel. Bolts were tightened to “finger-tight” levels, which amounted to a torque of approximately 0.5 Nm. All bolts were initially centred in their holes in the simulation. This was also done in the experiments with a special jig.

### 3.2 Model parameters

The models were frictionless, which was assumed reasonable given the low torque level. The model had 17 contact bodies and 19 contact pairs. All contact pairs were of three-dimensional deformable-deformable type. Single-sided contact was used in the analysis.

By default, MSC.Marc detects the possibility of contact relative to all other bodies. This is not desirable here since not all contact bodies contact each other (e.g. upper and lower washers never contact each other). Hence, the default feature was overridden using the CONTACT TABLE option of MSC.Marc, which specifies which contact bodies can potentially contact each other. The MSC.Marc “analytical contact” option was used to improve the accuracy for a deformable-deformable contact analysis.

For each clearance case, the finite element model had 18096 elements and 23696 nodes. Eight-node, three-dimensional, arbitrarily distorted brick elements (MSC.Marc element type 7) were used. The element has three displacement degrees of freedom at each node, and eight Gauss integration points. The sparse direct solver was used in the analysis.

### 3.3 Results

Fig. 23 shows the load versus deflection from the simulations for the two different clearance cases. The C4\_C1\_C1 joint shows an initial lower stiffness when compared to the C1\_C1\_C1 joint. This is because the load in the C1\_C1\_C1 joint is taken up by all three bolts from the start. In contrast, in the C4\_C1\_C1 joint, only Bolts 2 and 3 carry load until the clearance is taken up in Hole 1. This can be seen clearly in Fig. 24, which shows the bolt load distribution in each case. In the C1\_C1\_C1 joint, the outer two bolts take more load than the inner bolt, which is in line with accepted design rules. However, in the C4\_C1\_C1 joint, Bolt 1 does not initially take any load, so that the load is shared between Bolts 2 and 3. The bolt loads were calculated by summing up the x-components of all the contact forces on each hole. Also shown in Fig. 24, is the load distribution obtained experimentally using instrumented bolts – for further details on these bolts see [14]. As can be seen, there is excellent agreement between the numerical and experimental results.

The change in load distribution due to clearance can change the failure mode of the joint, as can be seen in Fig. 25. In the C1\_C1\_C1 joint, ultimate failure was through failure of Bolt 1. This is in line with accepted

design rules, which would predict equal likelihood of failure in Bolt 1 and 3. However, in the C4\_C1\_C1 joint, failure occurred simultaneously in Bolts 2 and 3. Failure of Bolt 2 would not normally be considered possible, but occurred in this case because of the large clearance in Hole 1. Fig. 26 illustrates the distribution of  $\epsilon_{xx}$  in each joint. It can be seen that in single-lap joints like these, the strain distribution is fully three-dimensional, reaching a maximum at the shear plane (faying surface). Two-dimensional analyses obviously cannot capture this. The low strain values in the loose-fit hole are clearly evident in the C4\_C1\_C1 case.

The results of this study show that three-dimensional finite element modelling can accurately capture the effects of small changes in variables such as clearance, and can thus be used to determine the correctness and limitations of existing design rules, as well as to generate new ones, in association with experiments. It can also help to direct experimental programmes for new designs, reducing the number of required tests. By automating the time-consuming model-creation process, BOLJAT makes the use of three-dimensional analysis a more realistic alternative in design.

#### **4. Conclusions**

A tool has been presented for design of composite bolted joints using three-dimensional finite element analysis. As computing capabilities continue to grow, the main barrier to using three-dimensional analysis routinely in joint design will be pre/post processing time. BOLJAT presently addresses pre-processing though it is planned to also incorporate automation of some standard post-processing tasks such as retrieving bolt load distributions, or determining the load transfer due to friction. BOLJAT allows fast creation of several models with different joint parameters and is easy to use. A particularly valuable aspect of BOLJAT is the automated treatment of contact, incorporating the authors' experience with manual modelling of bolted joints. For the first-time modeller, it can take a great deal of time to achieve a correctly-working bolted joint contact analysis, so BOLJAT can shield the user from these problems.

Future short-term developments include: extension to other configurations (e.g. double-lap, other simple bolt patterns), allowing each plate in the model to have different dimensions and material properties, and allowing variable bolt diameters. In the medium term it is planned to extend BOLJAT to interface with other

solvers such as ABAQUS, and incorporate automated post-processing features. Long-term, expansion to much more complex joint configurations is possible, as well as incorporation of progressive damage routines.

### **Acknowledgements**

BOJCAS - Bolted Joints in Composite Aircraft Structures is a RTD project partially funded by the European Union under the European Commission GROWTH programme, Key Action: New Perspectives in Aeronautics, Contract No. G4RD-CT99-00036.

### **References**

1. McCarthy MA. BOJCAS: Bolted Joints in Composite Aircraft Structures. *Air and Space Europe* 2001; 3-4(3): 139-142.
2. Camanho PP, Bowron S, and Matthews FL. Failure mechanisms in bolted FRP. *J. Reinforced Plastics and Composites* 1998; 17(3): 205-233.
3. Wang H, Hung C, and Chang F. Bearing failure of bolted composite joints. Part I: Experimental Characterisation. *J. Composite Materials* 1996; 30(12): 1285-1313.
4. Starikov R, Schön J. Quasi-static behaviour of composite joints with protruding-head bolts. *Composite Structures* 2001; 51: 411-425.
5. Ireman T. Three-Dimensional Stress Analysis of Bolted Single-Lap Composite Joints. *Composite Structures* 1998; 43: 195-216.
6. Chen WH, Lee SS. Numerical and experimental failure analysis of composite laminates with bolted joints under bending loads. *J. Composite Materials* 1995; 29(1): 15-36.
7. Camanho PP, Matthews FL. A Progressive Damage Model for Mechanically Fastened Joints in Composite Laminates. *Journal of Composite Materials* 1999; 33(24): 2248-2280.
8. Camanho PP, Matthews FL. Delamination Onset Prediction in Mechanically Fastened Joints in Composite Laminates. *Journal of Composite Materials* 1999; 33(10): 906-927.

9. Lin WH, Jen MHR. The Strength of Bolted and Bonded Single-Lapped Composite Joints in Tension. *Journal of Composite Materials* 1999; 33(7): 640-666.
10. PCL Reference Manual, Volume 1, Function Descriptions, Version 9, MSC Software Corporation, CA (USA) (1999).
11. PCL Reference Manual, Volume 2, Code Examples, Version 9, MSC Software Corporation, CA (USA) (1999).
12. MSC.Patran PCL and Customisation, Version 9, MSC Software Corporation, CA (USA) (1999).
13. Camanho PP, Mathews FL. Stress Analysis and Strength Prediction of Mechanically Fastened Joints in FRP: A Review. *Composites Part A* 1997; 28A: 529-547.
14. Stanley WF, McCarthy MA, Lawlor VP. Measurement of Load Distribution in Multi-bolt, Composite Joints, in the Presence of Varying Clearances, Ninth International Conference on Fibre-Reinforced Composites, University of Newcastle, 25-28 March, 2002; 296-307.

Joint Code	Hole 1 Clearance	Hole 2 Clearance	Hole 3 Clearance
C1_C1_C1	C1 (10 microns)	C1 (10 microns)	C1 (10 microns)
C4_C1_C1	C4 (240 microns)	C1 (10 microns)	C1 (10 microns)

**Table 1: Clearances in multi-bolt case study**

$E_{11}$ (N/mm <sup>2</sup> )	$E_{22}$ (N/mm <sup>2</sup> )	$E_{33}$ (N/mm <sup>2</sup> )	$G_{12}$ (N/mm <sup>2</sup> )	$G_{13}$ (N/mm <sup>2</sup> )	$G_{23}$ (N/mm <sup>2</sup> )	$\nu_{12}$	$\nu_{23}$	$\nu_{31}$
54200	54200	12600	20700	4550	4550	0.309	0.336	0.0796

**Table 2: Orthotropic material properties for the joint plates**

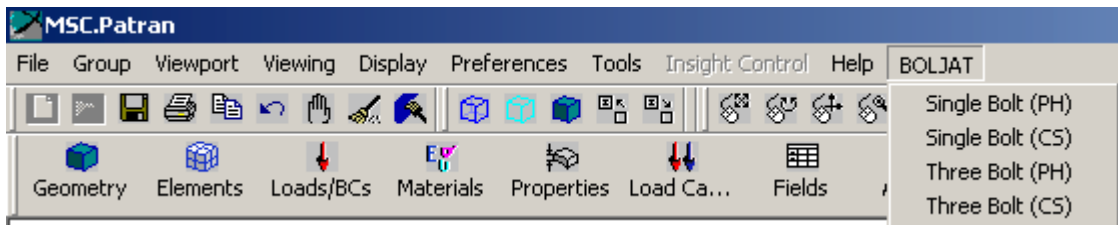


Figure 1. Patran main menu with BOLJAT

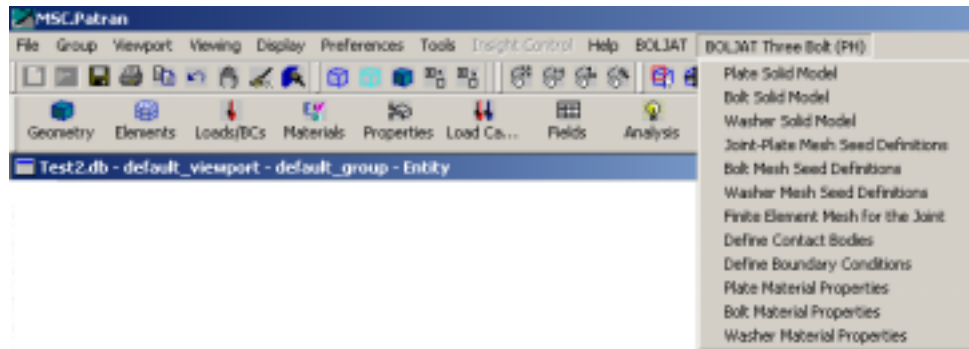


Figure 2. BOLJAT Three Bolt (PH) menu



Figure 3. GUI for creation of solid model of plate (three bolt, protruding head)

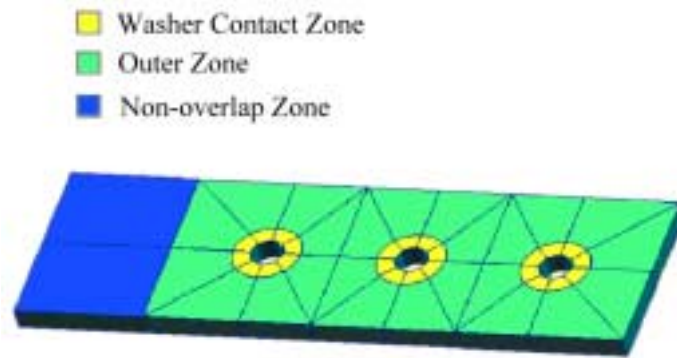


Figure 4. Zones in the upper plate of the joint (three bolt, protruding head)

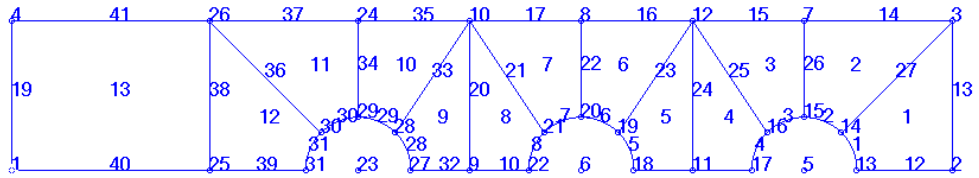


Figure 5. Necessary points, curves, surfaces for half of upper surface of joint plate

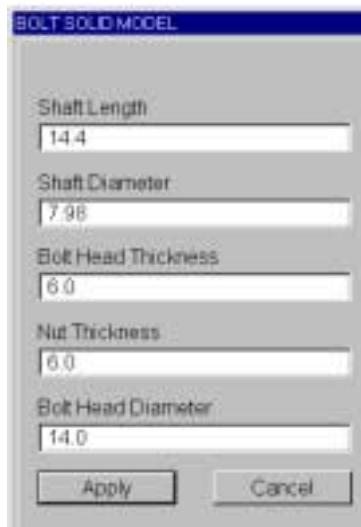


Figure 6. GUI for creation of solid model of bolt (protruding head)

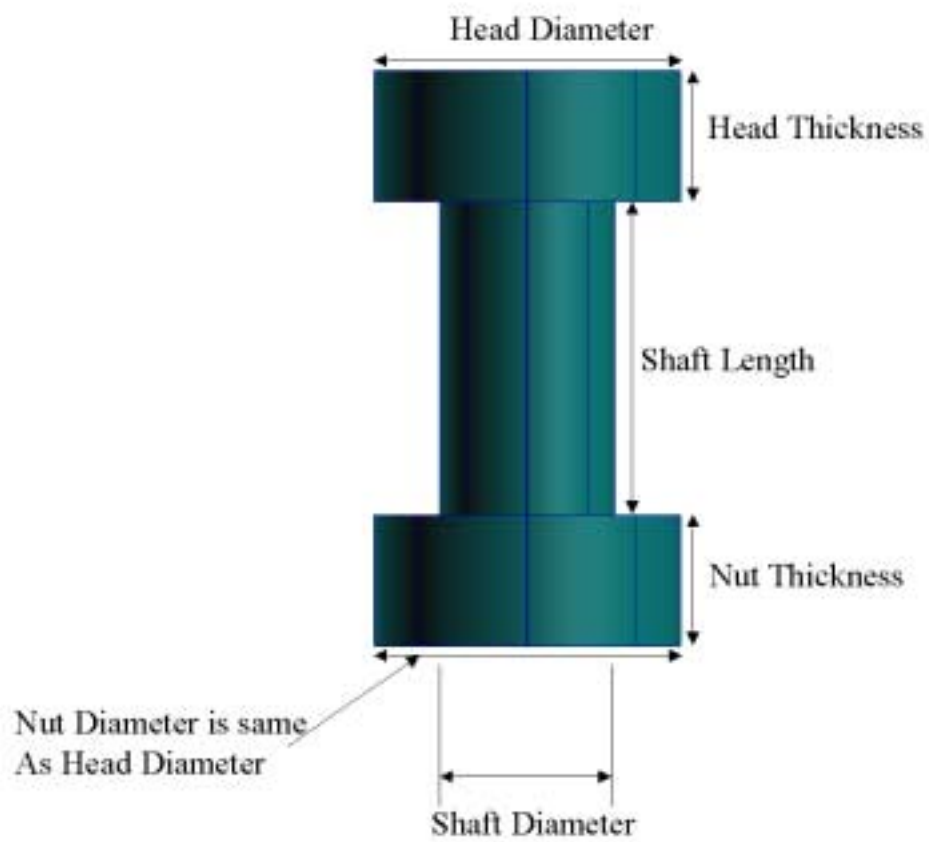


Figure 7. Protruding head bolt dimension definitions



Figure 8. GUI for creation of solid model of washer (three bolt, protruding head)

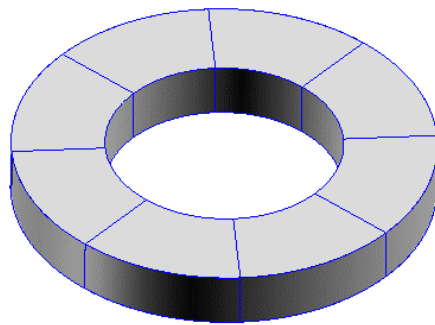


Figure 9. Solid Model of the Washer



Figure 10. GUI for creation of mesh seeds on the joint plate (three bolt, protruding head)

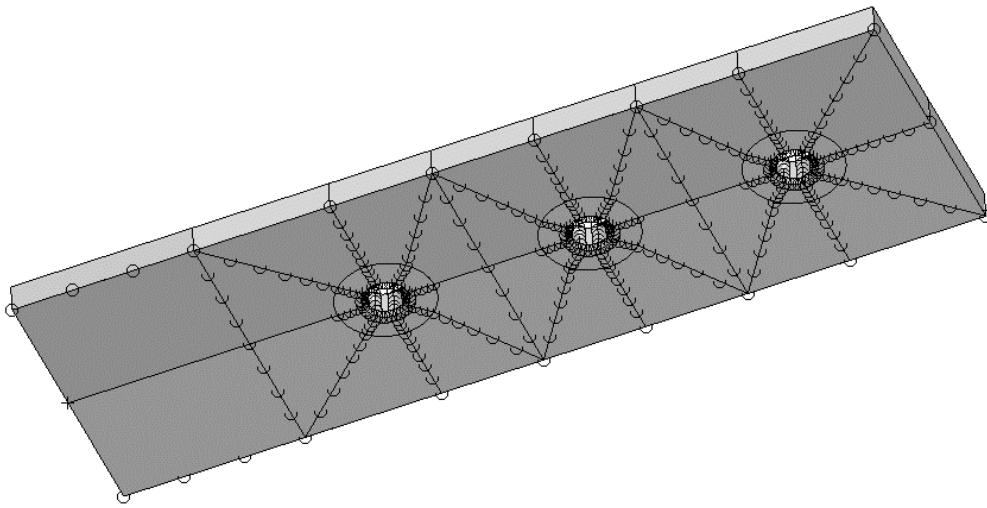


Figure 11. Mesh seeds on upper plate (three bolt, protruding head)

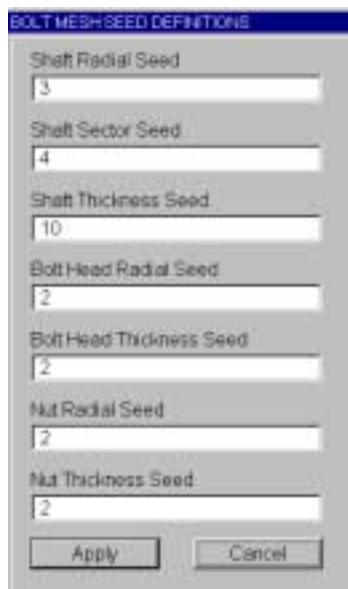


Figure 12. GUI for creation of mesh seeds on the bolt (protruding head)

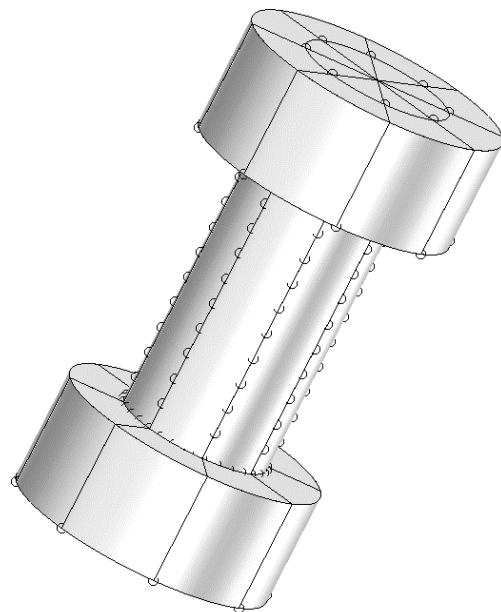


Figure 13. Mesh seeds on Bolt



Figure 14. GUI for creation of mesh seeds on washer

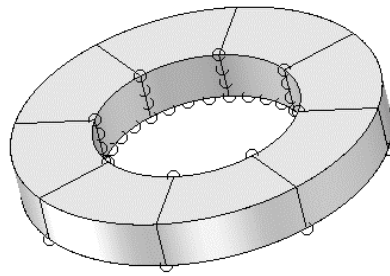


Figure 15. Mesh seeds on washer



Figure 16. GUI for creation and assembly of finite element mesh



Figure 17. GUI for creating the contact bodies

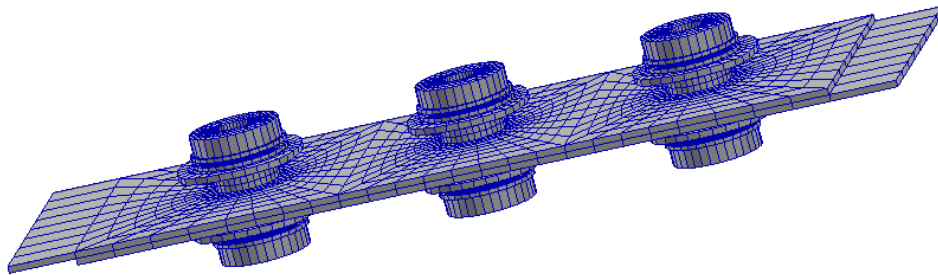


Figure 18. Contact bodies (three bolt, protruding head)

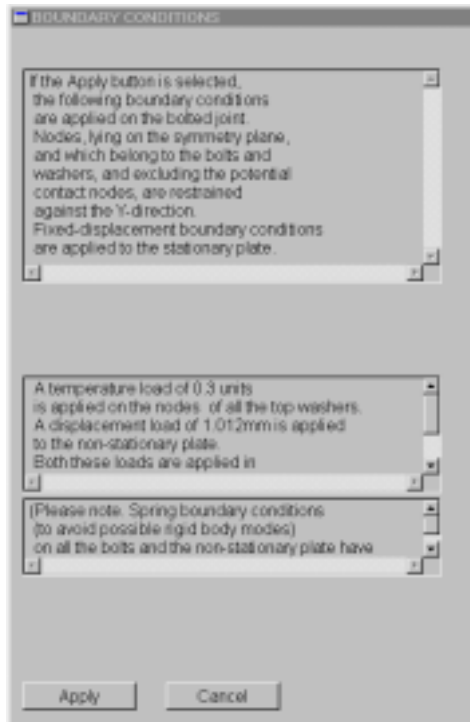


Figure 19. GUI for application of boundary conditions

The image shows a software dialog box titled "MATERIAL PROPERTIES FOR JOINT PLATE". It contains several input fields for material properties, each with a label and a text box containing a numerical value. At the bottom, there are "Apply" and "Cancel" buttons.

Property Label	Value
E11	54200.0
E22	54200.0
E33	12600.0
NU12	0.308
NU23	0.336
NU31	0.0796
G12	20700.0
G23	4550.0
G31	4550.0

Figure 20. GUI for plate material properties

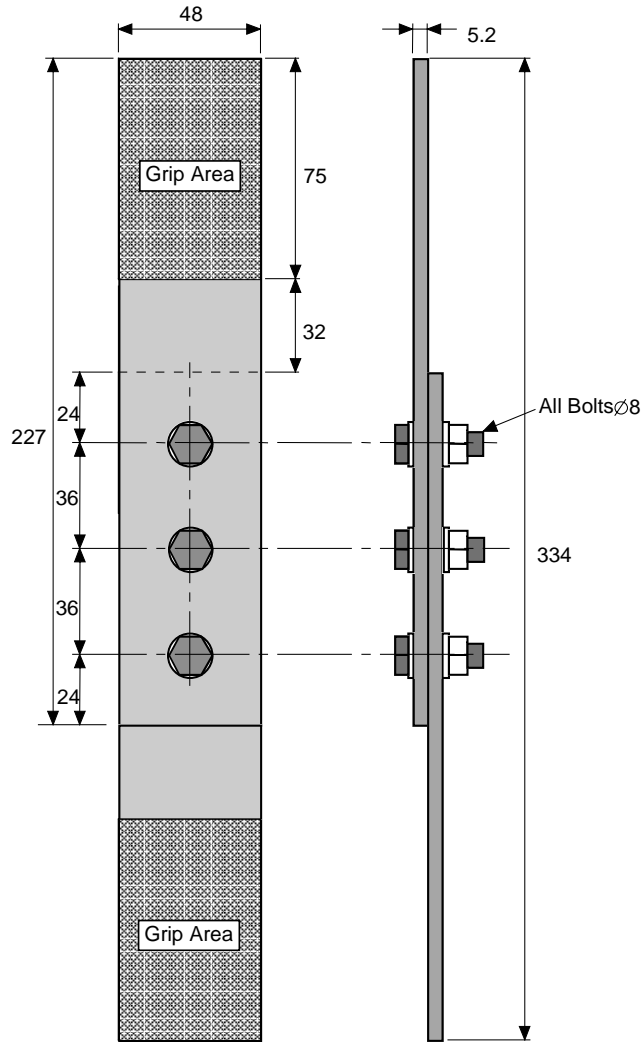


Figure 21. Single-lap three-bolt geometry (all dimensions in mm)

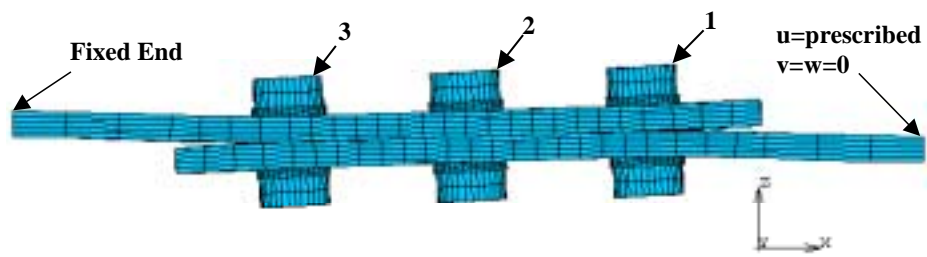


Figure 22. C1\_C1\_C1 deformed shape (display magnification factor = 2) with boundary conditions and bolt numbering

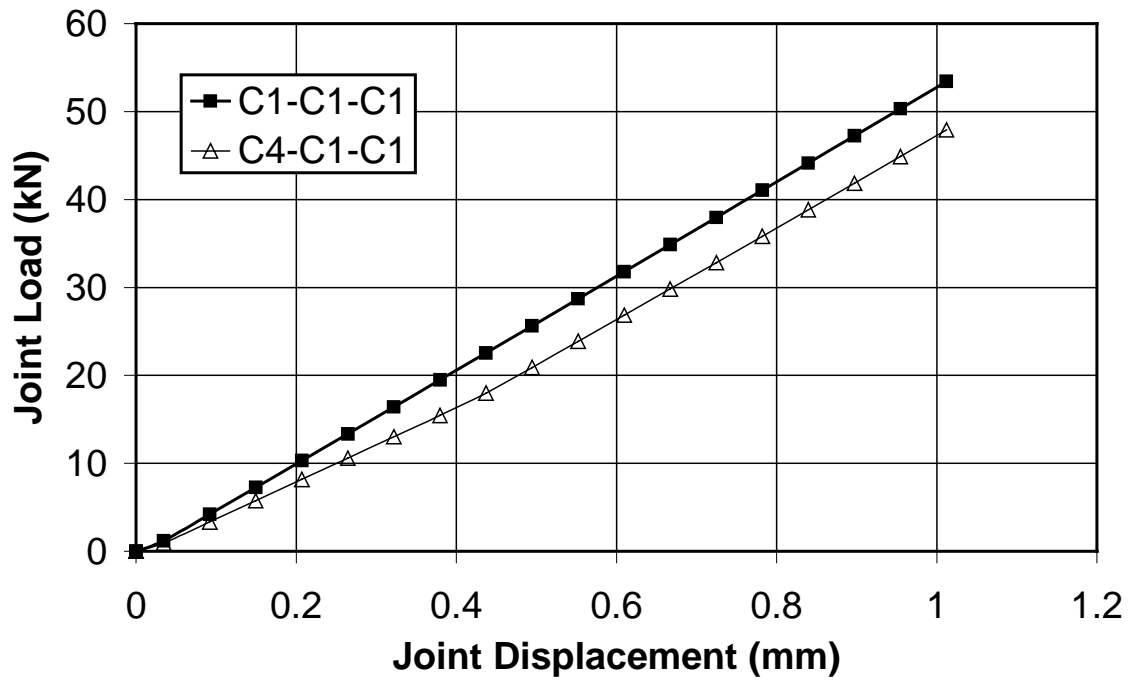


Figure 23. Load vs. deflection (from FE analysis) for C1\_C1\_C1 and C4\_C1\_C1 clearance cases

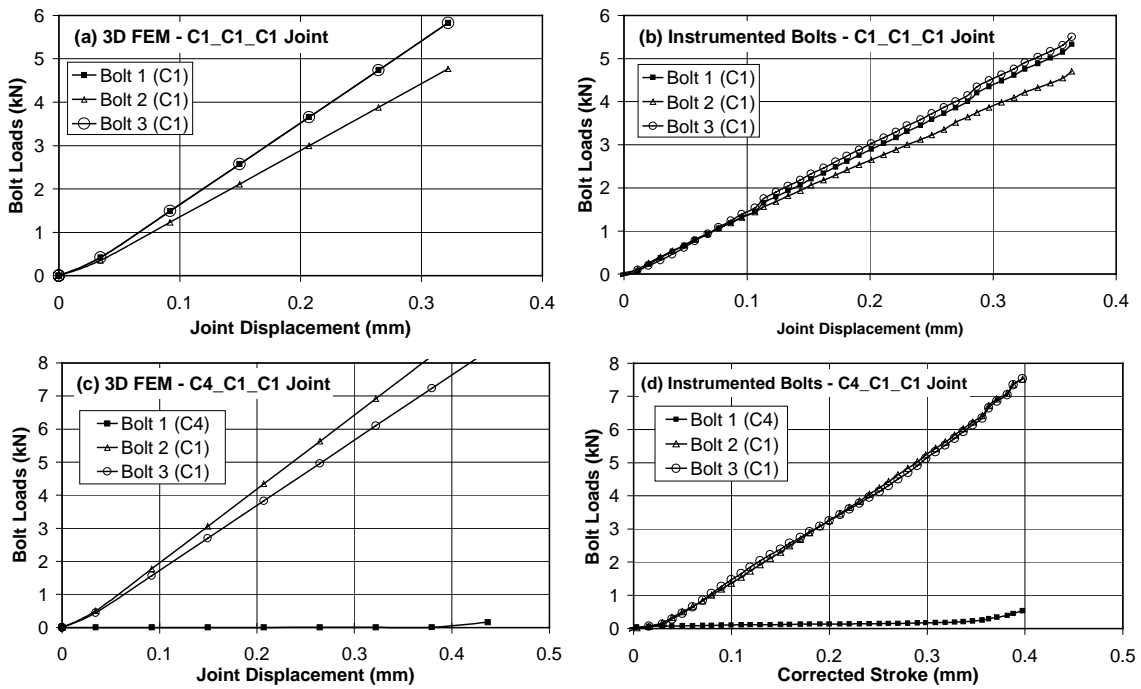


Figure 24. Bolt Load distribution  
 (a) C1\_C1\_C1 joint, FE analysis  
 (b) C1\_C1\_C1 joint, instrumented bolts  
 (c) C4\_C1\_C1 joint, FE analysis  
 (d) C4\_C1\_C1 joint, instrumented bolts

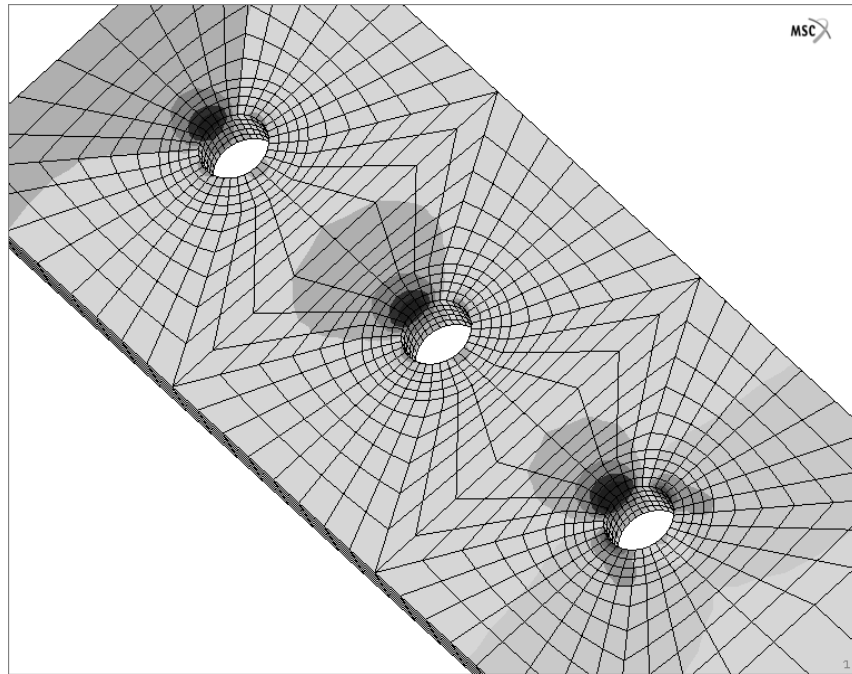


(a) C1\_C1\_C1 joint

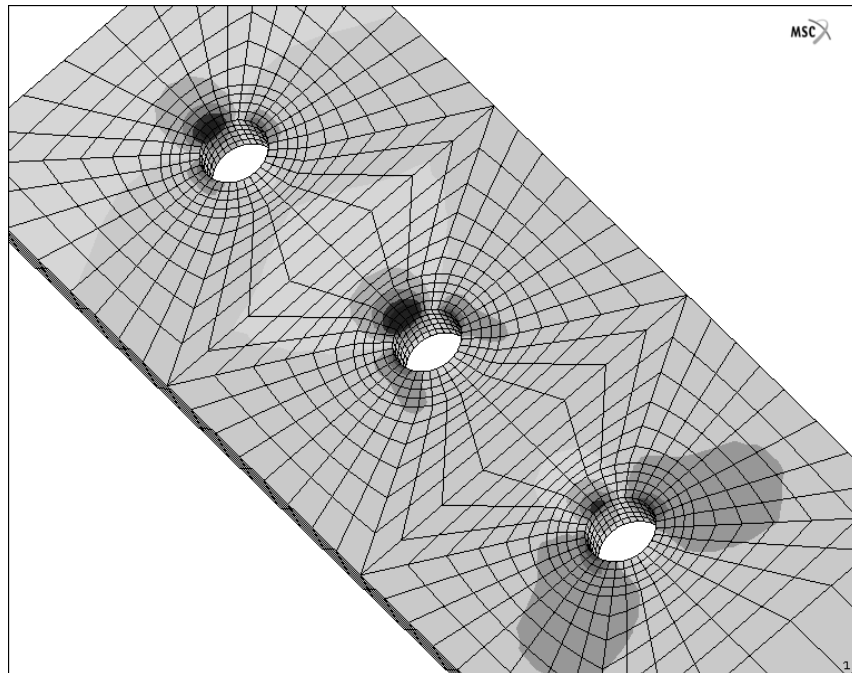


(b) C4\_C1\_C1 joint

Figure 25 Ultimate failure modes



(a) C1\_C1\_C1 joint



(b) C4\_C1\_C1 joint

Figure 26 Distribution of  $\epsilon_{xx}$  - view shown is looking down on shear plane (faying surface)