



Norwegian University
of Life Sciences

Master's Thesis 2017 30 ECTS
Faculty of Science and Technology

Analyzing the behavior of a hybrid steel to timber connection by modifying different parameters.

Youcef Amara and Samuel Embaye

Preface and acknowledgements

This master thesis is written in the spring semester of 2017 at the Faculty of Science and Technology at the Norwegian University of Life Science (NMBU). It is the final work of a 2-year master's degree in Building technique and Architecture. The supervisor through the semester has been Associate Professor Themistoklis Tsalkatidis.

The goal of this thesis is to analyse the behaviour of a hybrid connection between a tubular steel column and a glulam beam, that are connected together with top and seat angles by the means of bolts. Different parameters are examined in order to capture the exact physical behaviour of the connection and to monitor the effect of each alteration. This type of hybrid connection has not been thoroughly examined in the past, especially in relation to changing several different parameters in order to have a simulation as realistic as possible. This is the main contribution of the presented thesis.

The major part of this thesis has been spent on modelling the connections. Many different scripts have been written and changed multiple times in order to achieve an accurate model. The authors have limited experience in contact problems and use of a finite element software, so the learning curve had to increase abruptly.

We would like to sincerely thank our supervisor, Associate Professor Themistoklis Tsalkatidis, for his guidance, support and ongoing positive attitude towards our thesis throughout the past semester. We have learned a lot by working on the hybrid connection and the topic was proven to be an excellent choice that gave us more insight about contact finite element problems. We would also like to thank Alejandro Figueres for his opinions and thoughts on the different obstacles that we ran into throughout the semester. Finally, we want to thank our families for their patience and for their understanding for all the late nights spent at the university working on our thesis.

Sammendrag

Det er ikke forsket like mye på knutepunkt mellom stål og tre, som det er forsket på knutepunkt mellom stål og betong. Ved å kombinere styrken til stål med den lave vekten til tre, får vi et miljøvennlig knutepunkt som kan være med å bidra betraktelig til et knutepunkt for fremtiden.

Ved å starte med teoretisk bakgrunn presenterer denne masteroppgaven prosessen ved å modellere et knutepunkt og dets tilhørende parametere. Det legges vekt på bruken av viktige parametere og effekten av å endre disse parametere i knutepunktet. En numerisk modell ble laget og verifisert slik at denne er troverdig. Normalt utføres det et eksperiment før det lages en numerisk modell slik at denne kan bli verifisert, men på grunn av mangel på ressurser her på universitetet kunne dette ikke utføres. Derfor fant vi etter søk på internasjonal litteratur, en publisert studie som vi har valgt å bruke som verifikasjon. Et av knutepunktene som er eksperimentelt testet i dette studiet er valgt som den grunnleggende knutepunkts modellen som har blitt numerisk analysert i denne oppgaven. Sammenligningen mellom eksperimentet og den numeriske modellen's resultater er grunnlaget for verifiseringen av den numeriske modellen. Etter at verifiseringen har blitt gjort har flere parametere fra den opprinnelige modellen blitt forandret slik at det kan analyseres hvor stor effekt disse forandringene har. Flere knutepunktsmodeller har blitt undersøkt og en optimal sammensetning har blitt funnet, og dette er sannsynligvis hovedbidraget fra denne masteroppgaven

Analysene har blitt gjort med bruk av skripting i Finite-element programmet Ansys Mechanical APDL. Dette er et avansert program som krever at enhver parameter er korrekt for å kunne produsere nøyaktige resultater. Den opprinnelige modellen består av vinkler med 15mm tykkelse som er montert oppe og nede på limtrebjelken med 12mm gjennomgående bolter, og 16mm bolter som kobler vinklene sammen til stålsøylen. Totalt ble syv knutepunktsmodeller undersøkt og det optimale knutepunktet besto av en kombinasjon av to forskjellige knutepunktsmodeller. Det ble funnet ut at ved å kombinere 20mm vinkler både oppe og nede på limtrebjelken, sammen med 20mm gjennomgående bolter ville rotasjonsstivheten øke markant med 137% ,mens von Mises stress vil synke med 20.2%

Abstract

The behaviour of hybrid connections between timber and steel has not been investigated as extensively as the connections between concrete and steel. By combining the strength of steel with the low weight of timber, an environment friendly connection is constructed and this option may be the rule for structural connections in the future.

By starting with theoretical background, this thesis presents the process of modelling a connection and the related parameters. It emphasizes on the use of important parameters and the effect of changes in these parameters on the connection. A base numerical model was constructed and verified, in order for this to be reliable. Normally an experimental analysis is done before developing a numerical model, so the latter can be verified, but due to lack of resources at the university, an experiment could not be done. Therefore, after searching the international literature, a research paper has been selected. A connection tested in this paper was selected as the base connection that was numerically analysed in this thesis. Moreover, the comparison between experimental and numerical results lead to the verification of the numerical model. After the verification was made, many different parameters of the base connection were altered in order to analyse and quantify the effect of these changes. Many case studies were examined. An optimal configuration has been found and this is probably the main contribution of the current thesis.

The analyses were done with scripting, using the Finite element program Ansys Mechanical APDL. This is a complex detailed program that needs every parameter to be correct in order to produce accurate results. The base model consists of 15mm thick top and seat angle connected with 12mm bolts that runs through the glulam beam and 16mm bolts connected to the tubular steel column. Together with the base model, a total of seven case studies have been investigated whereas the optimal connection was found to be a combination of two cases. It was found that by combining a 20 mm thick top and seat angle and 20 mm vertical bolts the rotational stiffness of the connection increased significantly with 137%, while the von Mises stress in the top angle was reduced by as much as 20.2%.

Content

1 Introduction	1
1.1 Background	1
1.2 Area of focus	1
1.3 Limitations	2
2 Theory	3
2.1 Glulam	3
2.2 Steel	6
2.3 Connections	7
2.3.1 The moment-rotation relationship	8
2.3.2 Component based method	9
2.3.3 Classification of connections by stiffness	10
2.3.4 Classification of connections by strenght	13
2.4 Bolted connection	14
2.5 Finite element method	16
3 Finite Element Model	18
3.1 Ansys Mechanical APDL	18
3.2 Model parameters	19
3.2.1 Element type	19
3.2.2 Contact	21
3.2.2.1 Contact elements	22
3.2.2.2 Target elements	23
3.2.2.3 Symmetric and asymmetric pairs	25
3.2.2.4 Contact analysis type	26
3.2.2.5 Contact method	27
3.2.3 Contact properties	27
3.2.3.1. Contact surface behavior	28
3.2.3.2 Contact detection	29
3.2.3.3 Normal penalty stiffness	30
3.2.3.4 Penetration tolerance	30
3.2.3.5 Pinball region	30
3.2.3.6 Coulomb's friction	31

3.2.4 Contact Algorithm	31
3.2.4.1 The penalty method	31
3.2.4.2 The Augmented Lagrange method	31
3.2.4.3 The pure Lagrange multiplier method	32
3.2.4.4 Multipoint constraint algorithm	32
3.3 Equation solver	32
3.3.1 The sparse direct solver	34
3.3.2 The Preconditioned conjugate gradient (PCG)	34
3.3.3 The Jacobi conjugate gradient solver	35
3.3.4 The Incomplete Cholesky Conjugate Gradient Solver	35
3.4 Newton-Raphson procedure	35
3.5 Pretension	37
3.6 MESH	38
4 Numerical Modelling	40
4.1 Scripting in Ansys	40
4.2 Connection description	40
4.3 Assumptions and simplifications	42
4.4 Contact parameters used in ANSYS APDL	44
4.4.1 Element type	44
4.4.2 Contact and target elements	44
4.4.3 Contact analysis type	45
4.4.4 Contact surface behavior	45
4.4.5 Friction	45
4.4.4 Contact algorithm	46
4.4.5 Equation solvers	46
4.5 External loading, pretension, and boundary conditions	46
4.6 Singularities	48
4.7 Model validations	48
4.8 Case studies	50
4.8.1 Moment rotation	51
4.8.2 von Mises stress	52
4.8.3 Initial rotational stiffness	52
5 Results	55

5.1 Case 1	55
5.2 Case 2	59
5.3 Case 3	63
5.4 Case 4	68
5.5 Case 5	72
5.6 Case 6	76
5.7 Case 7	81
5.8 Initial stiffness of connections	85
6 Discussion	86
6.1 Effect of changing the angle thickness	86
6.2 Effect of changing the diameter of the bolts connected to the beam	87
6.3 Effect of reducing the width of the angles	87
6.4 Effect of using a stiffener in the top angle	87
6.5 Seat angles	88
6.6 Wood failure	88
6.7 Vertical bolts	89
6.8 Initial rotational stiffness	89
7 Conclusion	91

List of Figures

Figure 2.1. Glued-laminated timber manufacturing process.....	4
Figure 2.2.frequency distribution of the ultimate strength of glulam and structural timber:.....	5
Figure 2.3. Stress-strain relationship.....	6
Figure 2.4. Different classes of connections.....	8
Figure 2.5 Possible idealizations for M-Ø curves.....	9
Figure 2.6. Connection stiffness in M-Ø curve.....	11
Figure 2.7 Schematization of rotational stiffness.....	12
Figure 2.8 Moment rotation curve according to strength.....	13
Figure 2.9 Load transfer in non-preloaded bolt.....	15
Figure 2.10. Load transfer in preloaded bolt.....	15
Figure 3.1 Modules in Ansys.....	18
Figure 3.2 Element geometry SOLID185.....	20
Figure 3.3 Element geometry SOLID186.....	20
Figure 3.4 Element geometry SOLID187.....	21
Figure 3.5 CONTA173.....	22
Figure 3.6 CONTA174.....	23
Figure 3.7 Functionality of the target elements.....	24
Figure 3.8 Principals of the target elements.....	24
Figure 3.9 Schematic representation of symmetric and asymmetric contact.....	26
Figure 3.10 Contact detection at Gauss point.....	29
Figure 3.11 Contact detection at Nodal detection.....	29
Figure 3.12 First iteration in Newton-Raphson procedure.....	36
Figure 3.13 second iteration in Newton-Raphson procedure.....	37
Figure 3.14 Tension after adjustments.....	38
Figure 3.15 fine vs coarse mesh.....	39
Figure 3.16 free vs mapped mesh.....	39
Figure 4.1 Abaqus vs. Ansys Model.....	41
Figure 4.2 Top and seat angle.....	42
Figure 4.3 Behaviour of angle.....	43
Figure 4.4 Constraining the column.....	47
Figure 4.5 Verification of the model.....	49

Figure 4.6 Deflection in the top angle with and without stiffener.....	52
Figure 5.1 The moment-rotation curve in case 1.....	55
Figure 5.2 von Mises stress in the top angle in case 1.....	56
Figure 5.3 von Mises stress in the seat angle in case 1.....	57
Figure 5.4 Stress perpendicular to the glulam beam at the first load step in case 1.....	57
Figure 5.5 Stress perpendicular to the glulam beam at the second load step in case 1.....	58
Figure 5.6 von Mises stress in the vertical bolts in case 1.....	58
Figure 5.7 Comparison of moment-rotation curves.....	59
Figure 5.8 von Mises stress in the top angle in case 2.....	60
Figure 5.9 von Mises stress in the seat angle in case 2.....	61
Figure 5.10 Stress perpendicular to the glulam beam at the first load step in case 2.....	61
Figure 5.11 Stress perpendicular to the glulam beam at the second load step in case 2.....	62
Figure 5.12 von Mises stress in the vertical bolts in case 2.....	62
Figure 5.13 Comparison of moment-rotation curves.....	64
Figure 5.14 von Mises stress in the top angle in case 3.....	65
Figure 5.15 von Mises stress in the seat angle in case 3.....	65
Figure 5.16 Stress perpendicular to the glulam beam at the first load step in case 3.....	66
Figure 5.17 Stress perpendicular to the glulam beam at the second load step in case 3.....	66
Figure 5.18 von Mises stress in the vertical bolts in case 3.....	67
Figure 5.19 von Mises stress in the top angle in case 4.....	69
Figure 5.20 von Mises stress in the seat angle in case 4.....	69
Figure 5.21 Stress perpendicular to the glulam beam at the first load step in case 4.....	70
Figure 5.22 Stress perpendicular to the glulam beam at the second load step in case 4.....	70
Figure 5.23 von Mises stress in the vertical bolts in case 4.....	71
Figure 5.24 Comparison of moment-rotation curves.....	72
Figure 5.25 von Mises stress in the top angle in case 5.....	73
Figure 5.26 von Mises stress in the seat angle in case 5.....	74
Figure 5.27 Stress perpendicular to the glulam beam at the first load step in case 5.....	74
Figure 5.28 Stress perpendicular to the glulam beam at the second load step in case 5.....	75
Figure 5.29 von Mises stress in the vertical bolts in case 5.....	75
Figure 5.30 Comparison of moment-rotation curves.....	77
Figure 5.31 von Mises stress in the top angle in case 6.....	78
Figure 5.32 von Mises stress in the seat angle in case 6.....	78
Figure 5.33 Stress perpendicular to the glulam beam at the first load step in case 6.....	79

Figure 5.34 Stress perpendicular to the glulam beam at the second load step in case 6.....	79
Figure 5.35 von Mises stress in the vertical bolts in case 6.....	80
Figure 5.36 Comparison of moment-rotation curves.....	81
Figure 5.37 von Mises stress in the top angle in case 7.....	82
Figure 5.38 von Mises stress in the seat angle in case 7.....	83
Figure 5.39 Stress perpendicular to the glulam beam at the first load step in case 7.....	83
Figure 5.40 Stress perpendicular to the glulam beam at the second load step in case 7.....	84
Figure 5.41 von Mises stress in the vertical bolts in case 7.....	84

List of tables

Table 1 Mechanical Properties for glulam GL28h.....	42
Table 2 Distances between the bolts and the edges.....	42
Table 3 Pretension calculated from given torque.....	47
Table 4 Stress values taken from experimental test.....	49
Table 5 Case studies	50
Table 6 Moment rotation at different load steps for case 1.....	55
Table 7 von Mises stress for top angle at different coordinates for case 1.....	56
Table 8 Moment rotation at different load steps for case 2.....	59
Table 9 von Mises stress for top angle at different coordinates for case 2.....	60
Table 10 von Mises comparison between case 1 and case 2.....	63
Table 11 Moment rotation at different load steps for case 3.....	63
Table 12 von Mises stress for top angle at different coordinates for case 3.....	64
Table 13 von Mises comparison between case 1 and case 3.....	67
Table 14 Moment rotation at different load steps for case 4.....	68
Table 15 von Mises stress for top angle at different coordinates for case 4.....	68
Table 16 von Mises comparison between case 1 and case 4.....	71
Table 17 Moment rotation at different load steps for case 5	72
Table 18 von Mises stress for top angle at different coordinates for case 5.....	73
Table 19 von Mises comparison between case 1 and case 5.....	76
Table 20 Moment rotation at different load steps for case 6.....	76
Table 21 von Mises stress for top angle at different coordinates for case 6.....	77
Table 22 von Mises comparison between case 1 and case 6.....	80
Table 23 Moment rotation at different load steps for case 7.....	81
Table 24 von Mises stress for top angle at different coordinates for case 7.....	82
Table 25 Initial stiffness of connections.....	85
Table 26 von Mises comparison between case 1 and the combined case.....	90

Symbols and Terminology

K_h	Size-effect factor for glulam strength
h	Height of the beam.
R_m	Ultimate tensile stress of the steel
$R_{p0,2}$	Upper yield stress of the steel (0.2%)
$R_{p0,01}$	Lower yield stress of the steel (0.01%)
$S_{j,ini}$	Initial rotational stiffness
S_j	Rotational stiffness
K_b	Mean value of I_b/L_b for every beam in the top story
K_c	Mean value of I_c/L_c for all columns in that story
I_b	Beam's second moment of inertia
I_c	Column's second moment of inertia
L_b	Span of the beam
EI_b	Bending stiffness of the beam
M_{Rd}	Design moment resistance of the connection
$M_{pl,Rd}$	Design strength of the beam .
K_i	Stiffness coefficient for every component
Z	Lever arm of the beam
μ	Stiffness ratio
M_a	Applied torque
d	Bolt diameter
F_p	Preloaded in the bolt
k	Coefficient of friction between mating surface
S	Local connection force
K	Stiffness matrix for the element
v	Displacement matrix for the element
R	Global connection force
K	Global stiffness matrix

R	Connection displacement
U	Global vector of unknown displacement
F	Applied global load vector
L	Lower triangular matrix
u	Upper triangular matrix
F ^a	Applied loads
K _i ^T	Jacobian matrix
F _i ^{nr}	Vector of restoring loads corresponding to the element internal loads.
d _b	Depth of the glulam beam
Δ _e	Elastic deformation
Ø	Rotation of the connection
l _e	Effective length of the angle
M	Moment in connection
P _e	Reaction force
K _{cf}	Bearing stiffness of the column face
t _c	Thickness of the wall of the column
ν	Poisson's ratio for steel
b _c	Width of the column face
K _t	Stiffness in the horizontal leg of the angle
p	Width of the angles
t _f	Thickness of the angle
a [`]	Distance from the inner bolt hole to the vertical leg of the angles
K _{sb}	Stiffness of the bolt
A _b	Area of the bolt
L _b	Length of the bolt subjected to axial tension
K _{TOP}	Stiffness of the top angle
K _{BOTTOM}	Stiffness of the seat angle

1 Introduction

1.1 Background

Over the years hybrid connections between steel and concrete have been studied and widely investigated. The study however on the hybrid connections between timber and steel is less common, even though these types of hybrid connections have proven to offer some advantages over the more commonly used steel to concrete connections.

In Norway the access to timber is very high, but unfortunately the behavior of different connection types using steel and timber has not been investigated extensively. Therefore, a contribution of this thesis is to expand the current knowledge by analyzing this type of hybrid connection. Different types of steel to timber connections exist, but the focus in this thesis is on one specific interesting connection. The chosen connection is between a tubular steel column and a glulam beam connected together with top and seat angles by the means of bolts. This connection has not been extensively studied in the past and the optimization of it is something of which little to no studies have been done. Therefore, the connection is examined while several different parameters that affect its behavior are modified.

1.2 Area of focus

The area of focus in this thesis is the connection between a tubular steel column and a glulam beam. The angle brackets, bolts, and the glulam beam itself are under the scope, but a closer analysis of the top angle has been made. After extensive research, the main topic of this thesis can be written as follow: *”Translate the results of several analyses where the change of different parameters is examined within a connection between a tubular steel column and a glulam beam”*. Several important subtopics need to be investigated:

How does changes in the angles affect the behavior of the connection?

How does changes in bolts affect the behavior of the connection?

What are the effects of introducing stiffeners to the connection?

1.3 Limitations

Modelling a connection between steel and timber with top and seat angles that are connected by the means of bolts, is a complex procedure. Many parameters need to be correct in order for the analyses to be more accurate, or at least to get fairly accurate results from the FE-software. Because of the time frame and the complexities related to a detailed analysis of a hybrid connection of such a type, the following limitations and assumptions have been implemented:

- The shear force in the connection has not been taken into consideration
- Wood crushing has been observed, but it was not the main scope of this thesis
- Plastic behavior was also outside the scope of this thesis
- The behavior of the column and the blind bolts have not been taken into consideration

This is due to the fact that the connection between the steel angles and the glulam beam is of primary interest.

2 Theory

2.1 Glulam

Introduction

Timber is one of the oldest and most sustainable construction materials which has a high number of excellent mechanical properties. The strength to weight ratio is very high. Timber is an environmental friendly material and it can easily be shaped and connected. In addition, timber has good insulating properties against heat and sound. Nevertheless, it also has some weak mechanical properties if it is not properly treated. Since timber is an anisotropic material, the strength and stiffness perpendicular to the grain is much lower than the one that is parallel to the fiber direction. The swelling and shrinking perpendicular to grain, at varying moisture contents, makes timber vulnerable to cracking. The mechanical properties of timber depends on the microstructure of the wood. [12]

Structural glued-laminated timber (glulam) is one of the competitive products used in constructions today. Glulam is manufactured by gluing timber boards parallel to the longitudinal axis together and end-jointing individual wood pieces to the desired shape. The thickness of laminates ranges from 19-50 mm and has a length between 1.5-5m. For straight or slightly curved members, timber boards with thickness 33mm or above are used as laminates. Due to small variability in strength, glulam has greater strength and stiffness properties when compared with structural timber. Although almost any type of wood species can be used for glulam, softwoods are more preferable due to adhesive properties. Figure 2.1 shows the production process of glue-laminated timber. [13]

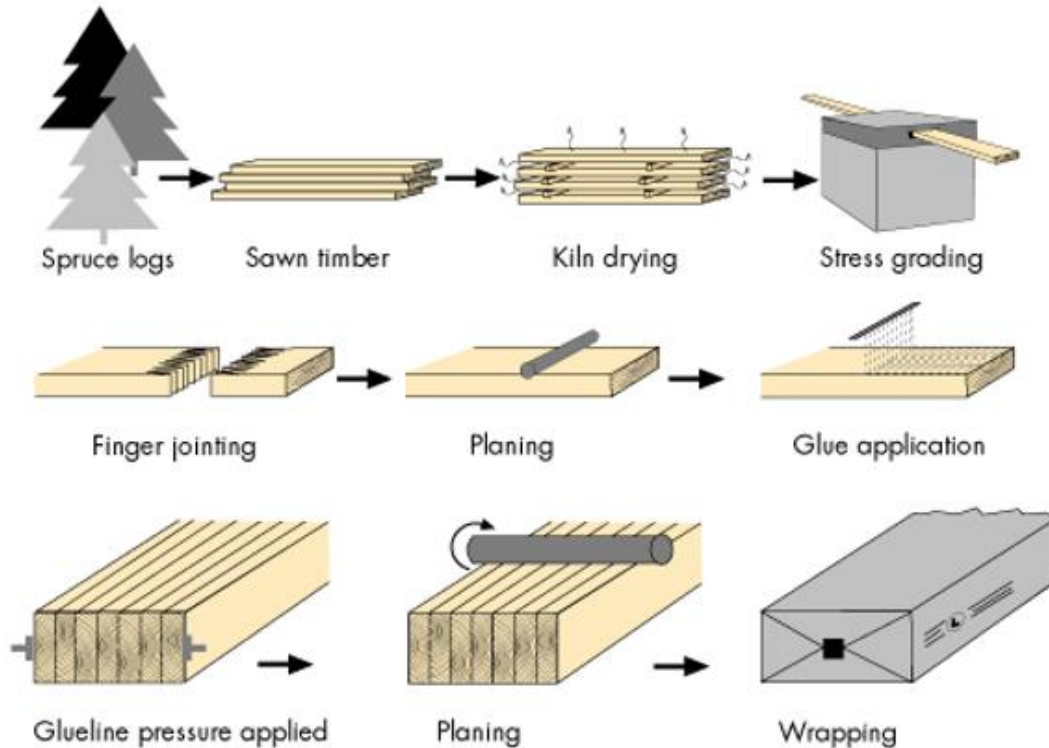


Figure 2.1. Glued-laminated timber manufacturing process

Strength and stiffness

The strength of glulam depends on the angle between the load and grain, the moisture content and the duration of loading. In production process the factors that affect the strength of the structural timber like knots are either removed or more uniformly distributed in the finished production. This makes the glulam stronger than the normal structural timber. The comparison of the frequency in the ultimate strength for glulam and structural timber are presented in figure 2.2. The strength of structural timber depends on the weakest cross-section. If there is a knot or a crack in the board's cross section, the strength is reduced considerably. Since there are several laminates which are glued together, the risk of having a weak section in glulam is lower. Due to these differences the material factor for glulam in designing is 1.25, which is lower than 1.3 which is the material factor for structural timber.

Laboratory results indicate that glulam beams experience a very brittle failure due to the existence of knot or finger joints when the beam is subjected to tension. The characteristic tensile strength parallel to the grain relates to members with a minimum width of 600mm and a minimum thickness of 150mm. The risk of failure increases as the volume of the beam increases. Therefore the

strength of thicker beams tends to be lower than that of thinner beams. According to Eurocode-5, the size-effect factor (K_h) that increases in bending and tensile strength for beams with height less than 600mm is given by the following factor: [13]

$$k_h = \left(\frac{600}{h}\right)^{0,1} \leq 1.1 \quad (h \text{ in mm}) \tag{2.1}$$

Where

K_h is the size-effect factor

h is the height of the beam.

Since glulam is strong compared to its self-weight, it can be used for wide spans with minimal supports.

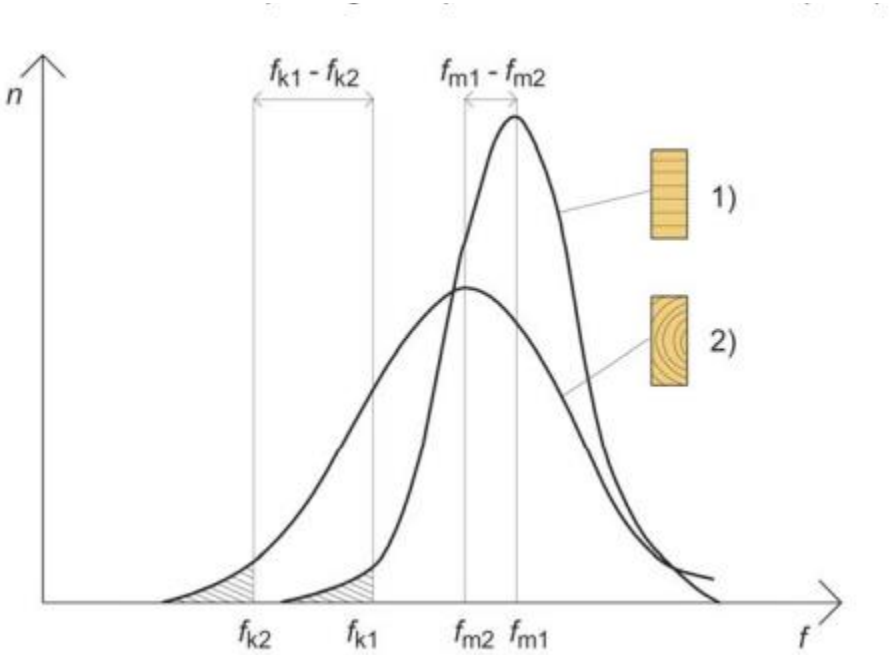


Figure 2.2. Frequency distribution of the ultimate strength of glulam and structural timber

2.2 Steel

Steel is one of the main construction material preferred and it has a lot of advantages over other structural materials. Steel has elastic and plastic mechanical deformation behavior. In elastic deformation the material returns to its original position after the applied load is removed. This obeys Hooke's law where the relationship between stress and strain is linear. In plastic deformation, material deforms permanently even after the applied force is removed from the material. The mechanical properties of steel are determined in laboratories by using standardized test specimens and following strict procedures. The following diagram shows the stress-strain curve of steel in the different stages of its deformation. [9]

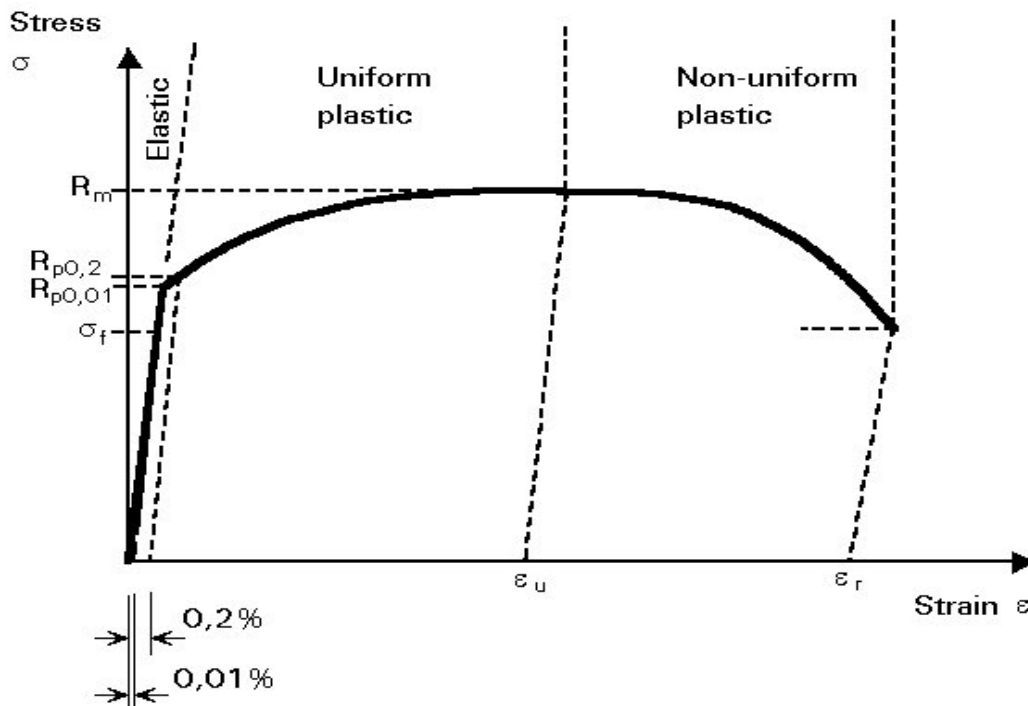


Figure 2.3. Stress-strain relationship

Where

R_m is the ultimate tensile stress

$R_{p0,2}$ is the upper yield stress

$R_{p0,01}$ is the lower yield stress

In structural engineering, the stress-strain diagram can be divided into three regions. In the first region the stress gives an elastic deformation where the strain (ϵ) is less than 0.2%. If the applied stress is beyond the yielding stress, the material has uniform plastic deformation. In this region the material deforms drastically for infinitesimal increments of stress. This happens due to the movement of dislocation through a crystal lattice. In the third region of deformation the material becomes narrower and results in local necking of the material. The ratio between the reduced area and the original area is defined by the material's ductility. [9]

2.3 Connections

The main role of the connections are to transfer loads from one structural element to another. When connections are designed, engineers should always consider the practical possibility of performing it in the construction site. There are different types of connections in structural engineering like column splices, column base, simple beam-to-column connection, moment resisting beam-to-column connection etc. Since the mechanical properties of the connections highly influences the strength, stiffness and stability of the structure, each connection must be properly designed. The main classification of connections are based on: [10]

- When a change of direction occurs, e.g. beam-to-column connection
- When considering manageable size of steelwork for transportation and erection
- When a change of component occurs including connection of the steelwork to other parts of the building

The figure below illustrates the different classes of connection

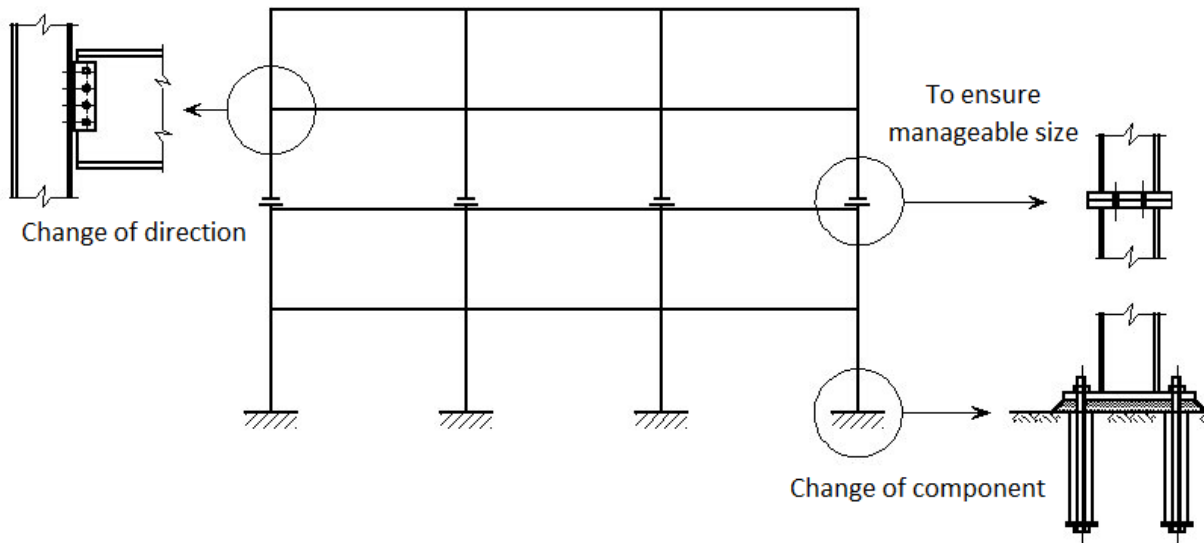


Figure 2.4. Different classes of connections

2.3.1 The moment-rotation relationship

Every deformable object that are subjected to force actions, can be represented by a force action-deformation curve. By using these curves an overall information about the behavior of the object can be understood. This is very relevant in structural engineering, as the entire behavior of a structure can be understood from several action-deformation relationships, for different members in the structure. The action-deformation curves can be obtained at several levels. Looking at a material level the action-deformation relationship can be plotted as a stress-strain curve. While at a cross-section level, the relationship can be plotted as a moment-curvature curve. Furthermore when looking at a member level the action-deformation relationship can be analyzed with a moment-rotation curve and at a structural level this relationship can be plotted as a load-deflection curve. The curvature of a cross section from any given moment can be translated into other deformations such as rotations, strains and deflections at any point in a member. [17] When defining whether a connection is rigid, semi rigid or pinned, the moment-rotation relationship approach can be used. From this curve the amount of moment needed in the connection in order to rotate the connection with a certain amount is given. In a rigid connection for instance a high moment is needed to rotate the connection even by a small amount, compared to a semi-rigid connection where a lower moment is needed to rotate the connection by the same amount. This curve offers important information about the rotational stiffness of a connection. In order to

classify the stiffness of a connection, the initial rotational stiffness has to be calculated. Since the relationship between moment and rotation is nonlinear, it is necessary to idealize the $M-\phi$ curves to be bilinear or trilinear for design purpose. [10]

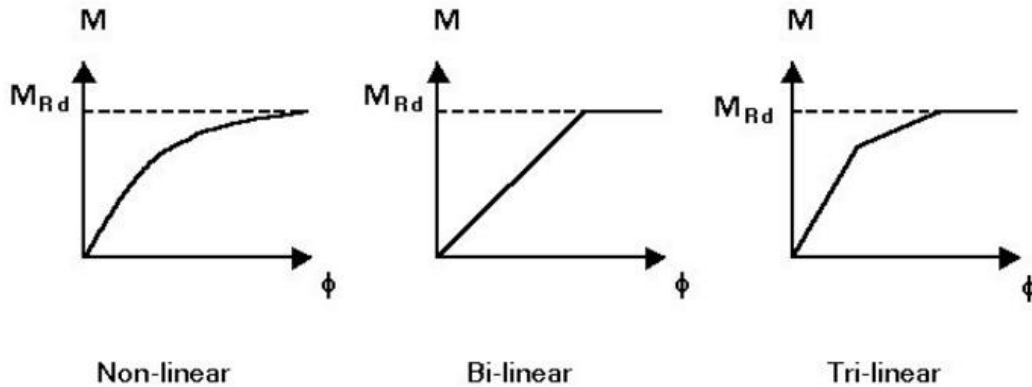


Figure 2.5 Possible idealizations for $M-\phi$ curves

2.3.2 Component based method

The general analytical procedure which is used for determining the resistance and stiffness properties of a connection is the Component based method. This method considers any connection as a set of individual basic components. Each of these components possesses its own strength and stiffness which together makes the stiffness of the total connection. The application of the component based method requires the following steps.

1. Identification of the active components
2. Evaluation of the stiffness and resistance characteristics of each component
3. Assembly of all the components
4. Evaluation of the stiffness and resistance characteristics of the whole connection.

The basic components used in Eurocode 1993-1-8, are identified in Table 6.1 and it is applicable to similar components in other connection configurations. For other connection configurations, the appropriate assumptions can be used for the determination of the distribution of internal forces.

The components which cannot be found in the Eurocode 3 may be determined either from experimental tests or analytical and numerical methods. [6]

The rotational stiffness of a connection should be determined from the flexibilities of its basic components. The stiffness coefficient for every component are in elastic zone and is given by K_i and can be found in section 6.3.2 in Eurocode 1993-1-8. The initial rotational stiffness $S_{j,ini}$ of a beam-to-column connection for a moment $M_{j,Ed}$ less than the design moment resistance $M_{j,Rd}$ of the connection, may be obtained with the following equation

$$S_j = \frac{Ez^2}{\mu \sum_i \frac{1}{k_i}} \quad (2.3)$$

Where:

K_j is the stiffness coefficient for basic connection component j ;

Z is the lever arm;

μ is the stiffness ratio $S_{j, ini}/S_j$

The stiffness ratio $\mu = 1$ if $M_{j, Ed} \leq \frac{2}{3} M_{j,Rd}$: and

$$\mu = (1.5 M_{j,Ed} / M_{j,Rd})^\psi \quad \text{if } \frac{2}{3} M_{j,Rd} < M_{j,Ed} \leq M_{j,Rd}$$

The coefficient ψ is obtained from Eurocode 1993-1-8 Table 6.8.

According to Eurocode 3 (Design of steel structures) connections may be classified either by their strength, or stiffness. [6]

2.3.3 Classification of connections by stiffness.

The connection stiffness can be defined as the slope of the moment rotation curve. When connections are classified according to their stiffness's, the initial rotational stiffness $S_{j,ini}$ of the connection needs to be evaluated with the criterion given in Eurocode-3 section 5.2.2.5. This classification of connections is relevant for elastic analysis of frames. The stiffness of the connection affects both the deflection of the beam and the level of the designed load it can transfer. Furthermore the connection classification are described in the following:

- In pinned connections the rotational stiffness is very low and this results in a high deflection of the beam. There is no moment transfer, but vertical or shear forces can be transferred.
- In rigid connections the rotational stiffness is very high and the connections moment resistance is approximately the same as the beam. This results in insignificant deformation of the structure.
- The behavior of connections that ends up between the criteria for being pinned and rigid, are defined as a semi rigid connection. The classification of connections are seen in figure 2.6

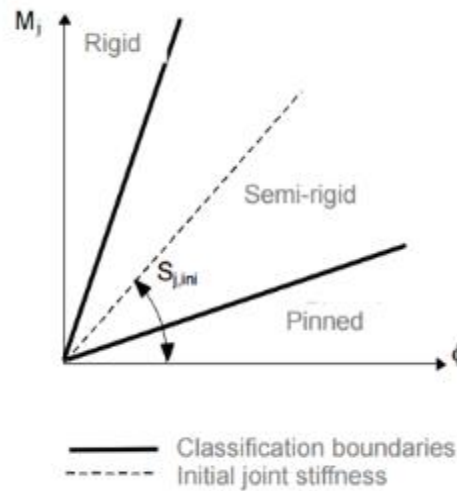


Figure 2.6. Connection stiffness in $M-\phi$ curve

As seen in chapter 2.3.3, the criteria that needs to be fulfilled in order to classify a connection as rigid is given by

$$S_{j,ini} \geq K_b * EI_b / L_b \quad (2.2)$$

Where

- K_b is the mean value of I_b/L_b for every beam in the top story
- $K_b = 8$ for frames where the horizontal displacement is reduced by 80% by the bracing system
- $K_b = 25$ for all other frames, with the condition that $K_b / K_c \geq 0.1$

- K_c is the mean value of I_c/L_c for all columns in that story
- I_b is the beam's second moment of inertia
- I_c is the column's second moment of inertia
- L_b is the span of the beam

To determine if the connection is rigid, semi-rigid or pinned, the following classifications are given:

- Nominally pinned $S_{j,ini} \leq 0.5 EI_b/L_b$
- Semi-rigid $0.5 EI_b/L_b < S_j < 8 EI_b/L_b$ (braced frame) or $0.5 EI_b/L_b < S_j < 25 EI_b/L_b$ (unbraced frame)
- Rigid $S_{j,ini} \geq 8 EI_b/L_b$ (braced frame) or $S_{j,ini} \geq 25 EI_b/L_b$ (unbraced frame)

Where

$S_{j,ini}$ is the initial rotational stiffness of the connection

EI_b is the bending stiffness of the beam

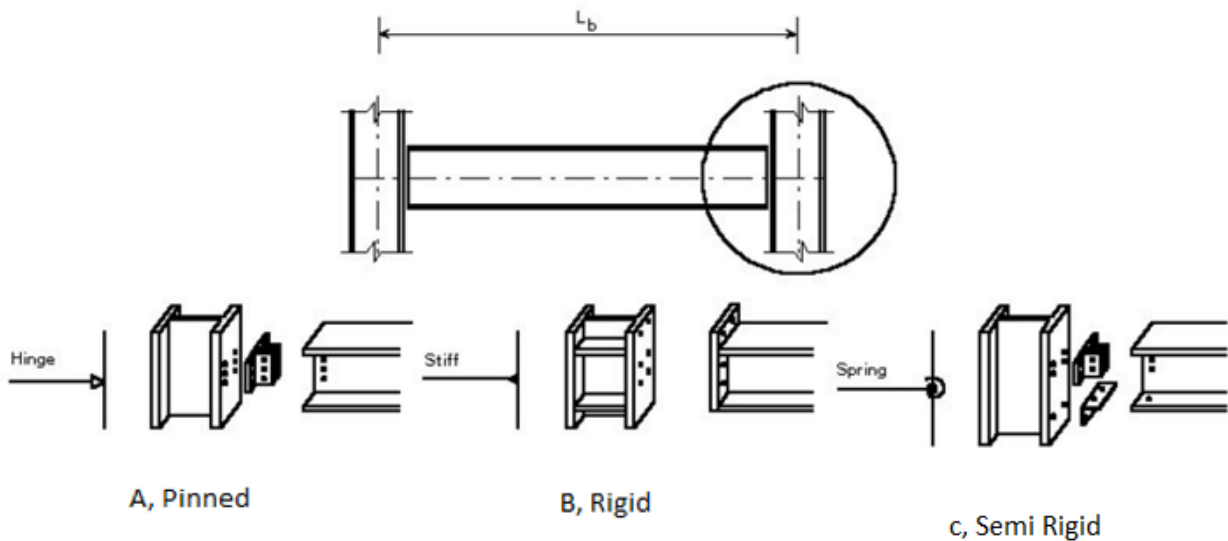


Figure 2.7 Schematization of rotational stiffness

2.3.4 Classification of connections by strength.

If connections are to be classified according to their strength, a static analysis must be done to determine its designed load bearing capacity. We compare the connection's designed moment resistance $M_{j,Rd}$ with the designed moment resistance of the members that are connected to it. This classification of connection is relevant for plastic analysis of frames. The connection can be classified as full-strength, nominally pinned or partial strength. If the connection designed moment resistance is not greater than 0.25 times the design moment resistance required for full-strength connection, it is classified as nominally pinned even though it has sufficient rotation capacity. In a full-strength connection the designed resistance $M_{j,Rd}$ should be greater than that of the connected members. According to Eurocode 5.2.3.3 the criteria for classifying strength is given. The following figure demonstrates the three possible classification of beam-to-column connection. [6]

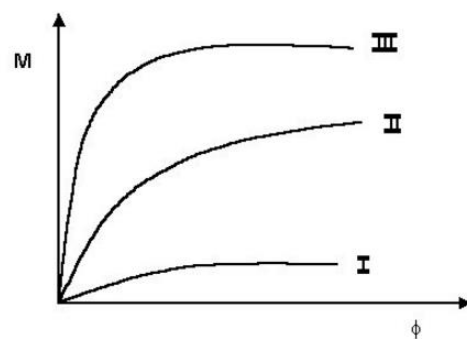
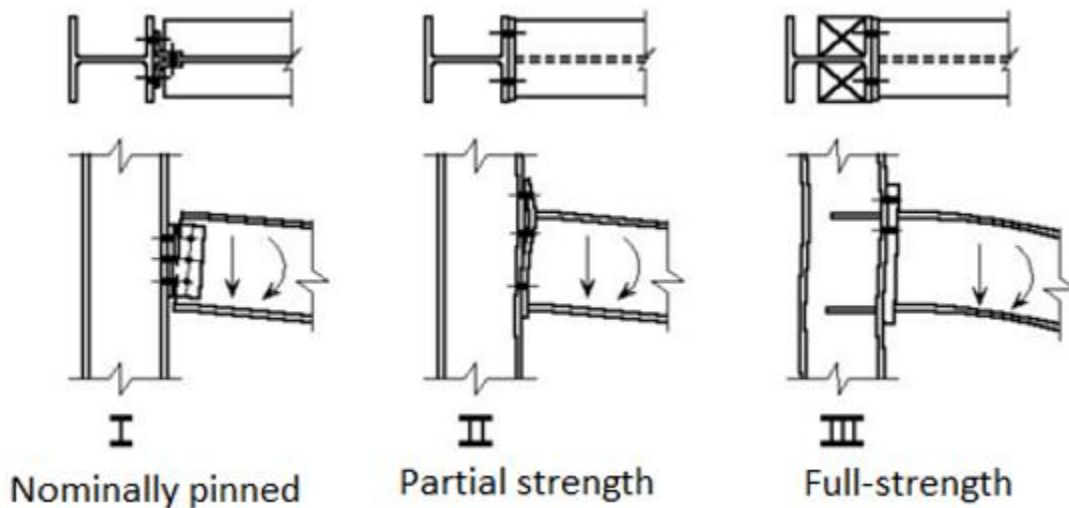


Figure 2.8 Moment rotation curve according to strength

- I The rotational stiffness and the moment resistance are very small and only the shear and axial forces are transferred in the connection. This connection can be categorized as a pinned or hinged connection.
- II The connection obtains a certain amount of moment, but less than the full moment resistance of the beam. Beside this the connection has relatively higher rotational stiffness than the first category. These connections can be semi-rigid and or partial strength. It is however also possible to have connections with full-strength and semi-rigid or vice versa.
- III The moment resistance of the connection is almost the same as the beams and the rotational stiffness is very high. The beam's end reaction and its end moment is transferred to the column. The connection is categorized as a full-strength or rigid.

For strength, the following classes can be distinguished:

- Nominally pinned $M_{RD} \leq 0.25M_{PL,RD}$
- Partial-strength $0.25M_{PL,RD} < M_{RD} < M_{PL,RD}$
- Full-strength $M_{RD} \geq M_{PL,RD}$
- Full-strength $M_{RD} \geq 1.2M_{PL,Rd}$ (without checking rotational capacity)

Where

M_{Rd} is the design moment resistance of the connection

$M_{pl,Rd}$ is the design strength of the beam.

2.4 Bolted connection

Bolted connections are frequently used when connecting members together. To determine the resistance of bolted connection, the resistance of each individual fastener and the connected parts must be evaluated. Bolts can be loaded in tension, shear or in combination of both. To design the connection, a linear-elastic analysis is mostly used. If the bolts are not preloaded they are called bearing type bolts. In splice connections the shearing load is transferred as bearing stress from one plate to another through the bolt's shank.

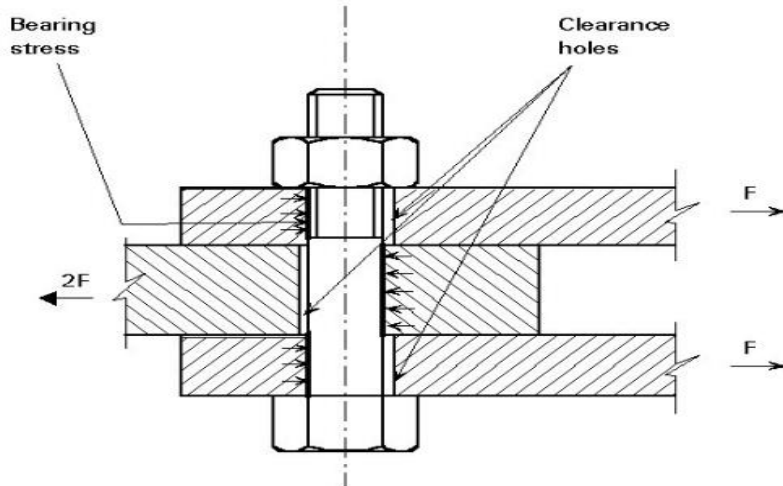


Figure 2.9 Load transfer in non-preloaded bolt

If the bolt is preloaded, the clamping pressure occurs between the connected parts and the load is transferred by frictional forces. This happens due to the exerted compressive stress on the connected plates.

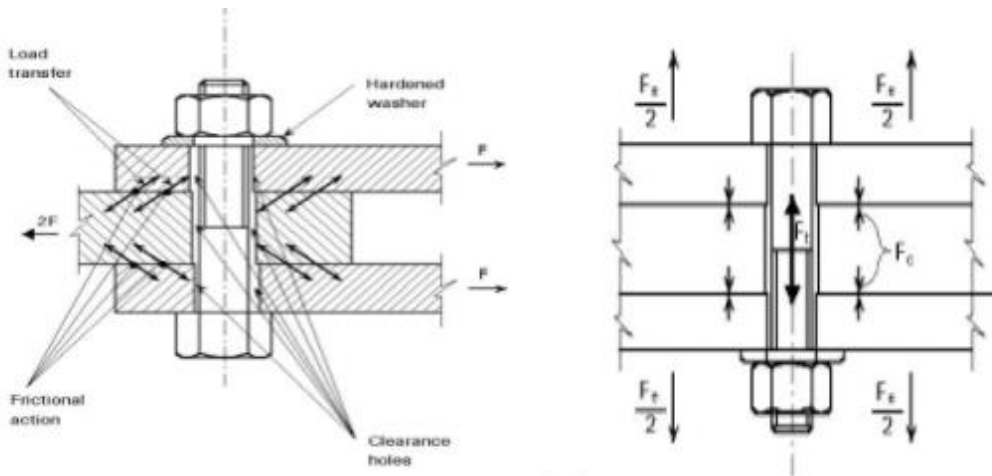


Figure 2.10. Load transfer in preloaded bolt

To obtain an adequate clamping force with a reasonable bolts size, high tensile steel bolts (usually grade 10.9) should be used. To tighten bolts a torque wrench is used. However in FE-software a preload can be defined. To calculate the amount of preload from a given amount of torque, the following formula is given: [10]

$$M_a = k \cdot d \cdot F_p \quad (2.3)$$

Where

- M_a is the applied torque (Nmm)

- D is the bolt diameter(mm)
- F_p is the preloaded in the bolt(N)
- k is the coefficient of friction between mating surfaces
-

2.5 Finite element method

Finite element method is a numerical method which can be used to predict the behavior of very complex constructions. This can be done by hand calculations, but even for simple structures this becomes an overwhelming job. Therefore, structural engineers use FEM software such as Abaqus, Ansys, SAP200 etc for these purposes. What these software do is to divide a complex construction into small elements that are called finite elements. For each and every one of these finite elements a mathematical model is calculated and a prediction of its behavior is made. Then all of the divided elements are added up to predict how the structure as a whole behaves. For each element the relation between displacement and stiffness are: [7]

$$S = K*v \quad (2.4)$$

Where

- S is the local connection force
- K is the Stiffness matrix for the element
- v is the displacement matrix for the element

When the behavior of all elements are calculated they are transformed into the global stiffness matrix and the relation between stiffness and displacement are given as:

$$R = K*r \quad (2.5)$$

Where

- R is the global connection force
- K is the global stiffness matrix
- r is the connection displacement

In order to perform an accurate analysis of a structure, it is very important to have an appropriate model with the correct inputs. We need the correct dimensions, external forces, boundary conditions and contact surfaces, target surfaces (where contact is present). Simplifications however can be done as long as it is conservative.

3 Finite Element Model

3.1 Ansys Mechanical APDL

The connection in this thesis has been modelled in the finite element software Ansys Mechanical APDL(Ansys Parametric Design Language). Ansys is a complex finite element program which is used for, static, modal, magnetic, fluid and transient problems. In this thesis, static analyses will be used. This program is often updated and refined and newer versions is developed. However the available version used is in this thesis is 17.2, which is one of the latest. The academic version of Ansys that are available on campus has a restriction of number of nodes, at which the maximum is 256 000. This means that the meshing is fairly restricted depending on the size of the model. The interface of Ansys is a complete platform in which everything is done. From building the geometry, assigning material properties, meshing, defining contact and target elements (if needed) and etc, to analysing the results. Ansys is logically divided into multiple modules that takes the user true every step of analysing a structure. As an alternative to using the graphical method, a script can be made and pasted into a text editor in the interface. Ansys has its own programming language which is huge. Therefore a help-library are available as support



Figure 3.1 Modules in Ansys

When building a model and defining geometry, element type, material properties etc, the preprocessor module is chosen. After these are defined and the model is constructed, the meshing of it can be done. This divides the structure into smaller elements. The next step is to enter the solution module and start the analysis of the structure. Here the loads, constraint, analyses type etc are defined. By clicking on the solve tab Ansys starts the analysis. Finally when the solution is done these can be viewed in the general postprocessor

3.2 Model parameters

Modelling connections is not an easy procedure. It consists of multiple parameters and these needs to be correct in order for the analysis to give accurate results. If the parameters are inaccurate, the analysis can be terminated because of nonconvergence. Depending on the type of analysis done, different approaches are needed. For connections, a set of contact and target surfaces needs to be defined with application of the correct properties which depends on the type of contact problem that are investigated. In the following paragraphs, the most important parameters used in contact problems are explained.

3.2.1 Element type

Ansys has a wide element library to choose element types from. Depending on the material and the behavior of the structure it simulates, the elements needs to be chosen carefully. When the mesh is generated, elements generate different amount of nodes depending on the element type used. Choosing one that produces a small amount of nodes can affect the accuracy of the results, while selecting an element that produces many nodes may be expensive in CPU-time. Also element types that produces a high amount of nodes may cause memory issues due to the amount of equations that needs to be calculated by the program. SOLID185, SOLID186 and SOLID187 are often used when modeling complex geometries, and it's a good choice when analyzing a connection.

Solid185 is a 3-dimensional, 8-noded structural solid element. This element has three degrees of freedom which is the nodal translations in x, y and z direction. The eight nodes have all the aforementioned degrees of freedom. [2]

SOLID185 Homogeneous Structural Solid Geometry

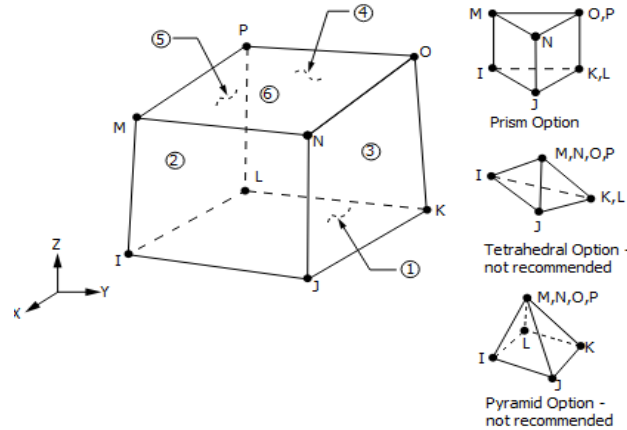


Figure 3.2 Element geometry SOLID185

Solid185 can be meshed like a prism, tetrahedral, pyramid or as a block. These options may lead to less accurate results if used together with a poor mesh. [2]

Solid186 is also a 3-dimensional structural solid element, but it generates 20 nodes when it is meshed. This produces a lot of elements which causes the analysis time to increase. The element has three degrees of freedom and they are in the x, y and z translational direction. The element exhibits a quadratic displacement behavior and is well suited for irregular meshes. [2]

SOLID186 Homogeneous Structural Solid Geometry

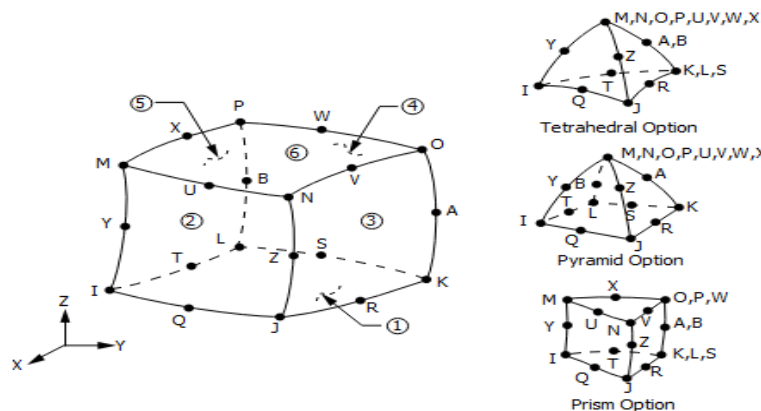


Figure 3.3 Element geometry SOLID186

As seen in figure 3.3 the element can form the same geometries as solid185, but here the element generates midside nodes which then again creates more elements. In solid185 the prism and pyramid option is not recommended whereas in solid186 no recommendation is mentioned, or preferred.

Solid 187 is 10 node, 3-dimensional solid structural element. This element has a quadratic displacement behavior and is well suited for irregular meshes as SOLID186. This element has three degrees of freedom in the translational x, y and z-direction. [2]

SOLID187-3-D 10-Node Tetrahedral Structural Solid

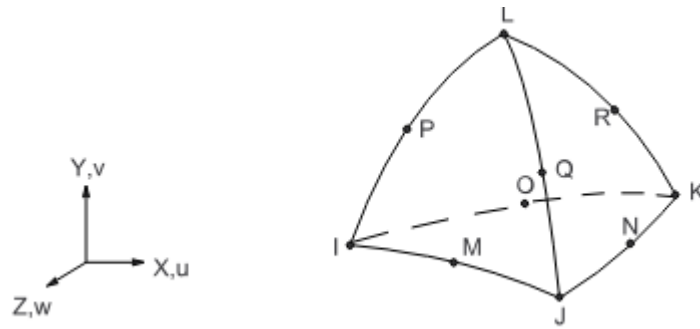


Figure 3.4 Element geometry SOLID187

3.2.2 Contact

In many FE software a determination of a connection between structural elements such as, rigid, semi-rigid, or pinned are often predefined. If however a specific detailed connection is to be analyzed there are multiple parameters that needs to be taken into account. When modeling contact between two connected members the problem becomes highly nonlinear, and the need for significant computer resources to analyze and solve the problem are crucial. It is important to be very accurate when generating the numerical model, and really understand the physics of the problem. When dealing with contact problems there a two general classes to choose from. This is flexible to flexible, or rigid to flexible contact

In rigid to flexible contact problems one of the contacting surfaces is to be considered as having much higher stiffness than the deformable body it contacts. A general rule is that whenever a hard material makes contact with a soft material, the contact problem is regarded as a rigid to flexible contact problem. Flexible to flexible contact problems occurs when two materials with similar stiffness come into contact. This is the more common type of the two mentioned. [2]

3.2.2.1 Contact elements

Ansys has multiple element types which can be used for contact problems. Also here there are some available only for 2-dimensional analyses while others are for 3-dimensional analyses. Specifically two contact elements are of interest, namely CONTA173 and CONTA174.

CONTA173 is a 3-dimensional, 4 node surface to surface contact element. It is used between 3-dimensional target surfaces and deformable surfaces which is defined by the CONTA173. This element can be used both in pairs-based and general contact definitions. For each definitions TARGET 170 can be used as the target surface or it can be defined by the CONTA173 itself. CONTA173 has no midside nodes and therefore it needs to be located on a surface who does not produce midside nodes. [3]

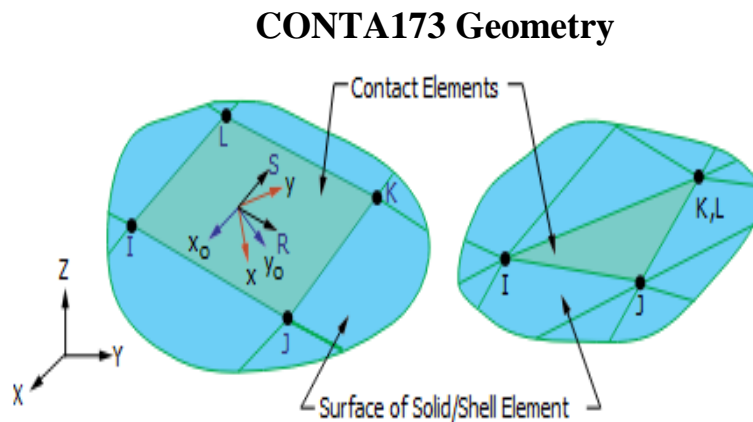


Figure 3.5 CONTA173

CONTA174 is 3-dimensional, 8 noded surface to surface contact element. This element is used to represent sliding and contact between a 3-dimensional target and a contact surface. CONTA174 can be used for both general, and pair-based contact. The main difference between CONTA173

and CONTA174 is that the latter has midside nodes. Meshing an element with midside node produces more elements within the structure, the more elements the software produces the more accurate the result become. However if an element with midside nodes are used and divided into many elements by creating a very finite mesh, the analyses of the problem can be very tricky and time consuming. CONTA174 can be used with solid elements weather it produces midside nodes, or not. However if a contact element with midside nodes is used on an element type that has no midside nodes the software will produce warnings and this may have an effect in the accuracy of the finite element model. [3]

CONTA174 Geometry

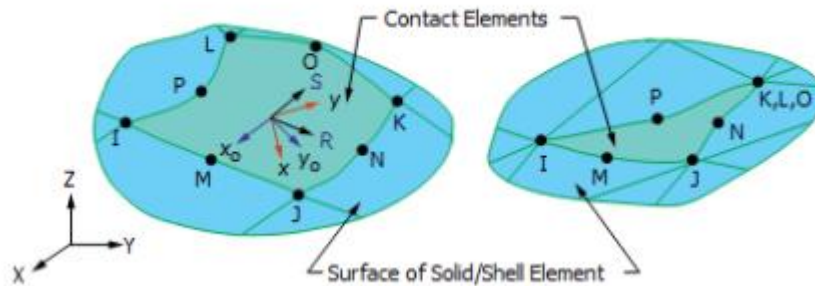


Figure 3.6 CONTA174

3.2.2.2 Target elements

As mentioned earlier every contact analysis needs both a contact and a target surface. For use in static analyses two types of target element are available, TARGET169 and TARGET170.

TARGET169 is as a 2-dimensional target surface which is combined either with CONTA171, or CONTA172. The contact surface overlays the deformable body, while the target surface is the surface the deformable body has as a target. [3]

TARGET169 Geometry

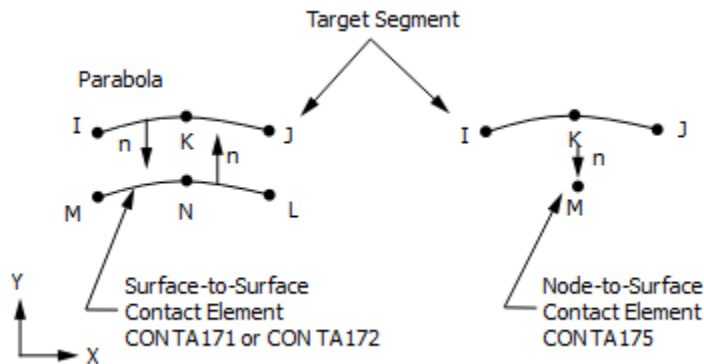


Figure 3.7 Functionality of the target elements

TARGET170 represent a 3-dimensional target surface which can be combined with contact elements CON TA173 or CON TA174. As for TARGET169 the contact elements used with TARGET170 overlays the deformable body, whilst the target elements are the target for the deformable body. The main difference between the two target elements is that TARGET169 is a 2-dimensional element while TARGET170 is a 3-dimensional element. [3]

TARGET170 Geometry

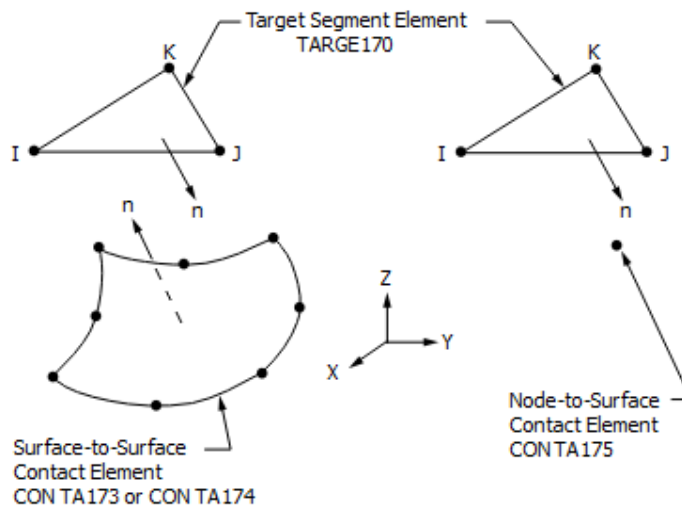


Figure 3.8 Principals of the target elements

3.2.2.3 Symmetric and asymmetric pairs

Asymmetric pair

When modelling an asymmetric pair, one of the surfaces is defined as a target and the other as contact. This is also known as a one-pass contact. To decide which is contact and which is target a general advice is that if one of the bodies moves towards the other, the moving body is the contact surface and the other is the target surface. However it is not always clear which is the moving body and which is the target body, sometimes they both move and sometimes some parts of the body move while the rest of the body stands still. Therefore a set of guidelines is provided by Ansys to ease the choice. These guidelines consist of the following. [2]

- If a convex surface makes contact with a concave surface, the latter should be the target surface
- If one of the surfaces has a finer mesh than the other, the surface with the coarser mesh should be considered as the target
- If one of the surfaces is stiffer than the other, the less stiffer surface should be considered as a target
- If one of the surfaces are markedly larger than the other surface, the larger surface should be considered as the target surface
- If a higher order elements underlie one of the surfaces, and a low-order element underlie the other surface the low-order elements should be considered as the target surface

If there still is some confusion regarding the contact and target surface, the more safety solution is to use symmetrical contact pairs.

Symmetric pair

When modeling symmetric contact pairs, Ansys assigns both a contact and a target surface at both bodies. This means that for every contact assembly, Ansys generates two set of contact pairs. This way Ansys automatically recognizes the contact and target surfaces even if some of the surfaces behaves different than the rest. Even though this type of contact is less efficient than the

asymmetrical contact, many analysis will need this to reduce the problem with penetration in the model due to wrong choice of contact and target surfaces when using asymmetric contact pairs. [2]

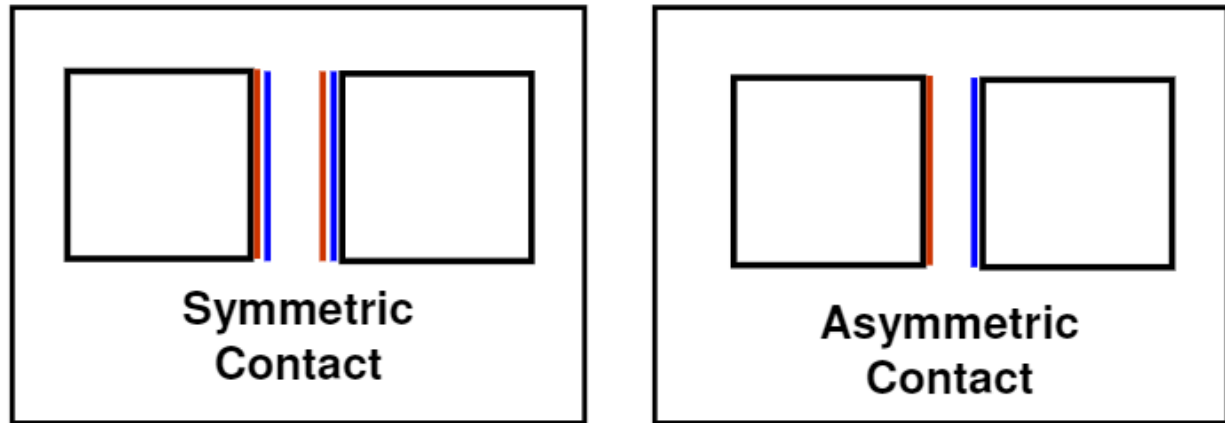


Figure 3.9 Schematic representation of symmetric and asymmetric contact

3.2.2.4 Contact analysis type

There are three different analyses types used when dealing with contact problems. These are: [2]

- Node to node contact
- General definition
- Pair-based definition

Node to node contact elements are used when the location of the contacts are known before modeling the structure. Typical for this type of contact problem is when there is small-sliding between the different contact surfaces.

The general and pair-based definition are suitable for solving both large and small sliding between the contacting surfaces. In general it is recommended if possible, to use either of these. When defining one of these two the following needs to be specified: [2]

- The definition of the surface for the contacting bodies
- The interaction and contact pairing for the surfaces that are in contact
- The behavior of the interface of the contacts

- The contact Properties
- Contact formularens

When using the general contact definition, general contact surfaces are defined. One of the contact elements (CONTA171 through CONTA174) overlays the surface of the deformable body while for the rigid bodies, TARGET 169 or TARGE170 are used. When this type of contact surface is generated, Ansys assigns a unique section ID for these, so each surface consist of contact or target elements with a unique section ID. When running the analysis Ansys search for contact interaction among all the assigned surfaces.

In the pair-based definition an interaction is defined between target surfaces and contact surfaces. Ansys assigns the same real constant number to the contact and target surface that are supposed to interact with each other. When the analysis is running it searches for contact interaction by using these real constants. [2]

3.2.2.5 Contact method

Surface to surface contact elements

This type of contact is available both when having rigid to flexible, and flexible to flexible contact problems. In this contact type, target and contact surfaces are defined to make a contact pair. TARGET169, CONTA171, CONTA172 are used in 2-dimensional problems, while TARGET170 with CONTA173 or CONTA174 are used in 3-dimensional problems. [2]

Node to surface contact

This type of connection uses CONTA175 elements to model either flexible to flexible, or rigid to flexible contact pairs between a given set of nodes and a surface. CONTA175 can be applied both for 2-dimensional and 3-dimensional problems, but does not support 3-dimensional contact surfaces with midside nodes. [2]

3.2.3 Contact properties

Managing contact pairs is a challenging procedure which requires accurate modeling with the correct parameters in order to achieve convergence. In the following, the most important parameters will be presented.

3.2.3.1. Contact surface behavior

There are several contact surface behavior types to choose between. Some support sliding along the surfaces, while others bond the surfaces together. The different types are as follows: [2]

- Standard contact
- Rough contact
- Bonded
- Bonded initially
- Bonded always
- No separation
- No separation always

The standard contact is a unilateral contact. This means that the normal pressure equals to zero if separation between the contact and target surfaces occurs. When using rough contact surfaces, no sliding will occur. Here the contact has an infinite coefficient of friction. The option of using bonded contact results in having the contact and the target surfaces bonded in all directions throughout the analysis from when the first contact between the surfaces occurs. When choosing bonded always the contact detection points which are initially inside the pinball region, or that once contact is established it always attaches to the target surface parallel to the normal direction to the contact surface. Here the contact is fully bonded. Choosing bonded initially, the contact detection points that initially are closed remains attached to the target surface while the contact detection points that initially are open remains this way throughout the analysis. If the option of no separation are used, the contact and target surface are tied throughout the analysis once contact are established. Sliding is however possible. The last option is to have no separation always, here the contact detection points that either are inside the pinball region or that once contact is established it always attaches to the target surface parallel to the normal direction of the contact surface. Here sliding is also possible. [2]

3.2.3.2 Contact detection

In the contact elements there are located contact integrations points interior to the element surface. The contact element itself cannot penetrate into the target surface, but the target surface on the other hand can penetrate into the contact surface. When creating contact pairs with surface to surface elements the default contact detection is Gauss integration points. This will normally produce much more accurate result than the nodal detection procedure, which used the nodes generated from the mesh as detection points. [2]

Contact Detection Located at Gauss Point

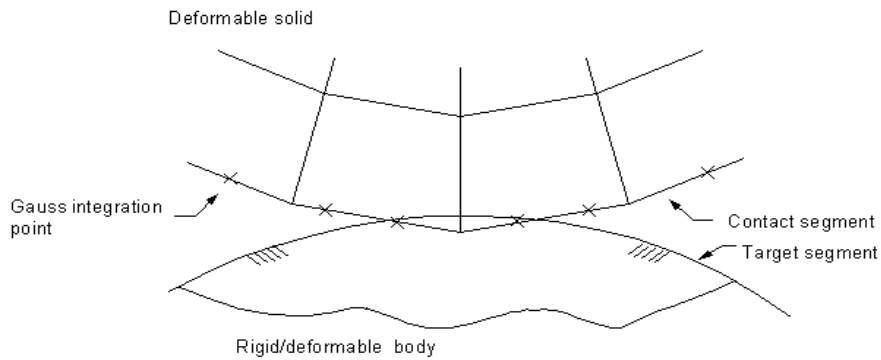


Figure 3.10 Contact detection at Gauss point

If the nodal detection is preferred, Ansys requires smoothing of surfaces to be done. Because this can take quite some time achieving, a recommendation is to only use this type of detection if the contact is to be done with corners, point or edge surfaces. [2]

Contact Detection Point Located at Nodal Point

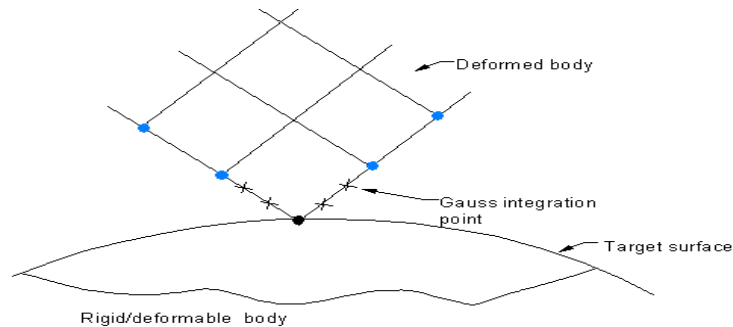


Figure 3.11 Contact detection at Nodal detection

3.2.3.3 Normal penalty stiffness

If the contact algorithm are either the Augmented Lagrange method or the penalty method, both normal and tangential contact stiffness are required. Slip can occur in the sticking contact between two surfaces depending on the tangential stiffness, and the amount of penetration by contact pairs can be adjusted with higher or lower normal contact stiffness. But there is a paradox here, because defining a high stiffness both for the tangential and the normal contact stiffness in order to minimize the slip and penetration, can result in a troubling global stiffness matrix and can create difficulties when the analysis tries to converge towards a solution. On the other hand using lower values for these stiffnesses can cause some penetration and slip, and may give inaccurate results. So the obvious method is to have high enough stiffness so that the slip and penetration is acceptably small, but still maintain a low enough stiffness so that convergence problems does not occur. As default a factor of 1.0 is used as a normal contact stiffness, but it can vary from 0.1-10 depending on the contact problem. The default tangential stiffness is the same as for the normal contact stiffness. [2]

3.2.3.4 Penetration tolerance

This feature is for use in the Augmented Lagrange method and the Lagrange multiplier method. This factor must be lower than 1. The default value is 0.1 and this is a tolerance factor that is applied parallel to the surface normal. The penetration factor is used to determine if the penetration compatibility is satisfied. In order for the contact compatibility to be acceptable it is measured with the allowable penetration which is defined by the penetration factor times the depth of the underlying elements. So even if the software manages to achieve the convergence criteria for all the forces and displacements, but the tolerance is beyond what is acceptable the analyses considers this as an unconverged solution. So it is very important for this factor to be correctly adjusted. [2]

3.2.3.5 Pinball region

When the integration points on the contact surfaces is in a near-field contact, this means that there is a small distance between these points and the corresponding target surface. This distance is

called the pinball region. When modeling in 2-dimensions, this region is a circle while modeling in 3-dimension this region is a sphere and they both are centered around the Gauss integration points. When Ansys analyses the structure and is searching for contact, the computational time depends on the size of the pinball region. For most contact problems the default value given by Ansys is almost always sufficient. [2]

3.2.3.6 Coulomb's friction

In Coulomb's friction model, before two surfaces starts to slide relative to each other the two contact surfaces can maintain some shear stresses in between them. When a contact is in this state this is referred to as a sticking contact. Another state of contact is when a contact is sliding. Coulomb's friction model defines a shear stress at which sliding begins as a fracture of the contact pressure on the surface. At the point when the shear stress is exceeded the phenomena of sliding occurs. [2]

3.2.4 Contact Algorithm

There are multiple algorithm alternatives to choose from when analyzing a contact problem. Below is an explanation of the different types and how they work.

3.2.4.1 The penalty method

In order for the contact to be established, the penalty method uses a spring stiffness to form the relationship between the contacting surfaces. This is also referred to as a contact stiffness. This method can optionally use the following parameters in its algorithm: [2]

- Normal penalty stress
- Tangent contact stiffness factor
- Allowable penetration
- Maximum sliding

3.2.4.2 The Augmented Lagrange method

This is the default method and is actually a series of iterative Penalty methods. In order for the penetration to be lower than the allowable tolerance, frictional stresses and pressure are augmented during equilibrium iterations. [2]

3.2.4.3 The pure Lagrange multiplier method

When a contact pair is closed, this method enforces zero penetration between the surfaces, and when sticking contact occurs the slip is also set to zero. This method does not use the normal penalty stiffness or the tangent contact stiffness factor during its analysis. The two following factors are implemented instead:

- Allowable penetration
- Maximum allowable tensile contact

The pure Lagrange multiplier method adds a contact traction as an additional degree of freedom and during the analysis it uses additional iterations to stabilize the contact conditions. [2]

3.2.4.4 Multipoint constraint algorithm

This method is best suited in connections where the contact type is either defined as bonded, or as a no separation contact.

An alternative algorithm is the combination of the Lagrange multiplier applied on the contact normal, and the Penalty method applied on the frictional plane. From this we get zero penetration combined with a small amount of slip for the sticking contact condition. For this method the following parameters needs to be specified. [2]

- Allowable penetration
- Tangent contact stiffness factor
- Maximum slip

3.3 Equation solver

When the modeling is done and the different parameters is chosen, the selection of the solver for the analysis can be chosen. Ansys has several methods for solving complex structural problems and the available solvers in this program are: Sparse direct solution, The Jacobi conjugate gradient solution, the incomplete Cholesky conjugate gradient solution, and the Preconditioned conjugated gradient solution. Some of these use a direct elimination process, while others uses an iterative procedure. [4]

The direct elimination method is a Gaussian process which solves the vector variable {u} in the following equations: [4]

$$[K] \{U\} = \{F\} \quad (3.1)$$

Where

[K] is the global stiffness matrix

{U} is the global vector of unknown displacement

{F} is the applied global load vector

When using direct elimination a factorization of the global stiffness matrix is done and it ends up as two triangular matrices, one upper and one lower.

$$[K] = [L] [u] \quad (3.2)$$

Where:

[L] = the lower triangular matrix

[u]= the upper triangular matrix

By inserting equation 3.2 into equation 3.1, the following is obtained

$$[L][U]\{u\}=\{F\} \quad (3.3)$$

By using the following substitution:

$$\{w\}=[U]\{u\} \quad (3.4)$$

We can use the forward pass operation by first solving the matrix system for $\{w\}$, and then obtain the global vector of nodal displacement $\{u\}$ by the following :

$$[L] \{w\}=\{F\} \quad (3.5)$$

Then finally by using the back pass operation on a triangular system matrix we can compute the global nodal solution vector $\{u\}$ with the following

$$[U] \{u\}=\{w\} \quad (3.6)$$

The iterative method on the other hand initializes a DOF guess of the solution vector and by using the global stiffness matrix this solver finds the real DOF solutions by iterating to convergence. In the following the different types of solvers will be presented. [4]

3.3.1 The sparse direct solver

This solver uses the direct elimination of equations and requires a factorization of a linear system of equations into triangular matrix of a lower order. When this matrix is at place the method substitutes back and forward using this triangular system. If there are problems producing solutions, then the sparse direct solver is brilliant even if the initial equations are poorly conditioned [1].

3.3.2 The Preconditioned conjugate gradient (PCG)

This type of solver starts the solution step with element matrix formulations. Starting with a guess of all the DOF solutions, the solver sets up the global stiffness matrix and by iteration calculates the real DOF solution by convergence. As a proprietary preconditioner the solver uses material property and element dependent variables. The PCG is about 4 to 10 times faster in calculating the solutions than the Jacobi solver, but the solver requires about twice as much memory as the Jacobi

solver. This solver performs well on static analyses but are very limited in nonlinear analyses. This solver also performs well in contact analysis as long as rigid body motion does not occur because of the non-linearity [1].

3.3.3 The Jacobi conjugate gradient solver

As for the case of the PCG, this solver starts with element matrix formulations. Then it assembles the global stiffness matrix and by guessing on the DOF solutions to solve the real DOF solution by iteration, it converges. As a precondition this solver uses the diagonal of the stiffness matrix. However this method is usually used for thermal analysis [1].

3.3.4 The Incomplete Cholesky Conjugate Gradient Solver

This solver has the same principals as the Jacobi solver except that this solver is more robust to solve matrices which are poorly conditioned, but the solution needs twice the memory than the Jacobi solver. This analysis is well suited for thermal and electrical analysis but is available also for static, transient or harmonic analyses [1].

3.4 Newton-Raphson procedure

When the solution is nonlinear a new set of equations is to be used. For this purpose Ansys uses the Newton Raphson method. The procedure of the calculation consist of a set of simultaneous equations [4]:

$$[K] \{U\} = \{F^a\} \quad (3.7)$$

Where :

[K] is the coefficient matrix

$\{U\}$ is the vector of unknown degrees of freedom
 $\{F^a\}$ is the applied loads

Whenever the coefficient matrix itself is a function of the unknown DOF`s, then the equation above becomes nonlinear. To solve this Ansys uses the Newton-Raphson method with the following equations [4]:

$$[K_i^T] \{\Delta U_i\} = \{F^a\} - \{F_i^{nr}\} \quad (3.8)$$

$$\{U_{i+1}\} = \{U_i\} + \{\Delta U_i\} \quad (3.9)$$

Where

$[K_i^T]$ is the Jacobian matrix

$\{F_i^{nr}\}$ is the vector of restoring loads corresponding to the element internal loads.

At least more than one iteration is needed to obtain a solution. First $\{U_0\}$ has to be assumed, normally this value is the converged solution from the previous time step. However if it is the first time step this value is set = $\{0\}$. Then the updated tangent matrix $[K_i^T]$ and the restoring load $\{F_i^{nr}\}$ is computed due to the configuration of $\{U_i\}$. **In the next step $\{\Delta U_i\}$ is calculated and now the two parameters of $\{\Delta U_i\}$ and $\{U_i\}$ can be added together to obtain $\{U_{i+1}\}$ which is the next approximation. This is repeated until convergence is achieved [4].**

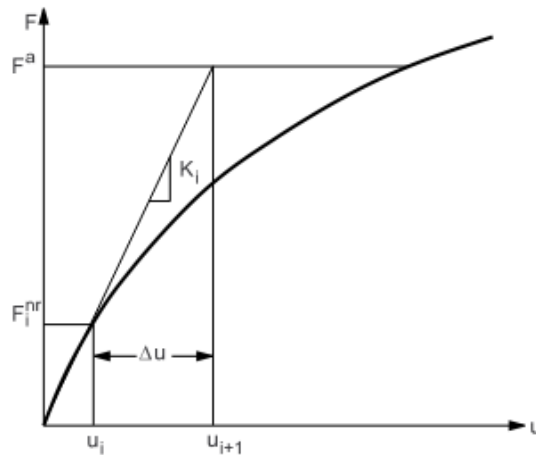


Figure 3.12 First iteration in Newton Raphson procedure

The solution obtained at the end of the iterations correspond to the load level $\{F^a\}$. None of the intermediate solution will be in equilibrium except for the last final converged solution. The principal for calculating the next iteration after the first one is shown below [4]:

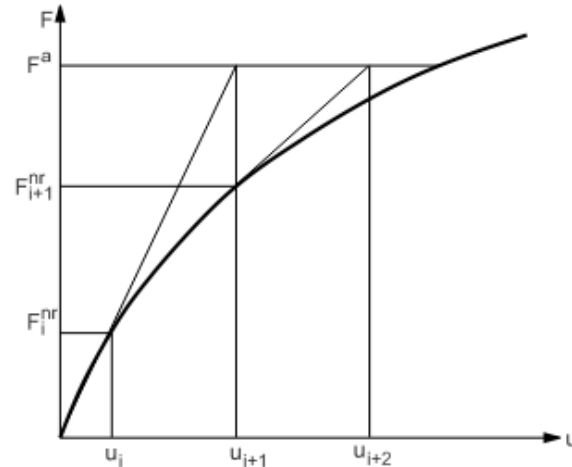


Figure 3.13 second iteration in Newton-Raphson procedure

3.5 Pretension

The beam and the column in this thesis are connected together with top and seat angle and bolts. When bolts are mounted at the construction site the bolts are tied to hold construction elements together before loading the structure. This also has to be simulated in Ansys, so two load steps has to be created. One where the bolt gets tightened, and a second load case where the external loads are applied. In the first case a pretension is defined in the bolts. This clamps the bolts together towards the center of the bolts. Ansys does this by generating a pretension element within the bolt. For this purpose Ansys used the PRETS179 element [4].

PRETS 179 can be used only when the model is meshed and it can be used to define both 2 and 3 dimensional pretension sections. This element has only one degree of freedom and that is in the translational x-direction, however if a pretension is needed in the y-direction Ansys transforms the geometry of the problem internally, so that even though the pretension-force arrow is always aligned with the x-axis the force works in the direction specified. After a pretension section is

defined a bolt load can be applied and 3 alternatives as to how the section behaves are given, these are [4]:

Lock = this value is the default, and connects the cutting plane on the pretension section.

Slid = this value disconnects the cutting plan on the pretension section

Tiny = this value applies a pretension load of 0.1 % of a specified load.

PRETS179 Geometry

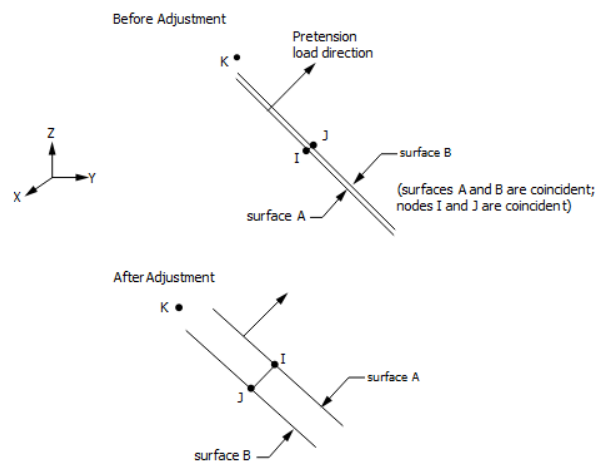


Figure 3.14 Tension after adjustments

Ansys also gives the opportunity to lock the pretension load after it has been applied, so that the bolts only get tightened during the first load step while in the next load step only the external loads are applied. If a lock is not made on the pretension, then for every load step a new pretension will be defined. This causes the bolts to clamp more than intended.

3.6 MESH

After a geometry is created, meshing of it is needed in order to run the analysis. The mesh of a model is crucial for the results predicted by the program. In meshing the model is divided into several elements and by using the finite element method it solves the mathematical equation needed to predict the solution for every element. When all element equations are solved the program adds up the solutions together to predict the behavior of the total structure. The finer the mesh is the more accurate the results become. The problem however with using a very fine mesh

is that the solution can become very time-consuming and the need for computers with a lot of free space is also needed. This is because a set of equations is needed for each element, and by using a fine mesh more elements are created and thus more equations is needed to obtain the solution. The optimal way to mesh is by using engineering sense to know where the stresses of interest are, this way a coarser mesh can be used in areas that are not of interest while a finer mesh can be created in the more interesting fields of the model.

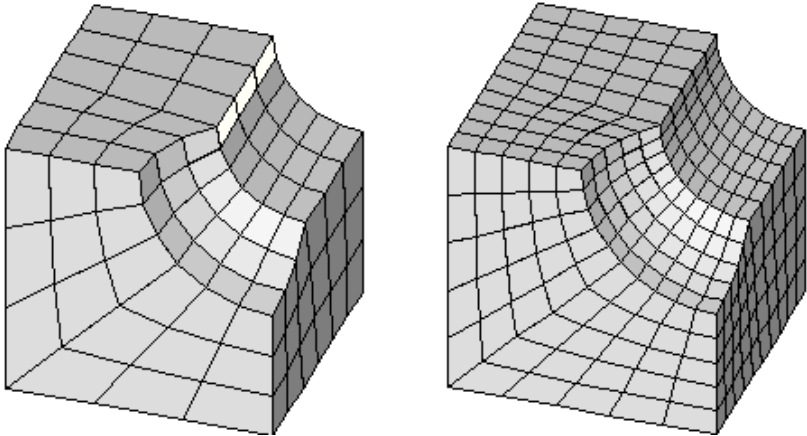


Figure 3.15 fine vs coarse mesh

In Ansys two main types of mesh can be used, this is free or mapped mesh. A free mesh has no restrictions in how the elements are shaped, and has no specific pattern assign to it. A mapped mesh however is restricted to the shape of the elements as well as the the pattern used to mesh it. In figure 3.16 the elements have a mapped mesh, and as seen the mesh has very obvious rows of elements. If this is preferred then when constructing the geometry it has to consist of fairly regular forms and shapes. For complex geometries the best option would be to use free mesh. This may visually not be the best choice, but it is a necessity in some cases.

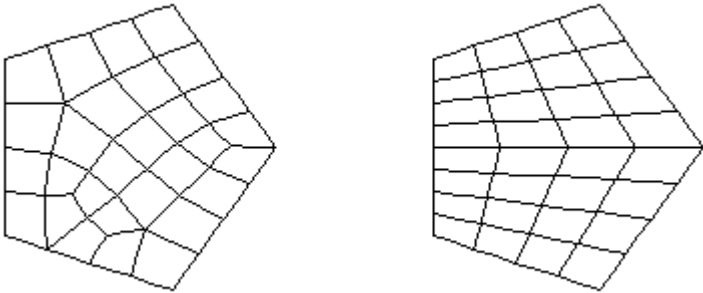


Figure 3.16 free vs mapped mesh

4 Numerical Modelling

4.1 Scripting in Ansys

In order to construct the model with its respective parameters, scripts has been written. The advantage of using a script based model is that any parameter can easily be changed unlike when using the graphical alternative. In the last case, the user needs to save every step during modeling formation. Since Ansys is a complex software, errors may occur for instance due to insufficient memory space of the computer used in the process. The possibility of losing the model and the need to start over would be time consuming and very frustrating.

The final scripts are presented in the Annex of this thesis. It is easy for the reader to copy and paste this script into the text editor in Ansys, in order to view our model and analyze the results. This is very convenient for structural engineers and for future research within the same field.

4.2 Connection description

In order to have reliable results, a validation of the FE-model needs to be performed. Normally experimental tests are done before modeling and the test results are used for the validation of the numerical model. The results from both cases are compared. If the results are within acceptable limits, the model can be safely used for further optimization and analysis of the construction. Because of the lack of resources at the university and the short time at hand, experimental testing could not be done for our model. So by searching the international literature, a paper published by a research team from Imperial College London related to testing of hybrid connections was selected for validation of our numerical model [14]. This paper presents hybrid connections between a tubular steel column and a glulam beam that are connected to each other in different ways. An experimental test as well as a numerical model in Abaqus of the connections are presented and the results are published. Because one of these connections was of interest for this thesis and for the validation of the results of the numerical analysis, the base numerical model used throughout the thesis is constructed according to an experimental specimen and the testing procedure described in the paper. Of course the effect on the connection's behavior as a result of

changing several parameters into the base model, in order to find an optimum solution, is the innovative part and the main contribution of the presented work.

The geometry of the connection under study (base model) is as follows:

- The glulam beam has a height of 405 mm, and width of 140 mm
- The steel column has the dimension 150 x 10 mm, with a steel grade of S355
- The top and seat angle has the dimension 150x200 mm, with a steel grade of S275
- The vertical bolts that goes through the glulam beam, are M12 with a steel grade 10.9
- The horizontal bolts that is connected to the column, are M16 with a steel grade 10.9

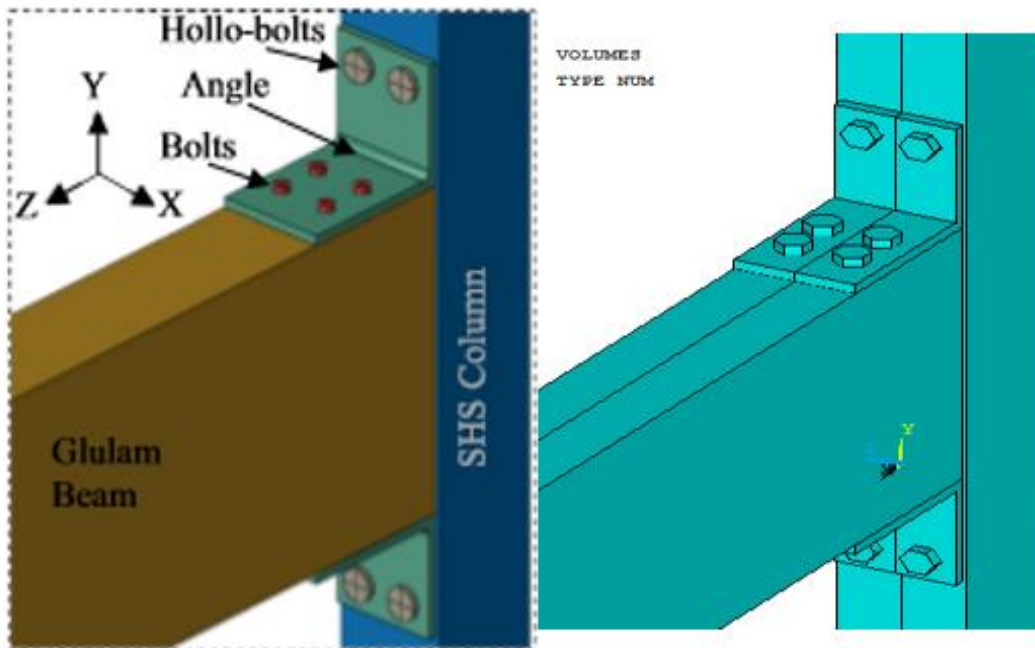


Figure 4.1 Abaqus vs. Ansys Model

Because timber is an orthotropic material its parameters are different in the various directions.

Table1
Mechanical Properties for glulam GL28h

Mech. Properties	Bending strength	Compressive Strength		Young's modulus		Shear modulus	Density
		Parallel	Perpend.	Parallel	Perpend.		
	σ_b [Mpa]	$\sigma_{c,0}$ [Mpa]	$\sigma_{c,90}$ [Mpa]	E_0 [Mpa]	E_{90} [Mpa]	G_{12} [Mpa]	ρ [Kg m^{-3}]
Average	33.5	33	3,6	12110	430	917	431

The geometry for the top and seat angle are seen in figure 4.2. [14]

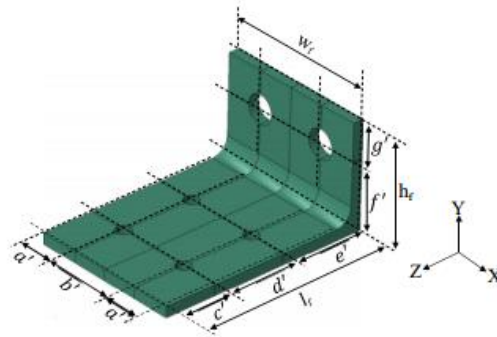


Figure 4.2 top and seat angle

Table 2
Distances between the bolt holes and the edges

a'	b'	c'	d'	e'	f'	g'
36	68	56	60	84	116	34

4.3 Assumptions and simplifications

Some simplifications had to be done because of the complexity of the experiment and the limit of the time available. Contact problems are highly nonlinear and there are many parameters to choose between. The following simplifications has been made when modelling and analyzing the connection

- The blind bolts that are mounted into the tubular column and then expanded beyond the thickness of the wall of the column, are simulated with a standard bolt with a nut. The behavior of the blind bolts used in the experiment are not well documented and modelling this would be highly time-consuming. This may cause some difference when validating the model

- Because of the computational time and the disk-space needed to converge towards a solution, only a quarter of the length of the beam is modelled.
- The verification is done in the elastic zone of the materials, so when plotting the moment-rotation curve a comparison is done within this area
- When plotting the moment-rotation curve we used the empirical formulas proposed by Sang-Sup Lee and Tae-Sup Moon [15], to calculate the generated elastic moment in the connection by using the rotation extracted from Ansys. The proposed formulas are presented below

$$M = (P_e \cdot l_e) \times (d_b + g_1) \times \left(\frac{d_b}{\Delta_e} \right) \cdot \phi \quad (4.1)$$

Where d_b = is the depth of the beam
 Δ_e = is the elastic deformation
 ϕ = is the rotation of the connection
 l_e = is the effective length of the angle

And the reaction force P_e is given by the following expression

$$P_e = \frac{12EI}{g_1^3} \left(\frac{g_1 + g_2}{g_1 + 4g_2} \right) \cdot \Delta_e \quad (4.2)$$

Where the gauge distances g_1 , and g_2 of the angle are given below

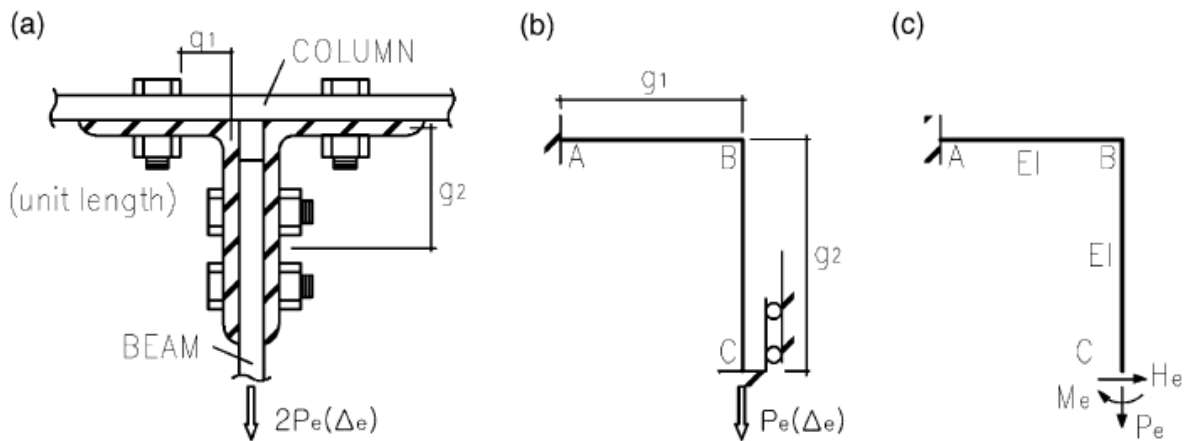


Figure 4.3 Behavior of angle

Sang sup-lee argues that it is the top angle only that resist the bending moment. This is also assumed for our case studies. In the formulas presented above the effective length of the angle are made as a simplification. Sang-Sup Lee assumed this length to be the diameter of the nut of the bolt connected to the column, multiplied with the number of bolts connected to the column. Because of lack of information about the blind bolts used in the experiment, the diameter of the nuts inside the column in the FE-analyses is assumed to cover the whole width of the tubular column. This gives us a more accurate value for the moment obtained in the connection compared to the experiment related to the rotation that occurs from any given load.

4.4 Contact parameters used in ANSYS APDL

The parameters described in section three will here be presented with the argued reason for its use.

4.4.1 Element type

Between SOLID185, 186 and 187, the last one is preferred for this model. SOLID185 does not have midside nodes, and thus creating a less accurate geometry of the meshed model. SOLID186 has midside nodes, but the element creates a total of 20 nodes within one element. This causes many iterations and can become very expensive in computational time. SOLID187 however produces only 10 nodes and the element has midside nodes. This creates fairly accurate result because of the midside nodes, without using much CPU time. For this reason it is the most suitable element for our model [3]

4.4.2 Contact and target elements

The surface to surface elements have several advantages over the node to node element CONTA 175. The surface elements provides better contact results for typical engineering purposes They have no restriction on the shape of the target surfaces, and they support both high and low order elements on both contact and target surfaces. This means that they are well suited both for elements that have midside nodes, or the ones with corner nodes only.

Because of the flexibility of the surface to surface elements and the provision of result better than that from node to node elements, we use surface to surface elements in our modelling. Between CONTA173 and CONTA174 the latter is the preferred one because of the special geometry of the model. When using the element with midside nodes we create a more accurate meshed model. This is because the geometry has more nodes to build up around and thus making it more accurate than an element without midside nodes. For this reason we use CONTA174 in our model.

The corresponding target element used in the model are TARGET 170. The main difference between TARGET 169 and the one used in the model, is that TARGET 169 are used on 2-dimensional models. We obviously have a 3-dimensional contact problem and for this reason TARGET 170 is the correct choice [3].

4.4.3 Contact analysis type

In general it is recommended to use either pair-based or general contact. Between these two, general contact is a less efficient and less robust solution compared to the pair-based. The general contact decides automatically definitions with limited user interventions. Also general contact is often used when the choice of contact and target surfaces are unclear or unknown. Because we want an efficient and a robust solution we use the pair-based contact with the option of using symmetrical pair [2].

4.4.4 Contact surface behavior

The standard contact surface is used in all contact pairs except between the bolt heads and the angles. Here the surface behavior are set to be bonded always. This is because after pretension is set, the bolt heads are supposed to have contact with the angles at all times [2].

4.4.5 Friction

Choosing the correct friction coefficient are crucial to the results. Too much friction can cause no movement of the elements at all, while if it is too small the elements can move quite a lot compared to each other. So in order for our model to be validated against the experiment the coefficient of friction is set 0.7 as in the numerical model done in the paper [14].

4.4.4 Contact algorithm

The contact algorithm has to be chosen carefully so that it matches the problem at hand. Even though the Augmented Lagrange method in some analyses may require more iterations than the penalty method, it usually leads to better conditioning and it is less sensitive to the magnitude of the contact stiffness than that of the penalty method.

The pure Lagrange multiplier requires additional iterations to make the contact condition stable. This often results in an increase in computational time and is therefore less effective than the Augmented Lagrange method. For some of the regular analyses it is recommended to use the Augmented Lagrange method instead of the Lagrange multiplier method. The MPC algorithm is recommended when using bonded or no separation contact behavior. Because we in the model are using both bonded, and standard contact, we use the Augmented Lagrange method for all cases except for the bonded ones. Here we use the MPC algorithm taking into consideration the reasoning above [2].

4.4.5 Equation solvers

Ansys has defined limits for the use of different equations solvers. For models with a maximum DOF of 500 000, the sparse direct solver is recommended. From 500 000 and beyond different types are relevant. In our model we are far away from 500 000 DOF, so the recommended solver for our analyses are the direct sparse solve. In addition to this the sparse direct solver is brilliant when convergence problems occurs, or when the equations are poorly conditioned. For these reasons the sparse direct solver is used in our analyses [1].

4.5 External loading, pretension, and boundary conditions

By trying with different displacement set on the end of the beam it became clear that for displacements larger than approximately 7 mm the materials enters into plastic behavior. Therefore the connections was analyzed for loads up until a displacement of 7 mm was obtained. The total displacement are not set at once, but it is divided into seven load steps so as to not cause failure. In each load step a deflection of 1 mm is therefore applied. As in real life the bolts has to be

tightened before the external load is applied. This is done by using two load steps as explained in chapter 3.5. The applied torque to the bolt connected to the column is 190 Nm and the applied torque to the bolts connected to the beam is 100 Nm [14]. The conversion from torque to pretension is given in the chapter 2.4 and are as follows

$$M_a = k \cdot d \cdot F_p \tag{4.3}$$

The coefficient K varies between 0.15-0.3 depending on the threading of the bolt and the surface friction. Because of lack of information about these parameters a factor of 0.3 is used in all cases [17].

Table 3
Pretension calculated from given torque

Diameter of the bolt	Pretension
M12 Vertical	27KN
M16 Vertical	20 KN
M20 Vertical	16 KN
M16 Horizontal	39KN

Our model is constrained from moving out of plane, and the column is fixed near to the top and near to the bottom so as not to move in any directions.

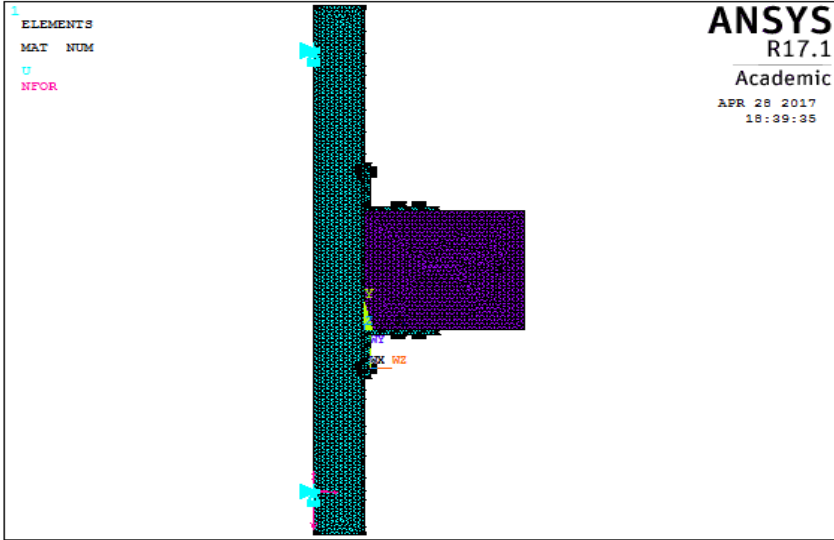


Figure 4.4 constraining the column

4.6 Singularities

When modelling in FE-software one of the most important parameters to observe is the stresses that occurs, specifically von Mises stress. This is the average stress that occurs in the model, and it has to be analyzed with accordance to the yielding of the material. What is important to know is that not all the stresses calculated from the FE- software are real stresses that needs to be taken into account. These stresses are referred to as singularities. A stress singularity is a point in the mesh model where the stress does not converge toward a specific value. If the mesh is refined in this point the stresses only increases and theoretically the stress singularity is infinite.

A typical place for the singularities is at the point which the load is applied, or at sharp re-entrant corners. Even though the values in these points are high this does not mean that the values for the entire model are incorrect, but the singularity will cause the nearby elements to have some higher value than in reality. So when analyzing the results in this thesis the values with some distance from the singularities are used [5].

4.7 Model validations

The FE model described in this thesis is validated with the experimental test done in Imperial College. We have been focusing our model on elastic behavior because of the complexity of plasticity in Ansys. One of the most reliable way to verify a numerical model with an experiment is by plotting the moment-rotation curve. For each load step up until the seventh, rotation in the beam is calculated from the given displacement from Ansys. Then the obtained moment is calculated with the formula proposed by Sang sup-Lee. As we can see from figure 4.5 a reasonable prediction of rotational stiffness can be achieved by the numerical model developed in Ansys. There are some minor differences but these are within the limit of what is acceptable.

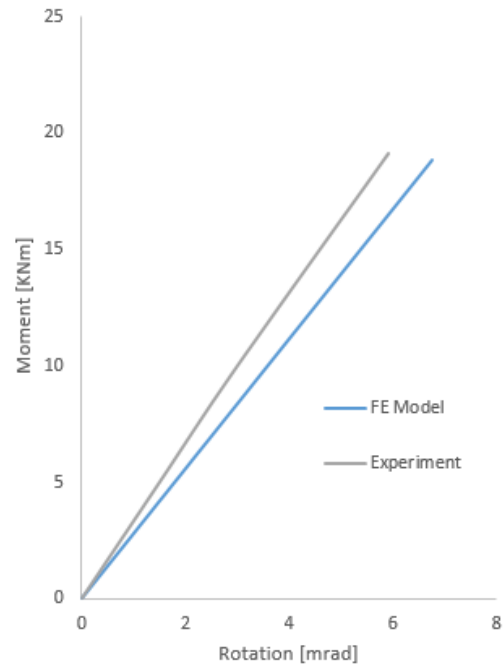


Figure 4.5 Verification of the model

Because the rotational stiffness are fairly accurate compared to the experiment, the numerical model made in Ansys is assumed to produce result which are both solid and reliable. Hence the optimization of the connection can be executed.

For the comparing of von Mises stress the values tested in the laboratory of the angle bracket will be used. These values showed a yield strength from tests on 14 angle brackets and their values are higher than the yield strength of 275 n/mm² defined from the factory. Therefore our models will be compared to, and analyzed with the values tested in the experiment [14].

Table 4
Stress values taken from experimental test

Top and Seat angle	Yield stress [Mpa]	Ultimate stress [Mpa]
L 200 x 150 x 15	408	514

4.8 Case studies

In order to for us to analyze the different parameters of the connection, 7 different case studies are carried out. To analyze between the original connection and the optimized ones, the verification result from the FE-software for the original connection are presented as well. The following cases and changes has been analyzed.

Table 5
Case studies

Case nr	Modifications in connection
1	This is the predicted results from the FE-software for the original connection
2	The thickness of the top and seat angle are changed from 15 to 18mm
3	The thickness of the top and seat angles are increased to 20 mm
4	The diameter of the vertical bolts are changed from 12 to 16mm
5	The diameter of the vertical bolts are changed to 20 mm
6	A reduction of 20 mm in the width of the top and seat angle are made
7	The thickness of the top and seat angle are changed to 12 mm, with a 14 mm stiffener placed in the middle of the top angle

In all the cases the main results to be compared are the moment-rotation relationship, von Mises stresses and the initial rotational stiffness. The methodology used to obtain these result are described below.

4.8.1 Moment rotation

In all case studies a displacement of 10 mm was set on the end of the beam. Then every load step was analyzed up until the seventh, as mentioned in chapter 4.5. For each load step the horizontal deflection of the glulam beam was together with the height of the beam (d_b) used to calculate the rotation by using the following equation

$$\varnothing = \arctan (\Delta_e / d_b) \quad (4.4)$$

Also when the horizontal deflection of the beam is known, the reaction force P as shown in figure 12 can be calculated. By inserting these parameters into the formula proposed by Sang Sup-Lee, the obtained moment in the connection can be calculated, and the moment-rotation relationship can be established

For the case study with stiffener, the formula proposed by Sang Sup-Lee to calculate the occurred moment in the connection cannot be used without adding a coefficient. In order to find this coefficient the following simplified method is proposed

First a model of the top angle itself is made. Then the horizontal leg is constrained in the y-direction and the angle itself, is constrained in the z- direction. To resemble the action of the bolt, the vertical leg is constrained at the top in x-direction as well. Then a load is applied in the same manner as the load P as seen in figure 4.3. The deflection in the x-direction is then extracted. The next step is to model the angle by itself with a stiffener and make the same loading and constraints. Then the deflection in the x- direction is extracted. When the two deflections are found the calculation of the stiffness of both angles in the x-direction are made with the following simplified equation

$$F = K * X \quad (4.5)$$

Where

- F is the applied load
- K is the stiffness of the angle in X-direction
- X is the deflection of the angle

In this manner it became clear that the changes in stiffness is quite some amount. By using equation 4.5 a ratio between the two stiffnesses is found and multiplied with the reaction force P , which again is used to calculate the occurred moment in the stiffened connection

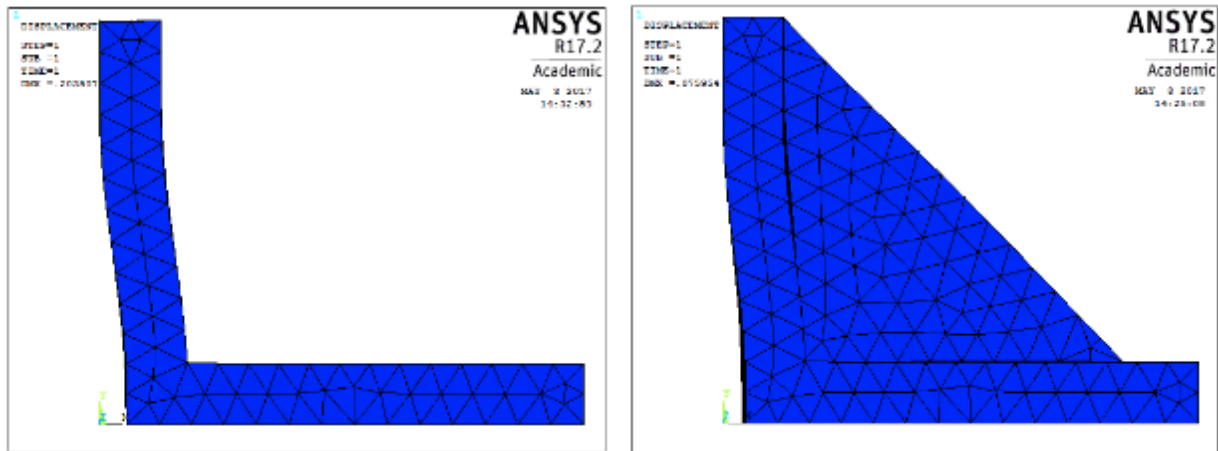


Figure 4.6 Deflection in the top angle with and without stiffener

4.8.2 von Mises stress

When analyzing von Mises stress, a set of coordinates is used to monitor the stresses at the same points in the top angle. By doing this we can monitor how the same areas on the angle changes with the different cases. Because case 1, 2 and 3 have different thicknesses of the angles, the coordinates differ slightly, but this is taken into consideration so that the points actually are in the same positions in every case. In case 7 where the stiffener is applied in the middle of the top angle a comparison of the stresses with case 1 is not accurate because the difference in geometry. The reason for this is that almost all the stresses goes through the stiffener instead of through the corner of the angle. So the magnitude of the stresses will not be comparable

4.8.3 Initial rotational stiffness

Eurocode 3 part 1-8, provides coefficients needed to calculate the initial rotational stiffness of connections. However for square hollow sections these are not included. For the connection in this thesis the coefficients needs to be calculated first in order to obtain the initial rotational stiffness

of the whole connection. The contribution to the rotational stiffness for the top and seat angle in total can be evaluated with the contribution from the bearing stiffness of the column face and the axial stiffness of the bolts connected to the column. For the calculation of the bearing stiffness of the column face, Elghazouli uses the following equation [16].

$$k_{cf} = \frac{\pi E t_c^3}{12(1 - \nu^2) C_t \left(\frac{b_c - t_c}{2}\right)^2} \quad (4.6)$$

Where

- E is Young's modulus for steel
- t_c is the thickness of the wall of the column
- ν^2 is Poisson's ratio for steel
- b_c is the width of the column face

To calculate the tension and compression stiffness in the horizontal leg of the top and seat angle Elghazouli uses the following equation [16]]

$$k_t = \frac{E p t_f}{a'} \quad (4.7)$$

Where

- E is Young's modulus for steel
- p is the width of the angles
- t_f is the thickness of the angle
- a' is the distance from the inner bolt hole to the vertical leg of the angles

With regards to the stiffness of the bolt that connects the top angle to the column, Elghazouli have presented curves from which the stiffness can be read for the blind bolts. For standard bolts Elghazouli uses equation 4.8 to calculate the stiffness, but this is for the connection of the top angle to the beam only.

$$k_{sb} = \frac{A_b E}{L_b} N \quad (4.8)$$

Where

- E is Young's modulus for steel
- A_b is area of the bolt
- L_b is the length of the bolt subjected to axial tension
- N is the number of bolts that contributes to the stiffness

As seen in equation 4.8, the area, length, and Young's modulus for the bolts are the parameters used to calculate the axial stiffness of the standard bolts. So even though the formula is related to the connection between the angles and the beam, it is assumed that this also is valid for the connection between the angle and the column as well. When the stiffness coefficients are found, they are used in the following formula to calculate the stiffness of the top angle [14].

$$K_{Top} = \frac{1}{\frac{1}{K_{CF}} + \frac{1}{K_{HB}} + \frac{1}{K_t}} \quad (4.9)$$

As seen in the equation the stiffness for the hollow bolts are used. This is substituted with the stiffness of a standard bolt as explained above. To calculate the stiffness of the seat angle however, the following formula are used [14].

$$K_{Bottom} = \frac{1}{\frac{1}{K_{CF}} + \frac{1}{K_t}} \quad (4.10)$$

The difference between the two stiffnesses is that there are no stiffness contribution from the bolts in the seat angle. This is because the bolts are not subjected to tension hereto calculate the stiffness of the total connection the following formula are used [14].

$$S_{config1,init} = \frac{d_b^2}{\frac{1}{K_{Top}} + \frac{1}{K_{Bottom}}} \quad (4.11)$$

Where d_b is the depth of the glulam beam. This final equation is used to calculate the initial rotational stiffness for the different cases. In order to further verify whether the connection is rigid, semi-rigid or nominally pinned the criteria mentioned in section 2.3.3 will be examined.

5 Results

In this chapter we will present the results obtained from the case studies. Because the discussion of the result will be done in the next chapter, only the result will be presented here.

5.1 Case 1

Table 6

Moment rotation at different load steps for case 1

Load steps	Horizontal deflection of Beam (mm)	ϕ (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,10	0,25	10,4	0,69
4	0,39	0,96	40,4	2,69
5	0,78	1,93	80,8	5,37
6	1,80	4,44	186,4	12,40
7	2,74	6,77	283,8	18,87

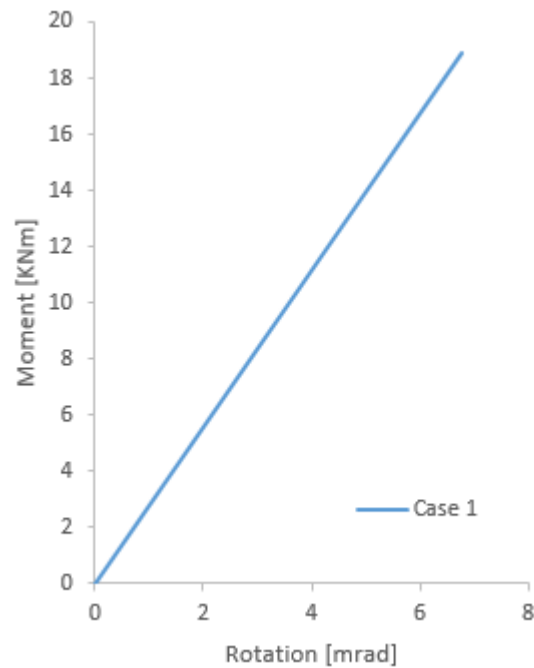


Figure 5.1 The moment-rotation curve in case 1

Table 7

Von Mises stress for top angle at different coordinates for case 1

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case 1
1	15	440	0	401
2	15	460	0	267
3	15	480	0	138
4	35	419	0	329
5	55	419	0	153
6	75	419	0	34
7	125	419	0	195

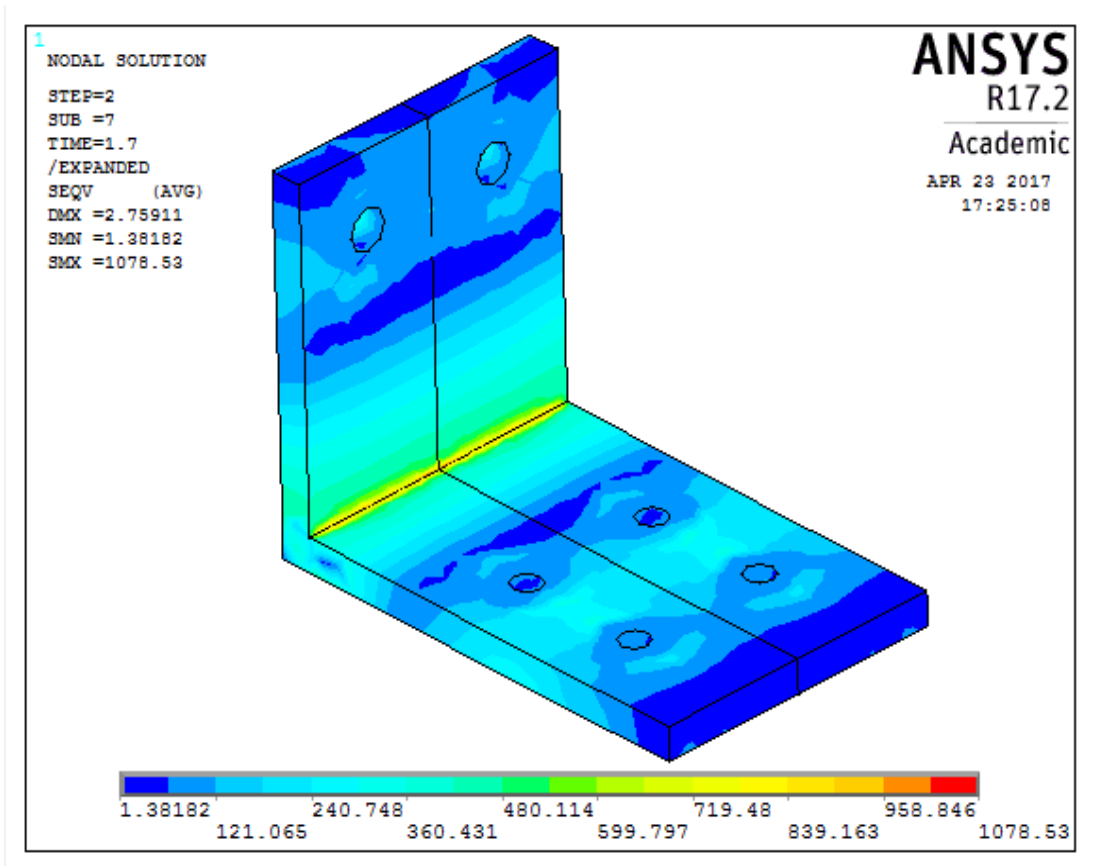


Figure 5.2 von Mises stress in the top angle in case 1

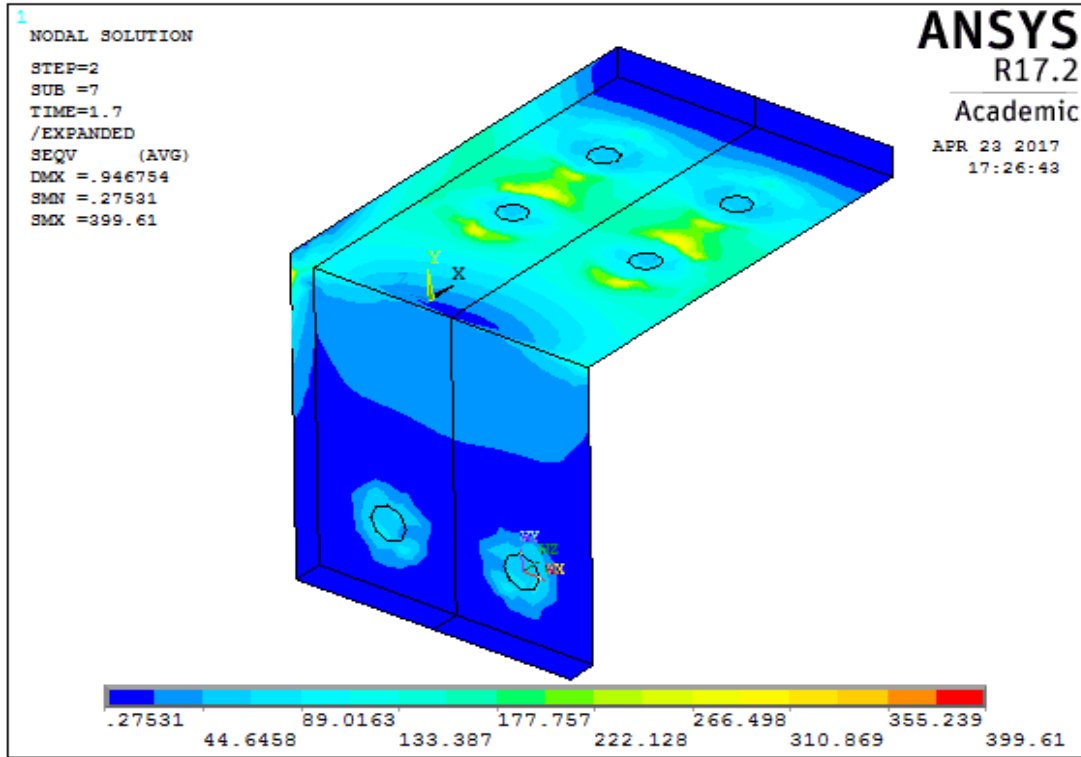


Figure 5.3 von Mises stress in the seat angle in case 1

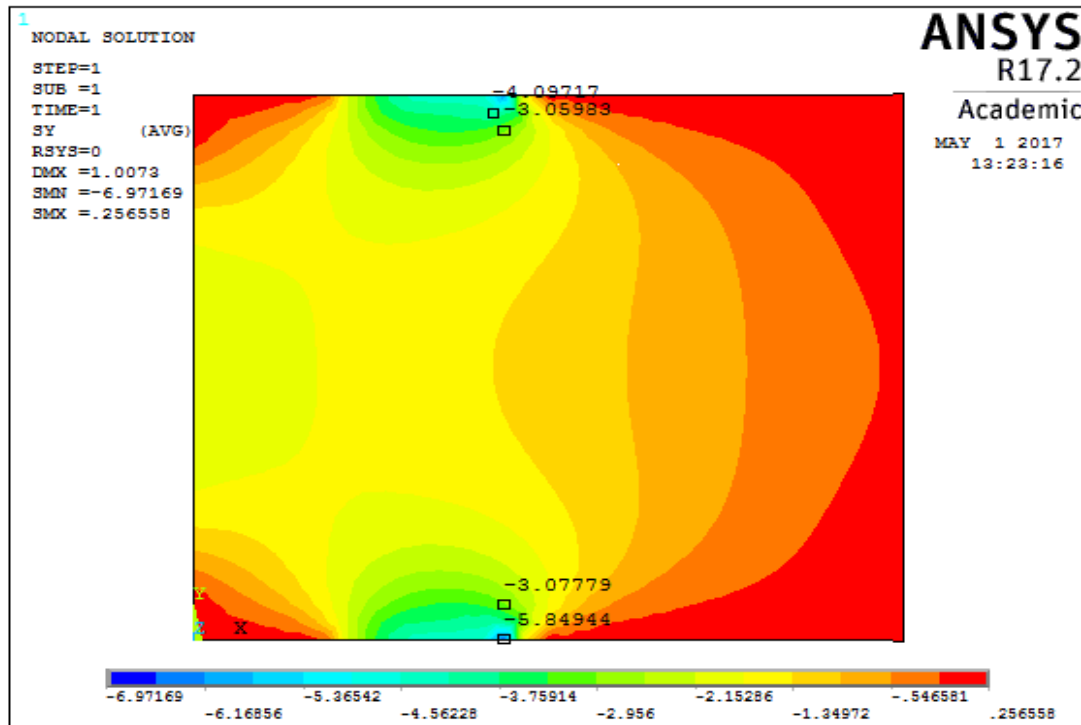


Figure 5.4 Stress perpendicular to the glulam beam at the first load step in case 1

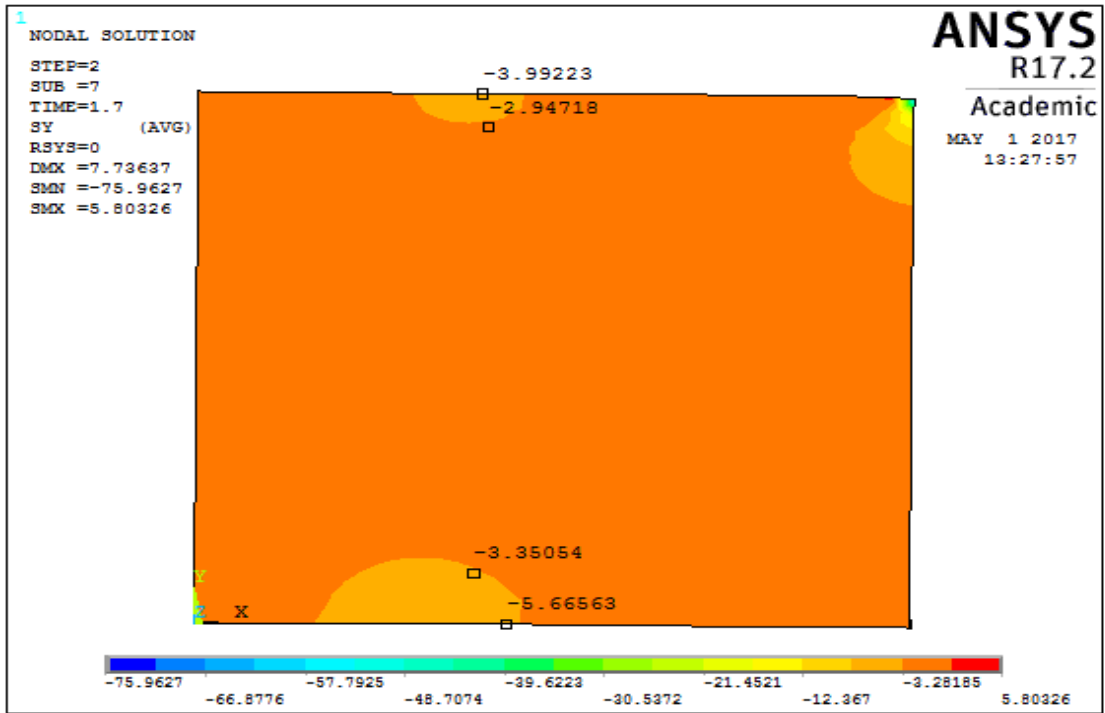


Figure 5.5 Stress perpendicular to the glulam beam at the second load step in case 1

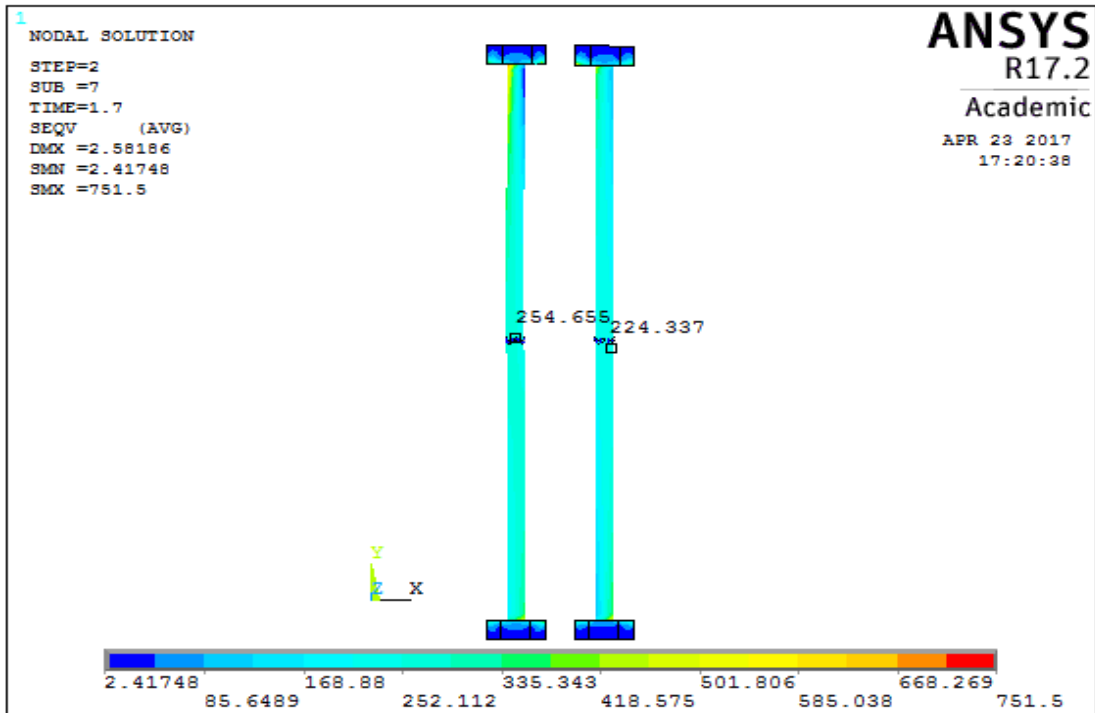


Figure 5.6 von Mises stress in the vertical bolts in case 1

5.2 Case 2

Table 8

Moment rotation at different load steps for case 2

Load steps	Horizontal deflection of Beam (mm)	\varnothing (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,00	0,01	0,4	0,03
4	0,23	0,58	41,9	2,78
5	1,17	2,89	209,4	13,92
6	2,10	5,19	375,8	24,99
7	3,03	7,48	542,3	36,06

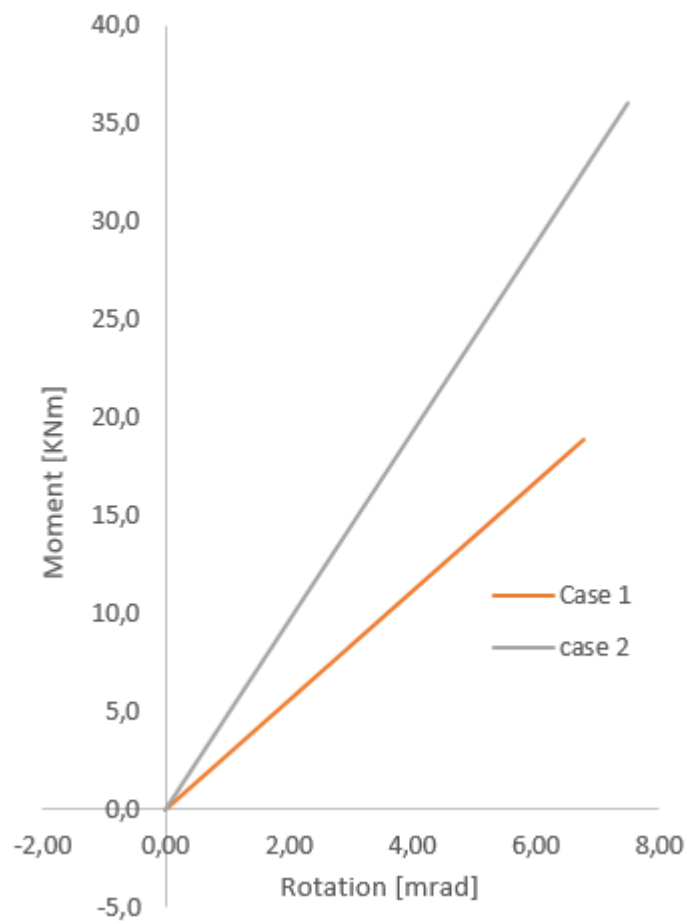


Figure 5.7 Comparison of moment-rotation curves

Table 9
 Von Mises stress for top angle at different coordinates for case 2

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case 2
1	18	442	0	330
2	18	462	0	250
3	18	482	0	156
4	38	422	0	260
5	57	422	0	139
6	77	422	0	29
7	126	422	0	139

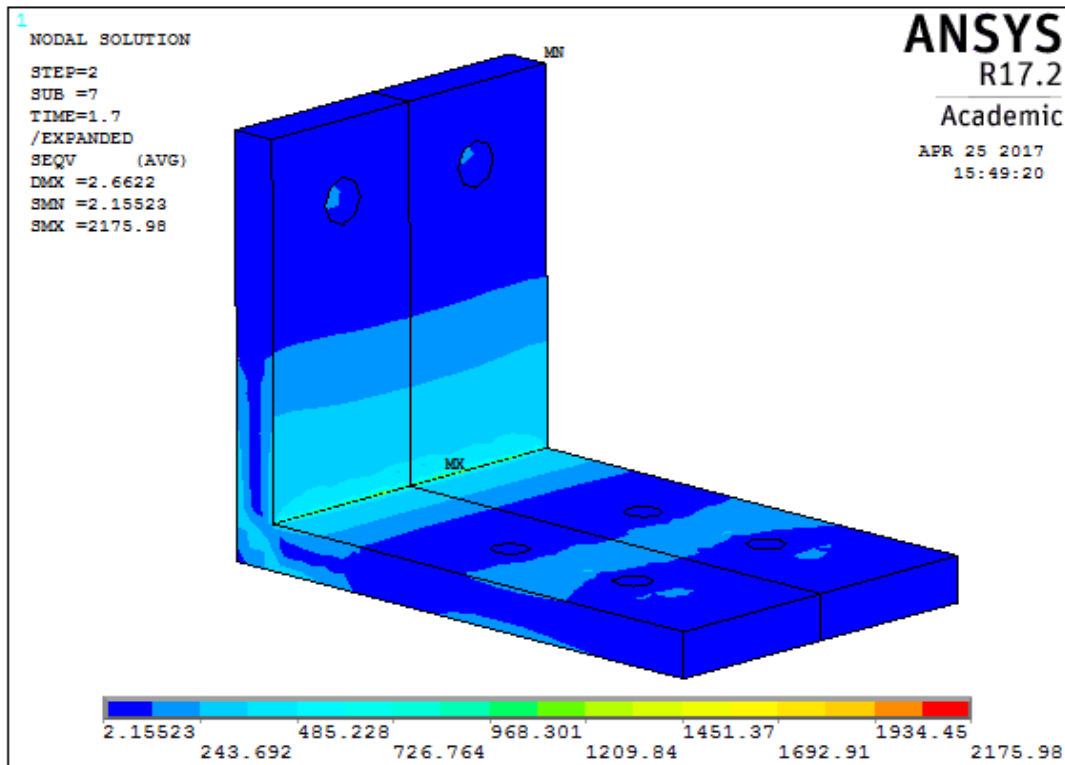


Figure 5.8 von Mises stress in the top angle in case 2

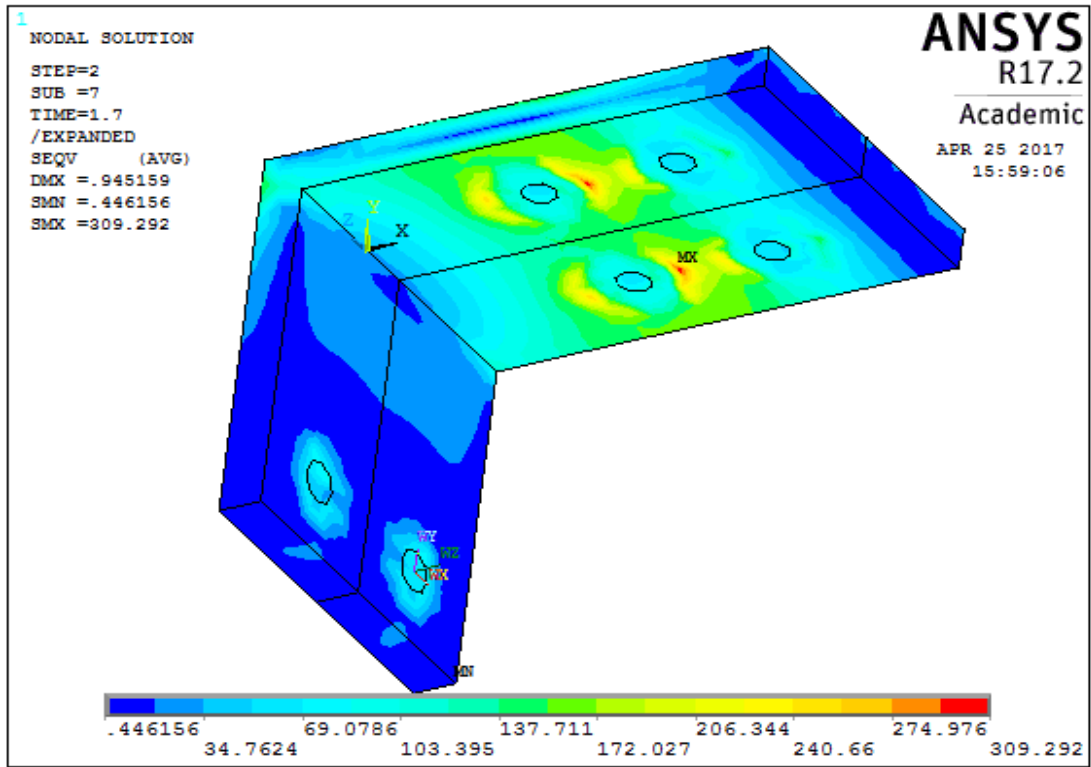


Figure 5.9 von Mises stress in the seat angle in case 2

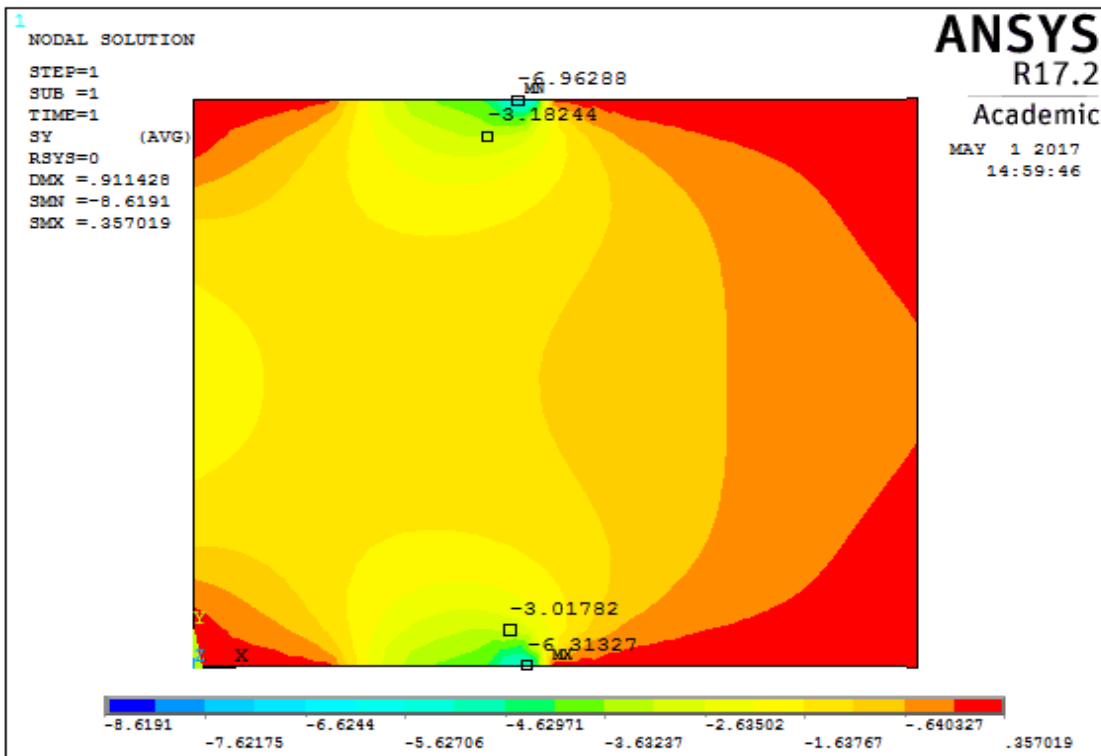


Figure 5.10 Stress perpendicular to the glulam beam at the first load step in case 2

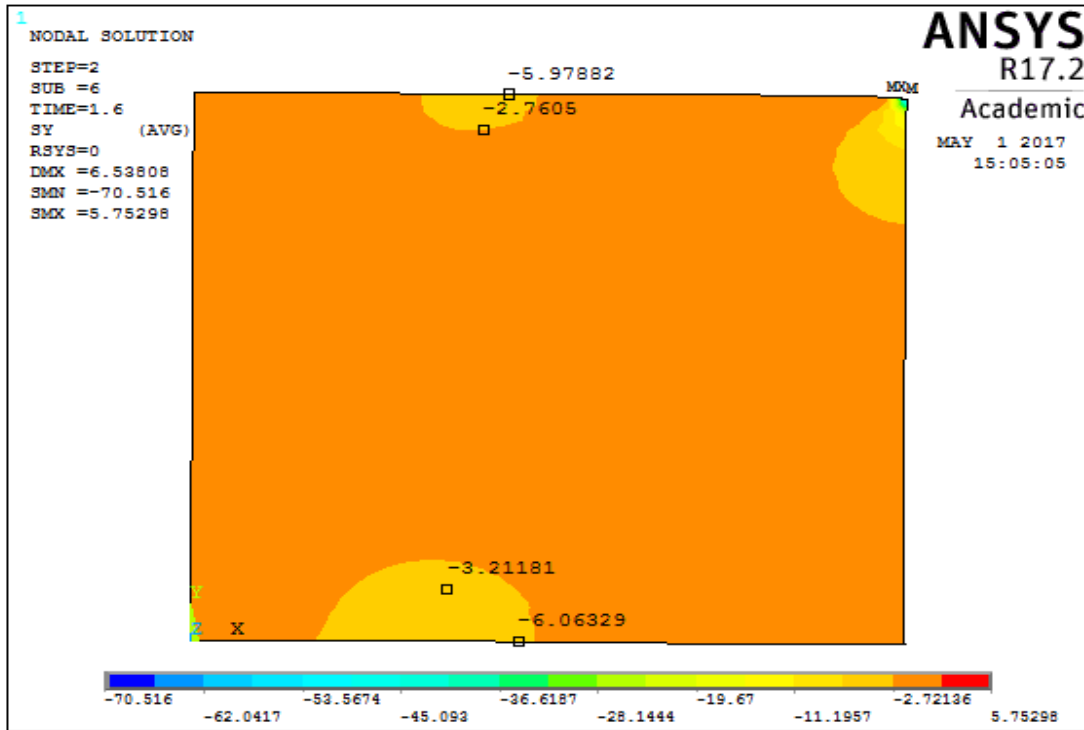


Figure 5.11 Stress perpendicular to the glulam beam at the second load step in case 2

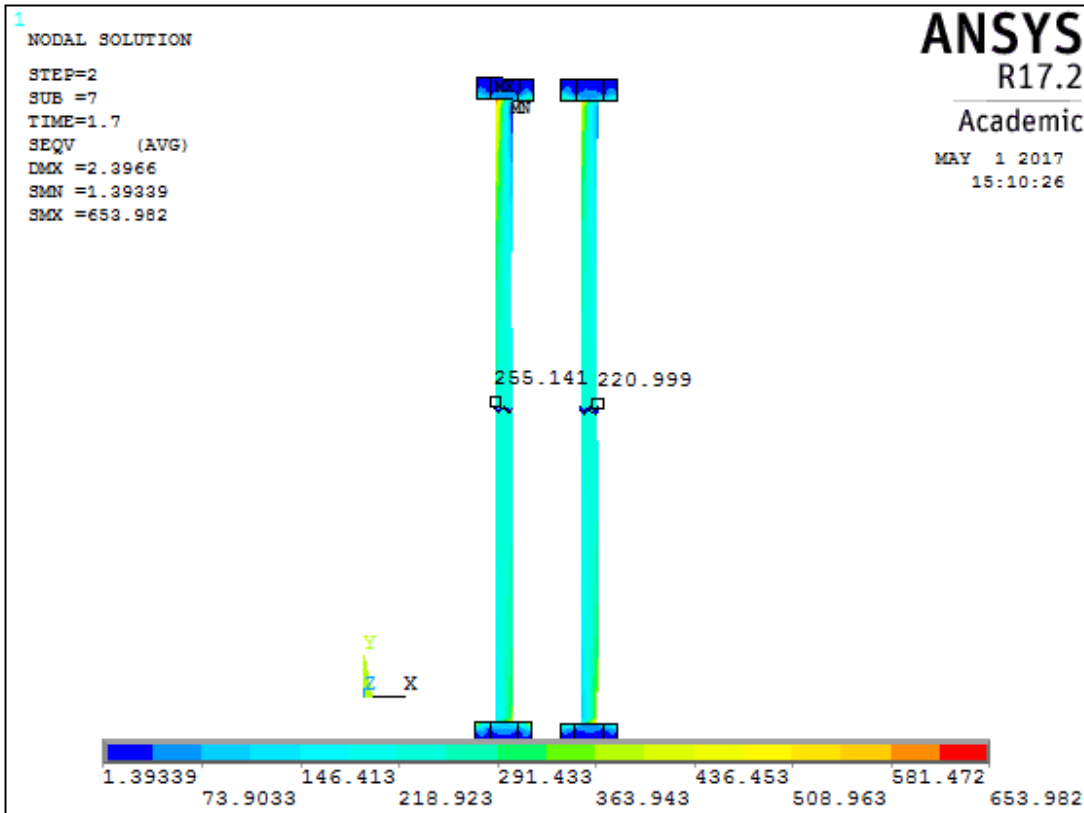


Figure 5.12 von Mises stress in the vertical bolts in case 2

Table 10

Von Mises comparison between case 1 and case 2

Reference points	S.EQV (N/mm ²)		% Reduction
	Case 1	Case 2	
1	401	330	17,6
2	267	250	6,5
3	138	156	-12,9
4	329	260	21,1
5	153	139	9,2
6	34	29	15,5
7	195	139	28,7
Average in reduction			12,2

5.3 Case 3

Table 11

Moment rotation at different load steps for case 3

Load steps	Horizontal deflection of Beam (mm)	\varnothing (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,20	0,49	49,1	3,27
4	0,37	0,91	90,3	6,01
5	1,29	3,19	316,7	21,06
6	2,26	5,58	554,8	36,90
7	3,19	7,88	783,2	52,08

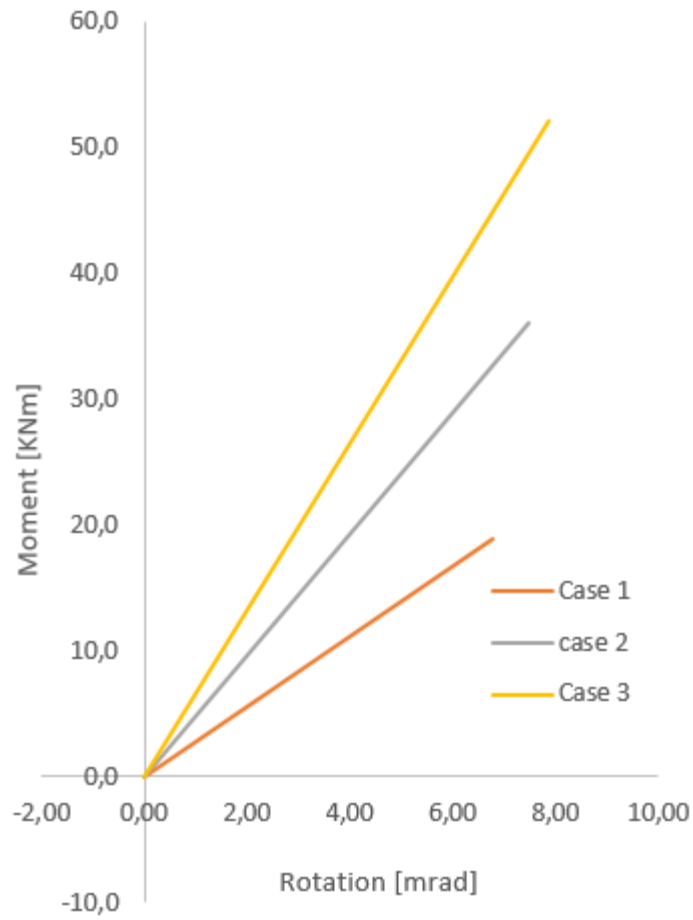


Figure 5.13 Comparison of moment-rotation curves

Table 12

Von Mises stress for top angle at different coordinates for case 3

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case 3
1	20	442	0	305
2	20	462	0	226
3	20	482	0	148
4	38	425	0	252
5	57	425	0	142
6	77	425	0	37
7	126	425	0	90

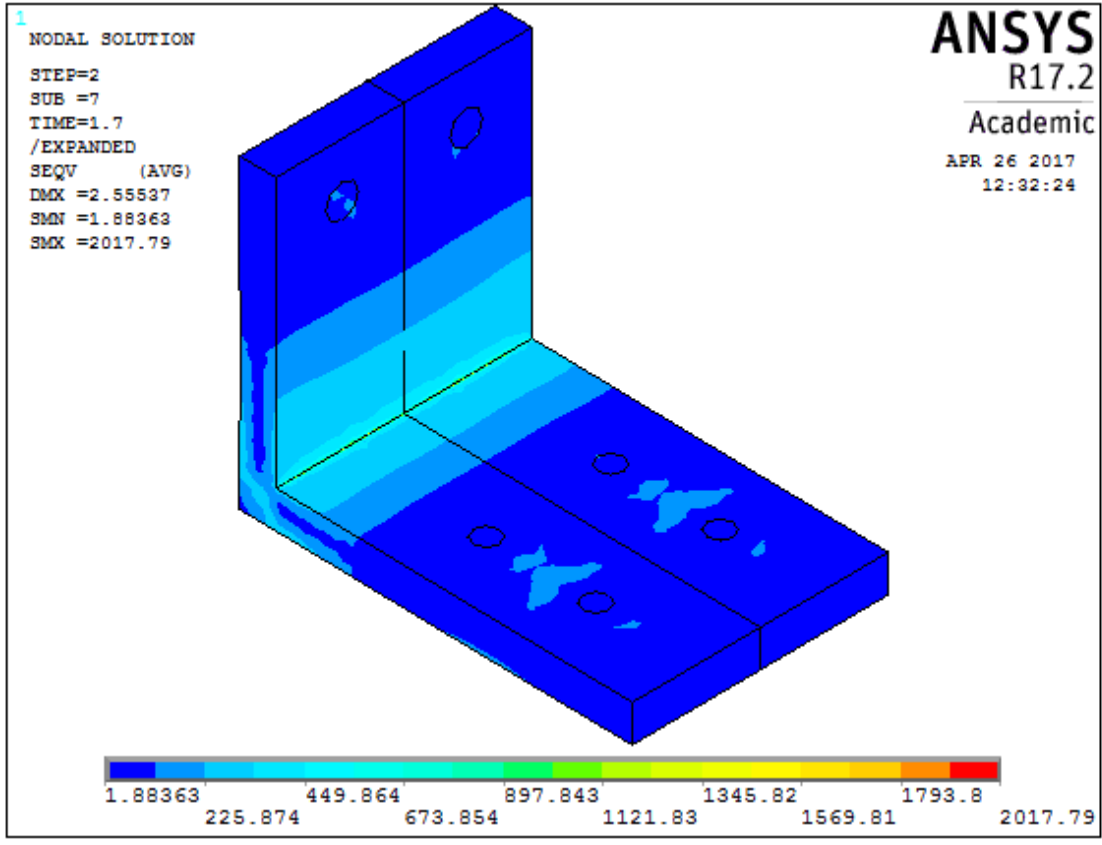


Figure 5.14 von Mises stress in the top angle in case 3

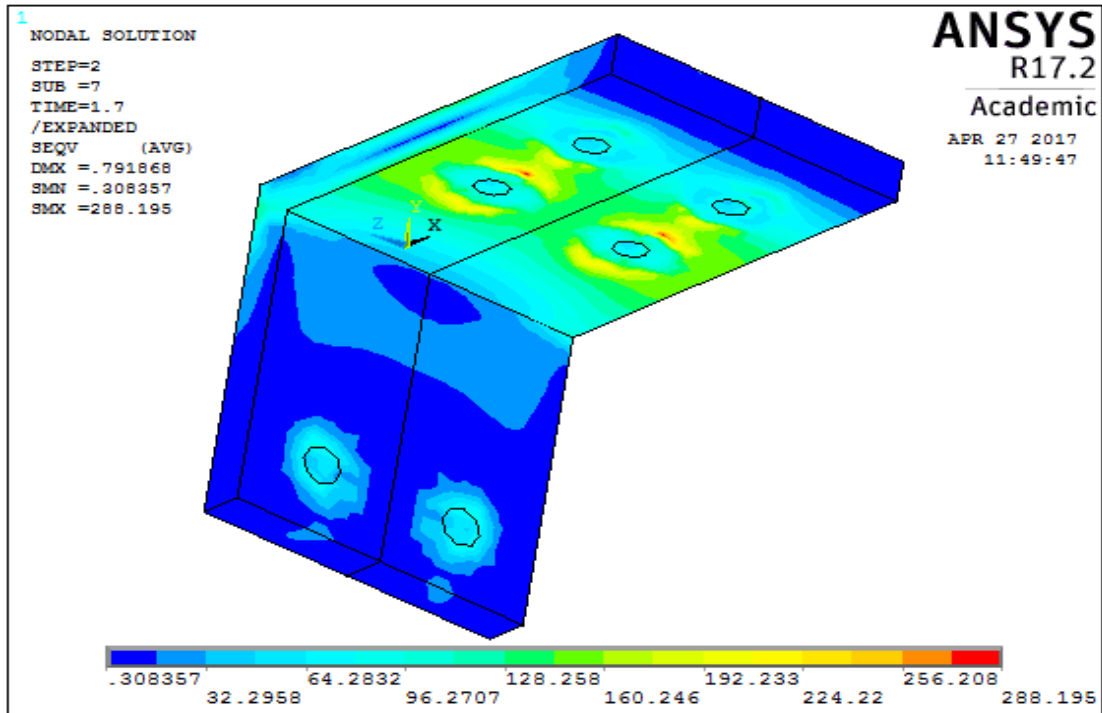


Figure 5.15 von Mises stress in the seat angle in case 3

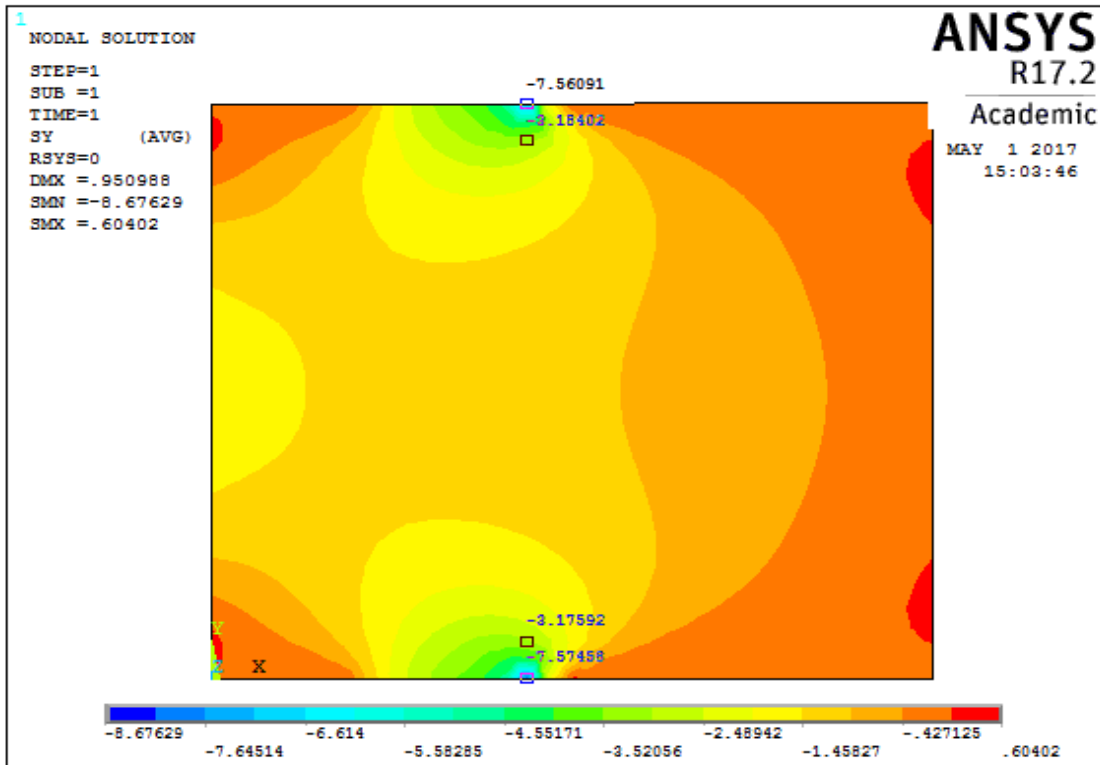


Figure 5.16 Stress perpendicular to the glulam beam at the first load step in case 3

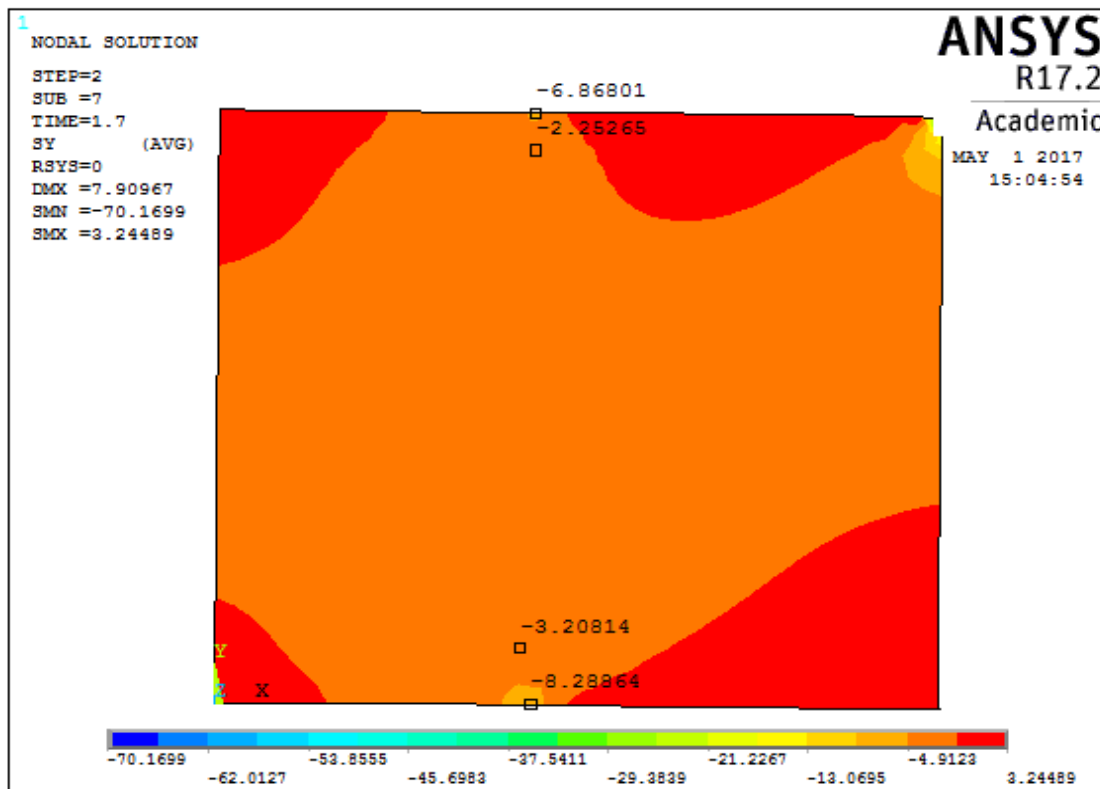


Figure 5.17 Stress perpendicular to the glulam beam at the second load step in case 3

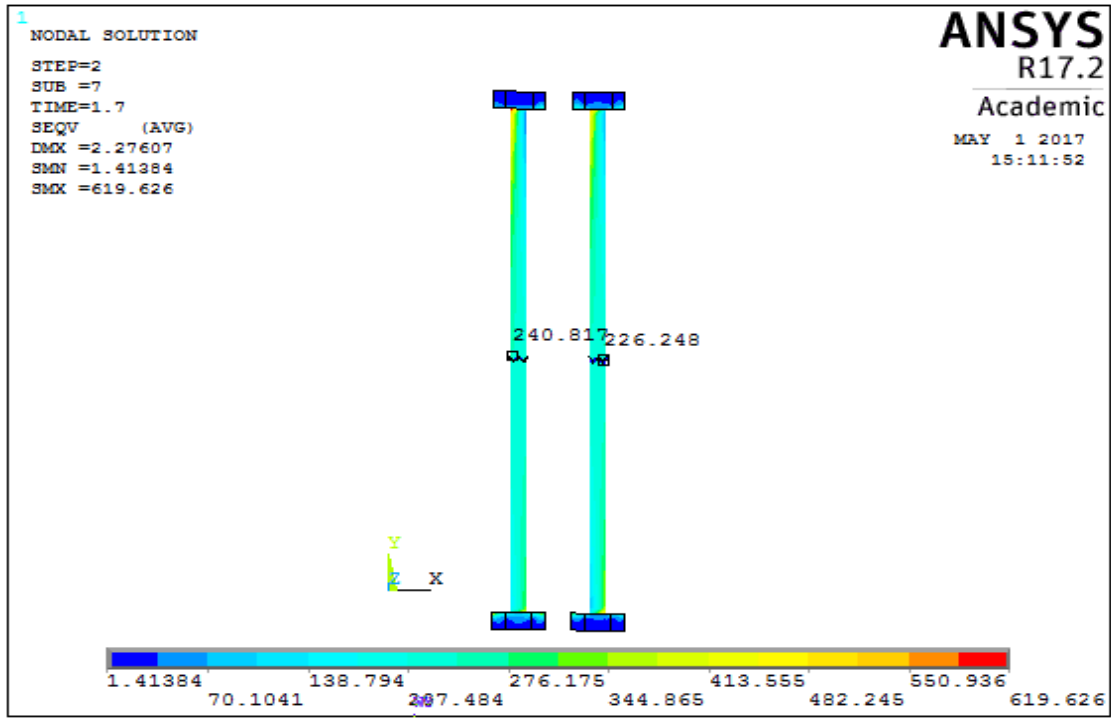


Figure 5.18 von Mises stress in the vertical bolts in case 3

Table 13
 Von Mises comparison between case 1 and case 3

Reference points	S.EQV (N/mm ²)		% Reduction
	Case 1	Case 3	
1	401	305	23,9
2	267	226	15,4
3	138	148	-7,2
4	329	252	23,4
5	153	142	7,5
6	34	37	-7,8
7	195	90	53,6

Average in reduction 15,5

5.4 Case 4

Table 14

Moment rotation at different load steps for case 4

Load steps	Horizontal deflection of Beam (mm)	\varnothing (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,27	0,67	28,0	1,86
4	0,92	2,27	95,3	6,34
5	1,83	4,52	189,5	12,60
6	2,79	6,89	289,0	19,22
7	3,72	9,18	385,3	25,62

Table 15

Von Mises stress for top angle at different coordinates for case 4

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case 4
1	15	440	0	320
2	15	460	0	218
3	15	480	0	120
4	35	419	0	265
5	55	419	0	130
6	75	419	0	27
7	125	419	0	123

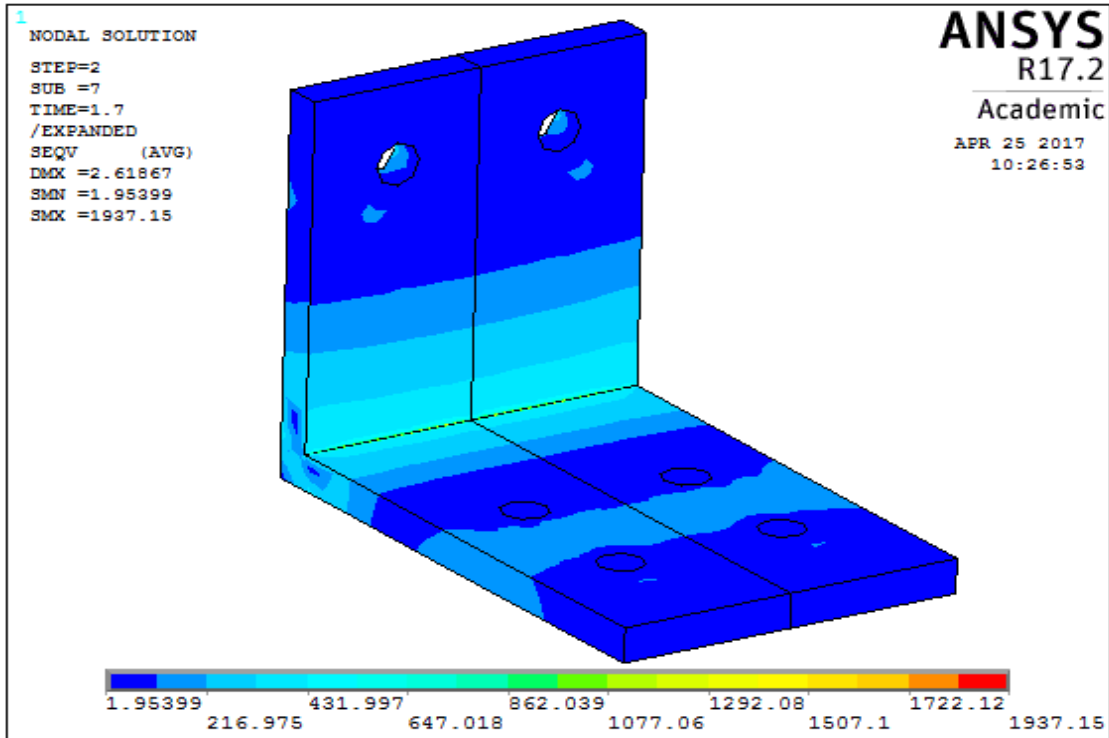


Figure 5.19 von Mises stress in the top angle in case 4

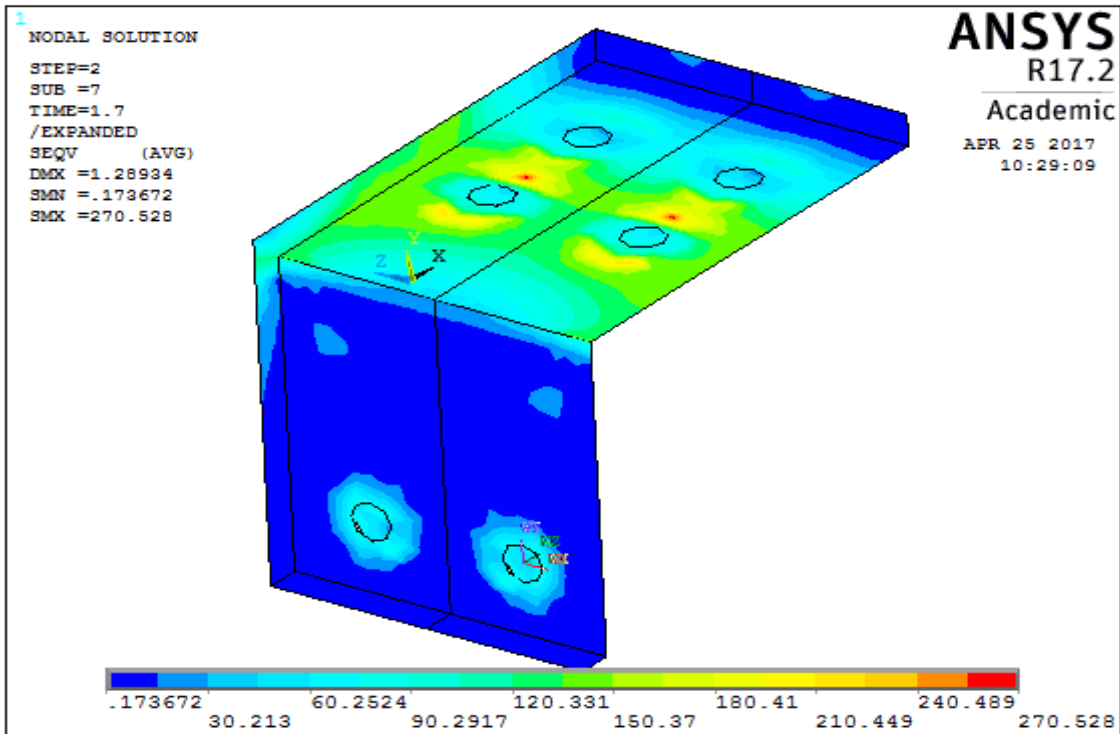


Figure 5.20 von Mises stress in the seat angle in case 4

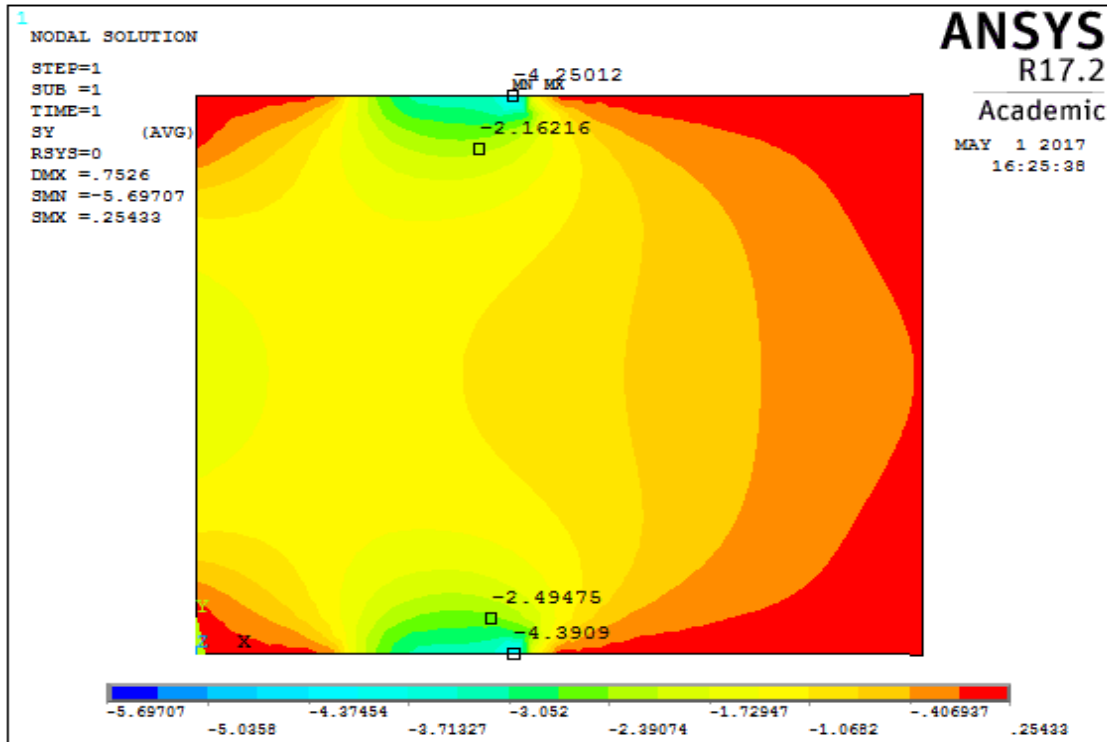


Figure 5.21 Stress perpendicular to the glulam beam at the first load step in case 4

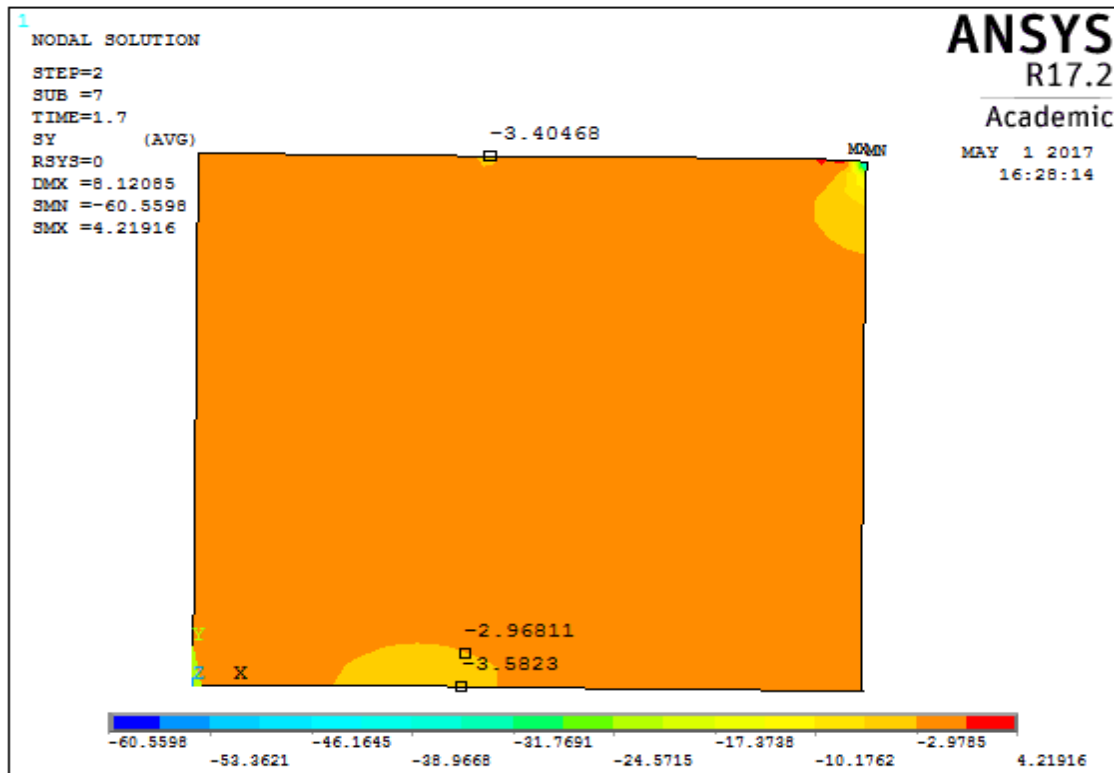


Figure 5.22 Stress perpendicular to the glulam beam at the second load step in case 4

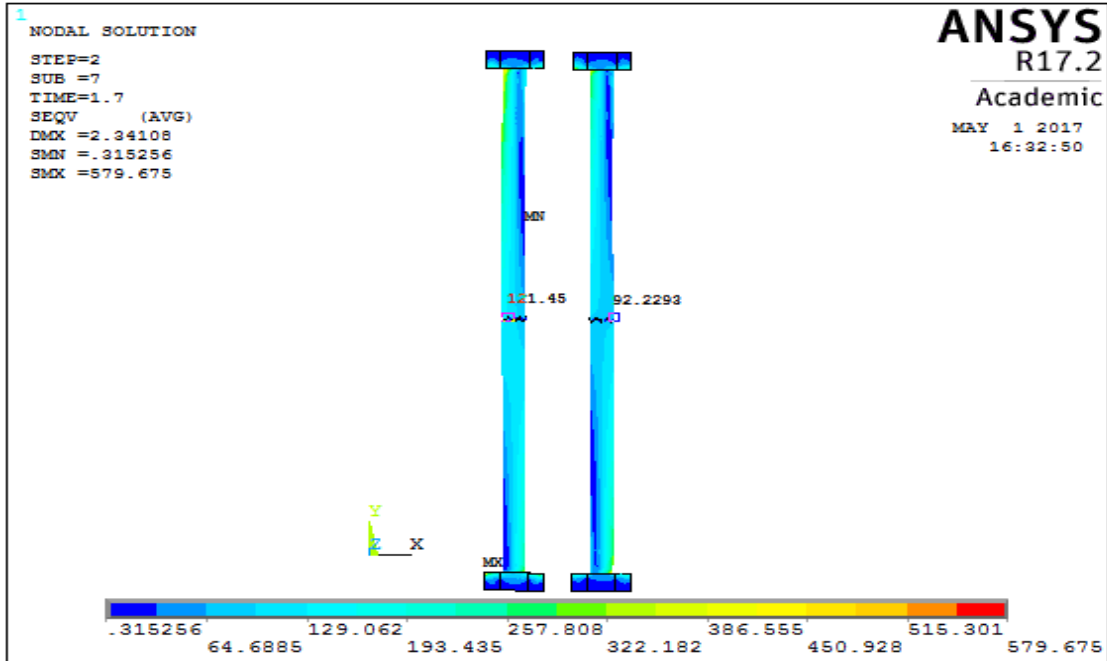


Figure 5.23 von Mises stress in the vertical bolts in case 4

Table 16

Von Mises comparison between case 1 and case 4

Reference points	S.EQV (N/mm ²)		% Reduction
	Case 1	Case 4	
1	401	320	20,2
2	267	218	18,4
3	138	120	13,0
4	329	265	19,5
5	153	130	15,0
6	34	27	21,7
7	195	123	37,2
Average in reduction			20,7

5.5 Case 5

Table 17

Moment rotation at different load steps for case 5

Load steps	Horizontal deflection of Beam (mm)	ϕ (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,60	1,47	61,8	4,11
4	1,53	3,78	158,5	10,54
5	2,46	6,07	254,7	16,94
6	3,39	8,36	350,6	23,31
7	4,30	10,62	445,4	29,62

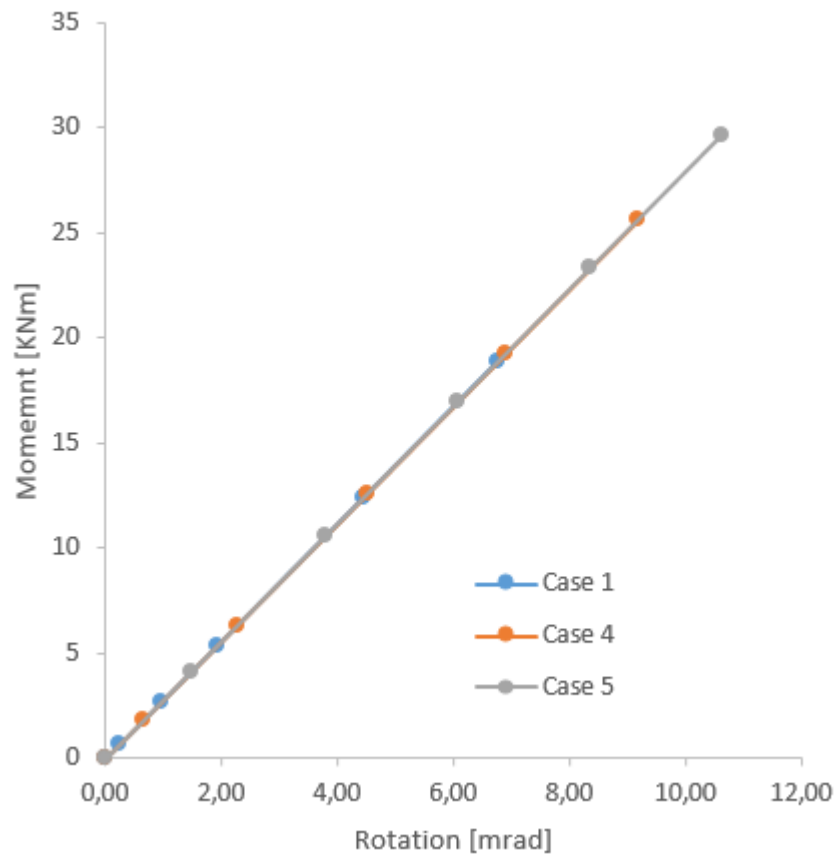


Figure 5.24 Comparison of moment-rotation curves

Table 18

Von Mises stress for top angle at different coordinates for case 5

Reference points	Global coordinates (mm)			S.EQV (N/mm ²) Case 5
	x	y	z	
1	15	440	0	277
2	15	460	0	189
3	15	480	0	106
4	35	419	0	232
5	55	419	0	116
6	75	419	0	23
7	125	419	0	77

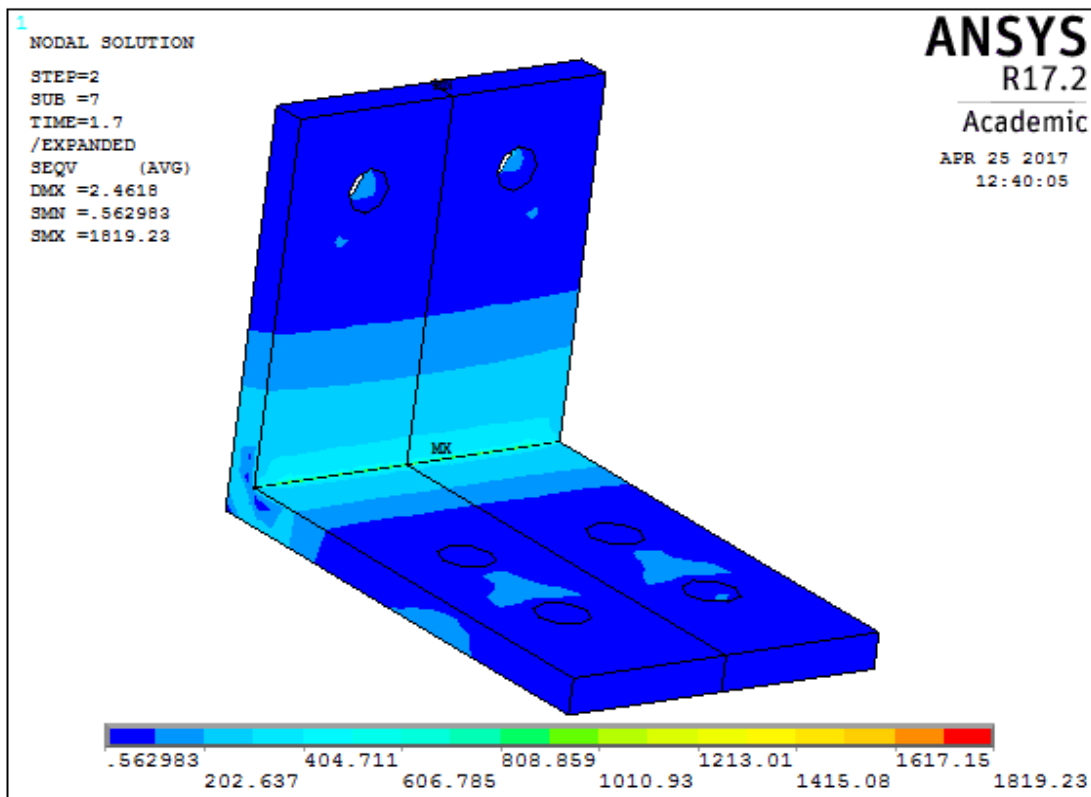


Figure 5.25 von Mises stress in the top angle in case 5

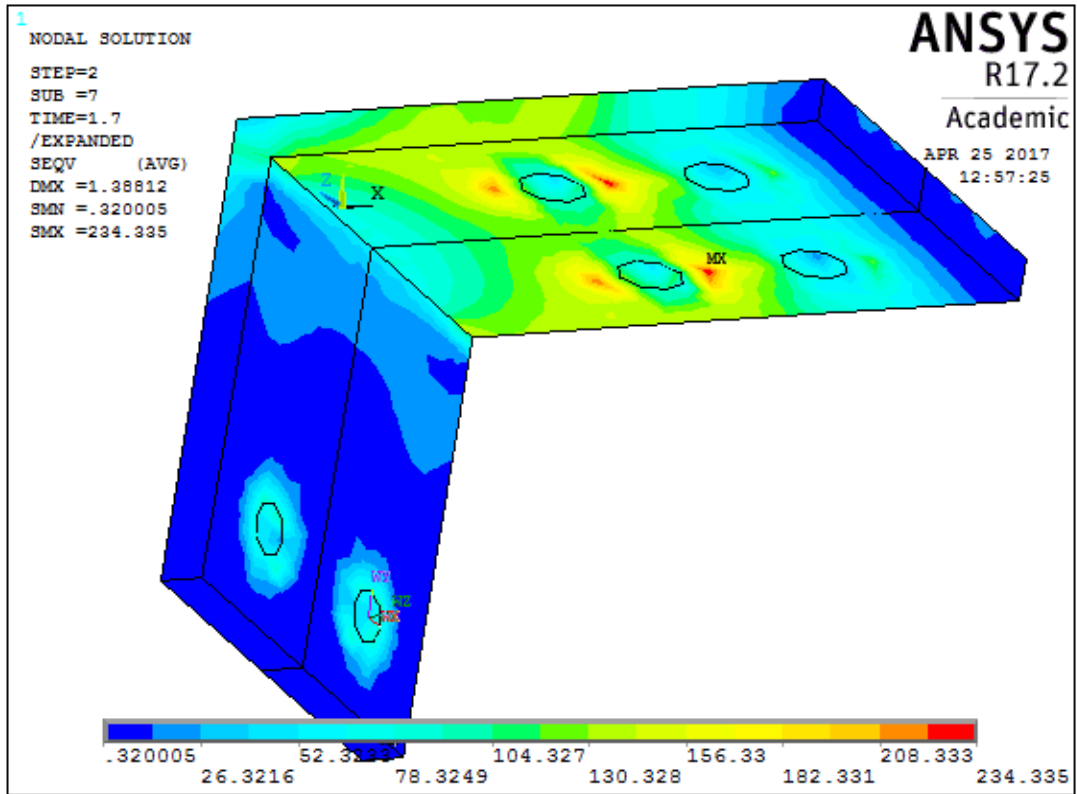


Figure 5.26 von Mises stress in the seat angle in case 5

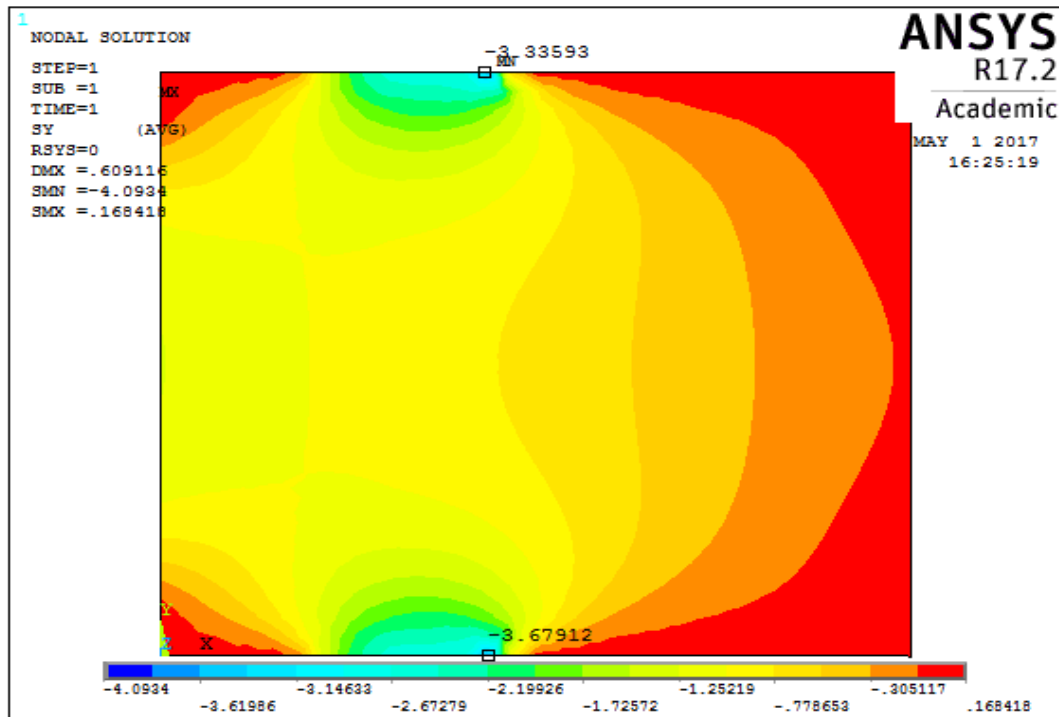


Figure 5.27 Stress perpendicular to the glulam beam at the first load step in case 5

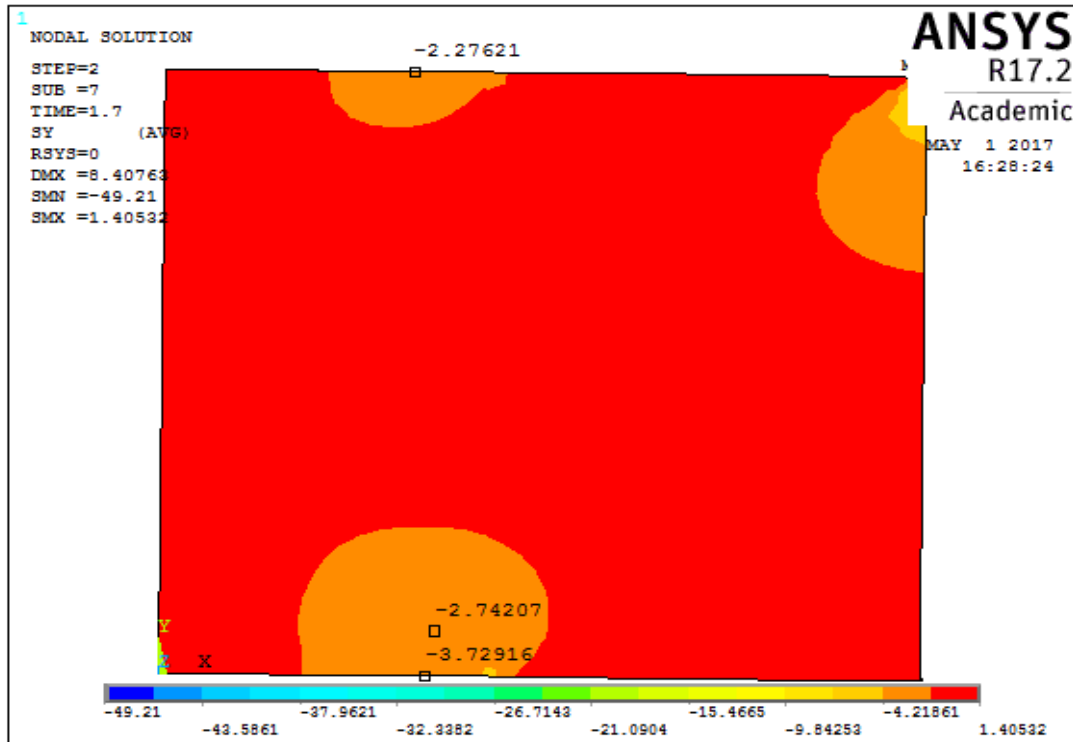


Figure 5.28 Stress perpendicular to the glulam beam at the second load step in case 5

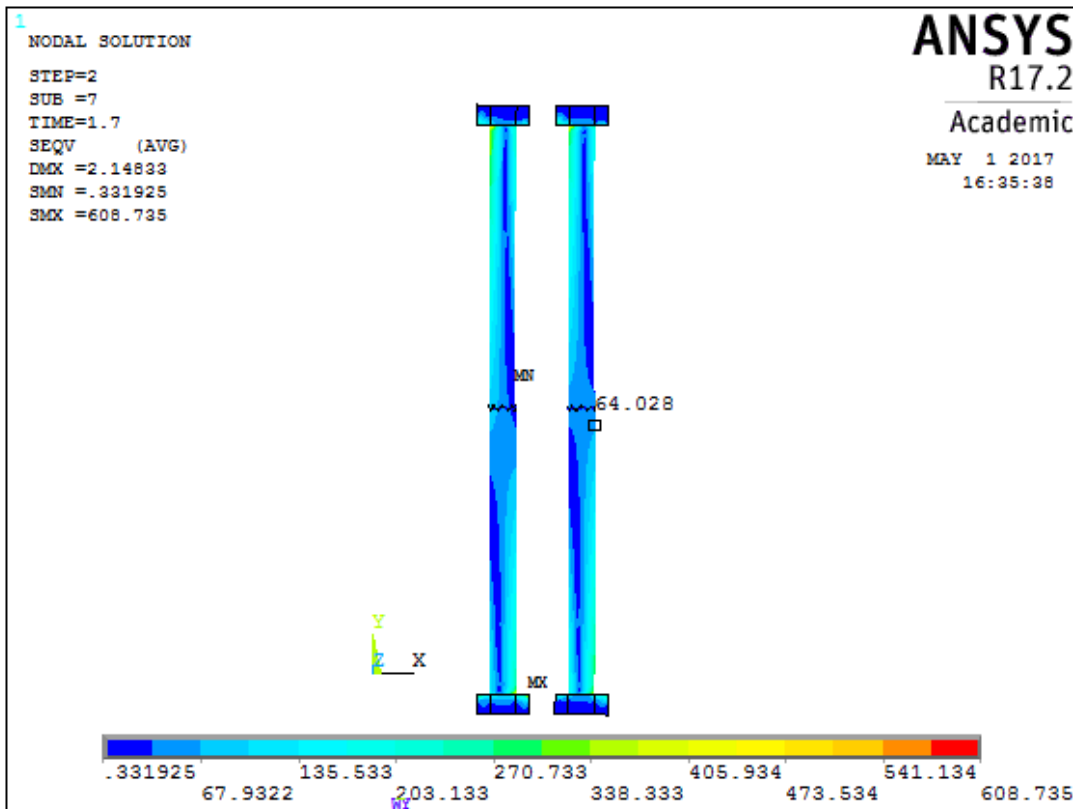


Figure 5.29 von Mises stress in the vertical bolts in case 5

Table 19

Von Mises comparison between case 1 and case 5

Reference points	S.EQV (N/mm ²)		% Reduction
	Case 1	Case 5	
1	401	277	31,0
2	267	189	29,1
3	138	106	23,6
4	329	232	29,5
5	153	116	23,9
6	34	23	31,2
7	195	77	60,8
Average in reduction			32,7

5.6 Case 6

Table 20

Moment rotation at different load steps for case 6

Load steps	Horizontal deflection of Beam (mm)	\varnothing (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,17	0,42	15,1	1,00
4	0,46	1,13	40,5	2,69
5	0,78	1,93	69,4	4,62
6	1,68	4,15	149,3	9,93
7	2,59	6,39	229,9	15,29

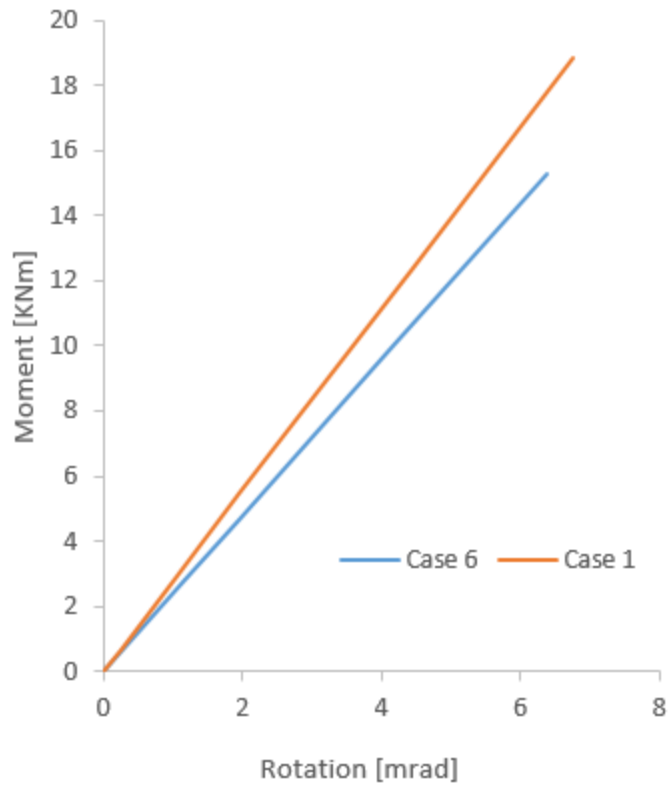


Figure 5.30 Comparison of moment-rotation curves

Table 21

Von Mises stress for top angle at different coordinates for case 6

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case 6
1	15	440	0	452
2	15	460	0	289
3	15	480	0	134
4	35	419	0	380
5	55	419	0	173
6	75	419	0	41
7	125	419	0	221

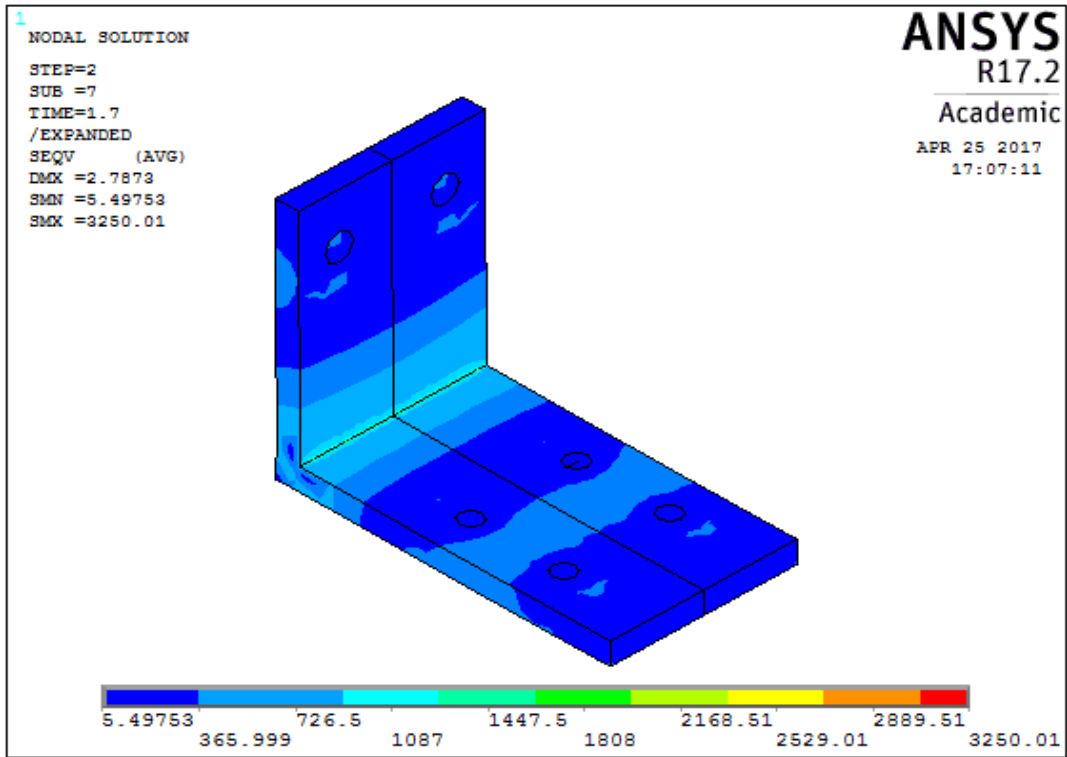


Figure 5.31 von Mises stress in the top angle in case 6

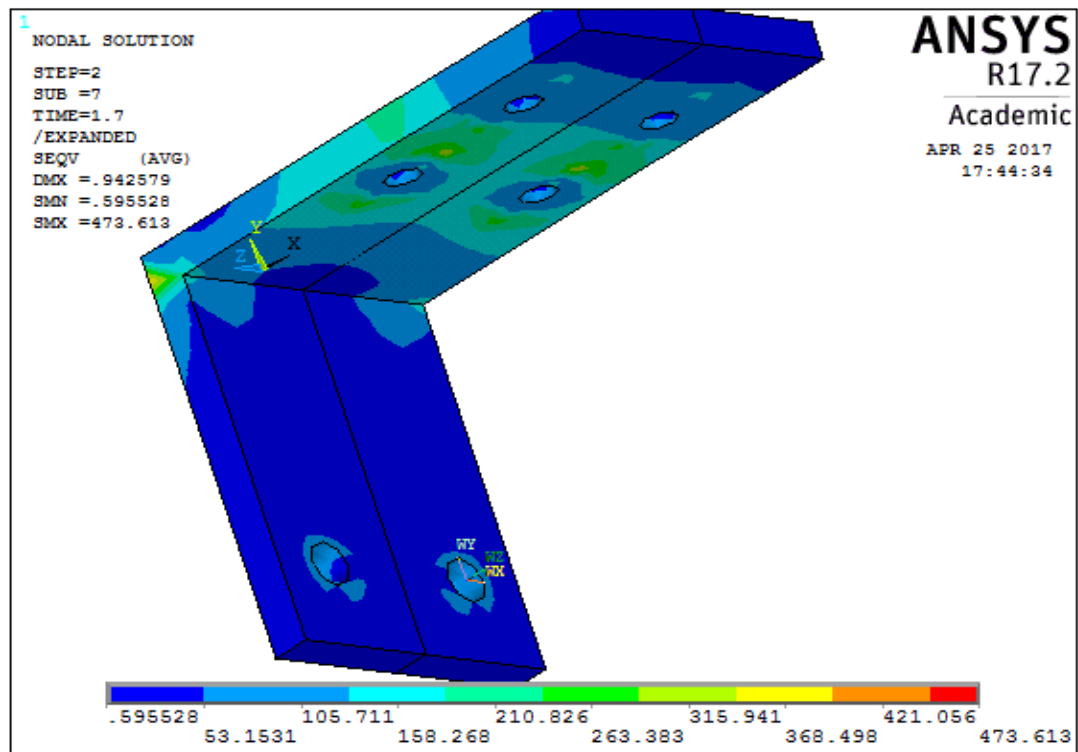


Figure 5.32 von Mises stress in the seat angle in case 6

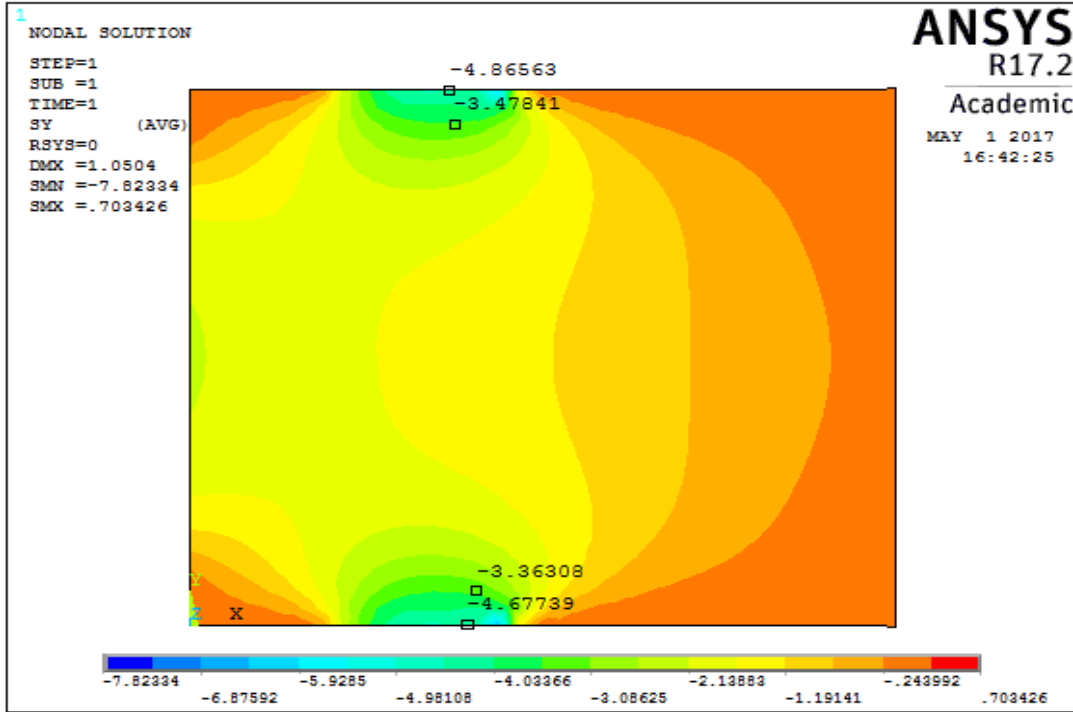


Figure 5.33 Stress perpendicular to the glulam beam at the first load step in case 6

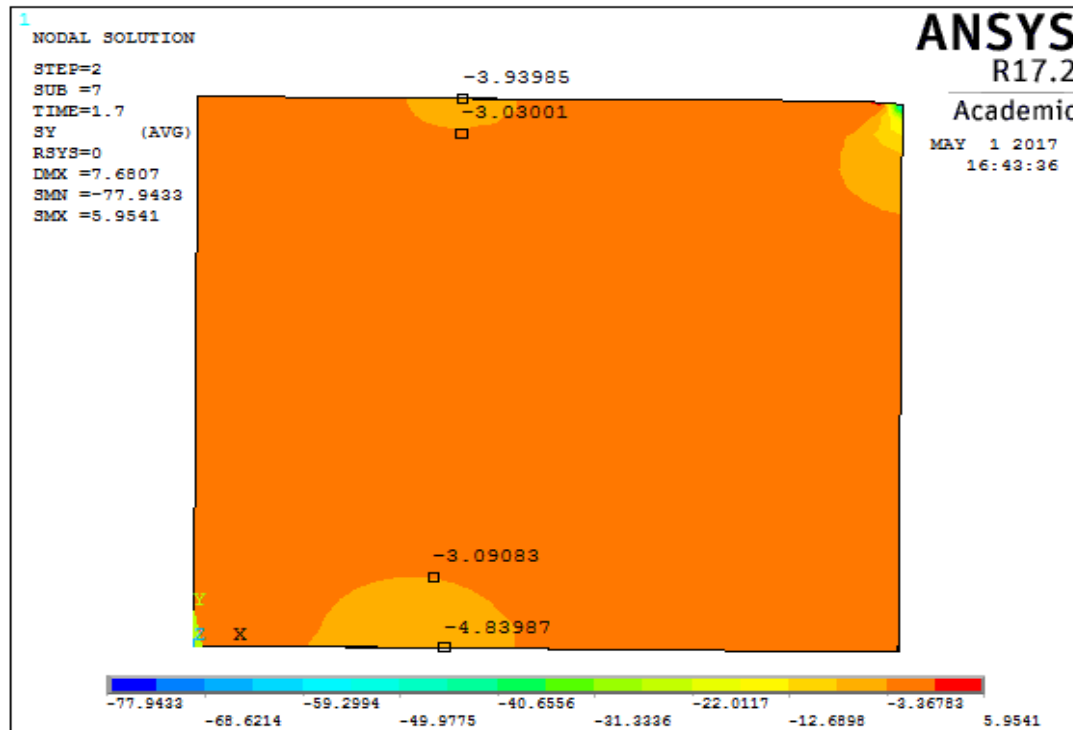


Figure 5.34 Stress perpendicular to the glulam beam at the second load step in case 6

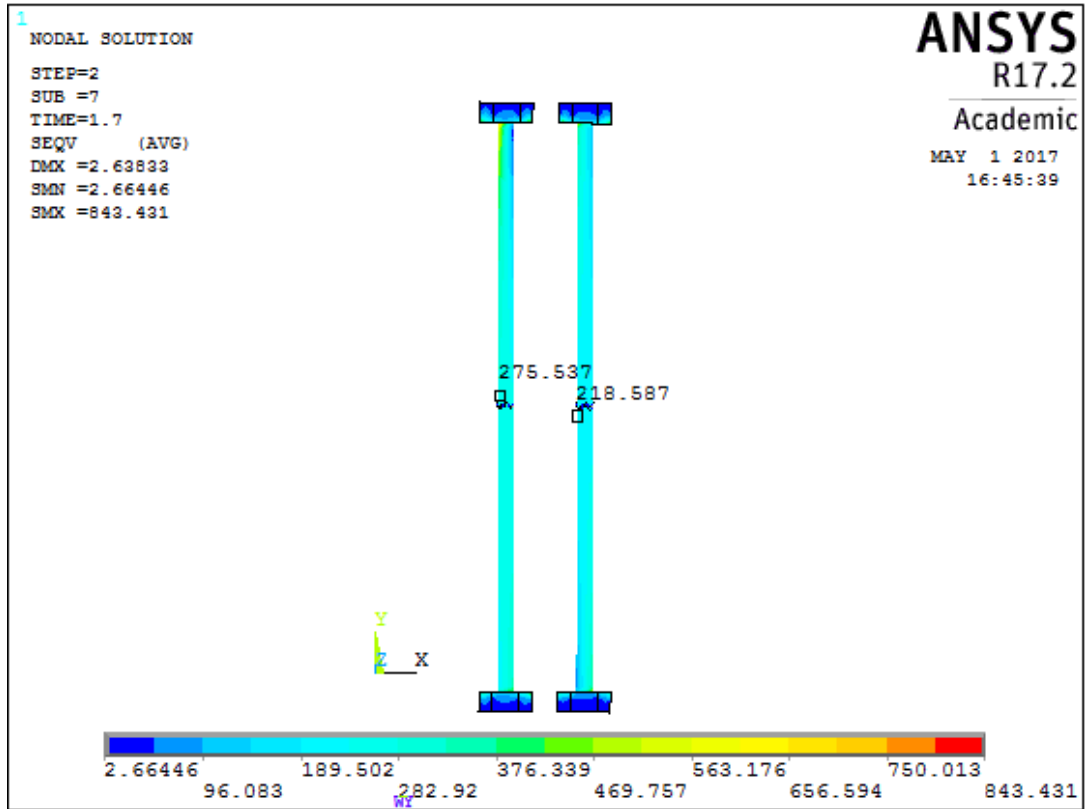


Figure 5.35 von Mises stress in the vertical bolts in case 6

Table 22

Von Mises comparison between case 1 and case 6

Reference points	S.EQV (N/mm ²)		% Increase
	Case 1	Case 6	
1	401	452	12,6
2	267	289	8,4
3	138	134	-2,6
4	329	380	15,4
5	153	173	13,2
6	34	41	22,0
7	195	221	13,3

Average in increment

11,8

5.7 Case 7

Table 23

Moment rotation at different load steps for case 7

Load steps	Horizontal deflection of Beam (mm)	\varnothing (mrad)	Pe(KN)	Moment (KNm)
0	0,00	0,00	0,0	0,00
3	0,20	0,49	31,8	2,12
4	0,38	0,94	60,5	4,02
5	1,15	2,84	182,9	12,17
6	1,91	4,72	303,9	20,21
7	2,66	6,57	423,2	28,14

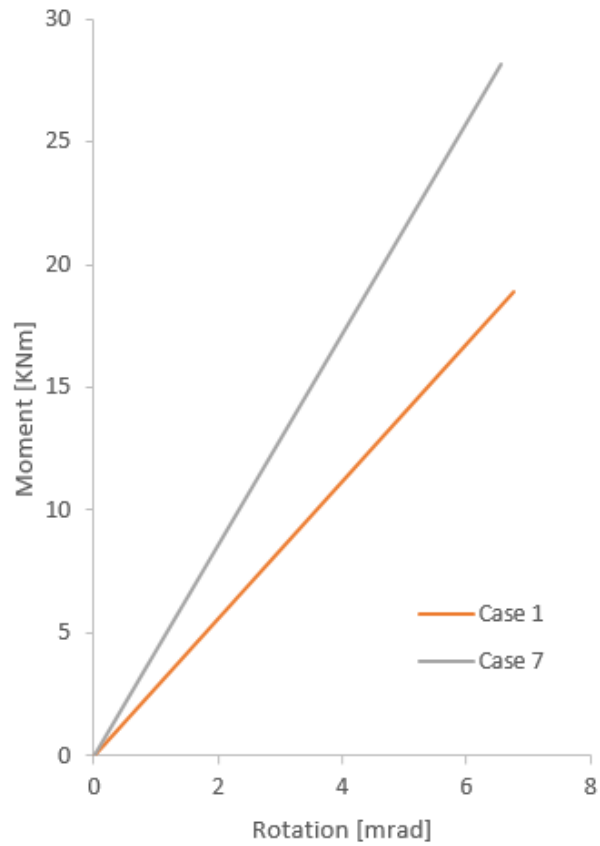


Figure 5.36 Comparison of moment-rotation curves

Table 24

Von Mises stress for top angle at different coordinates for case 7

Reference points	Global coordinates (mm)			S.EQV (N/mm ²)
	x	y	z	Case-7
1	12	500	0	179
2	12	511	0	213
3	12	521	0	245
4	12	540	0	329
5	12	550	0	372
6	12	560	0	406

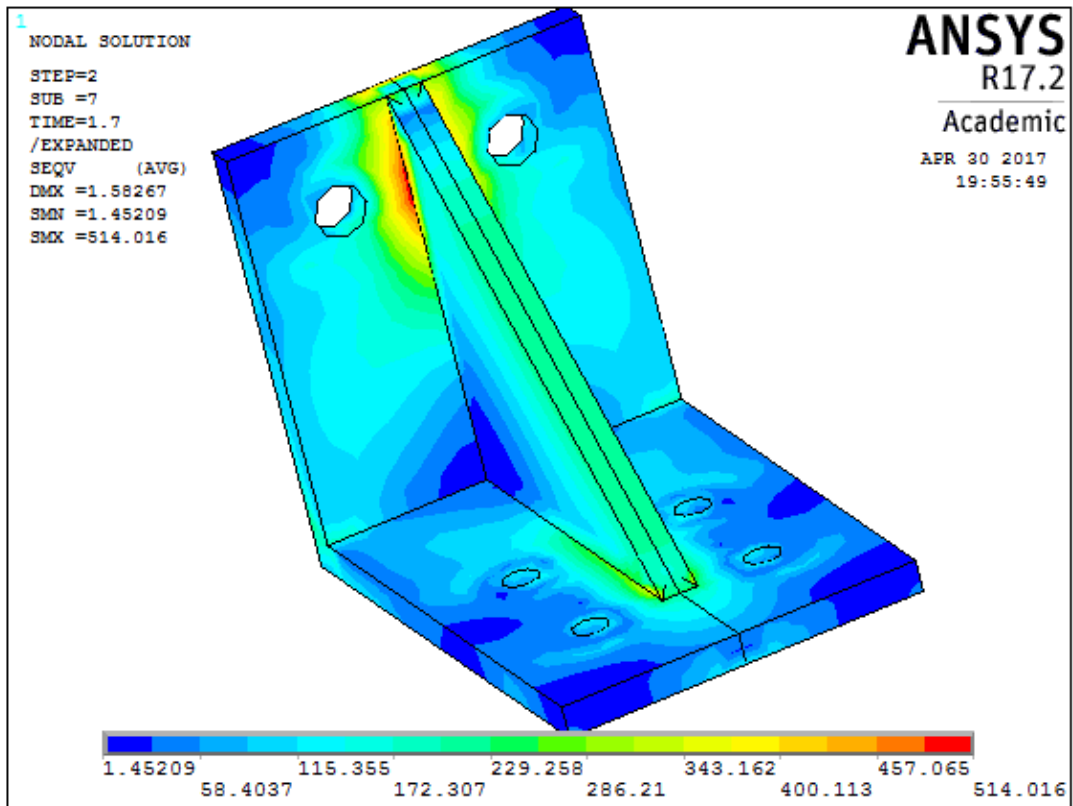


Figure 5.37 von Mises stress in the top angle in case 7

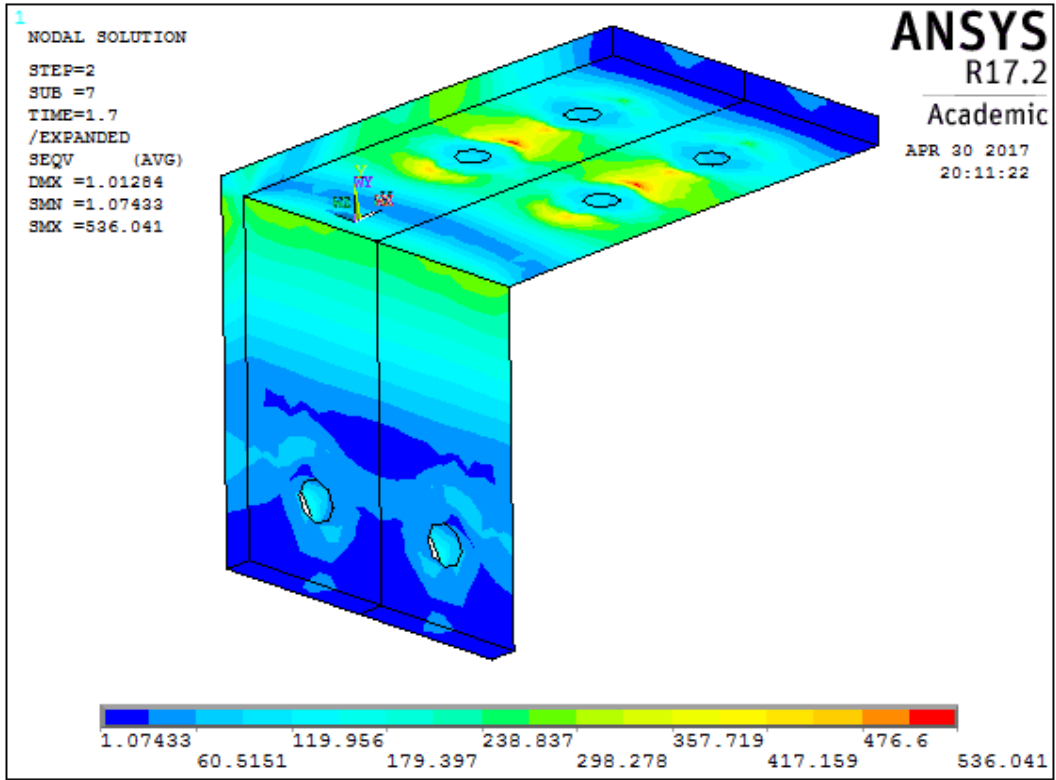


Figure 5.38 von Mises stress in the seat angle in case 7

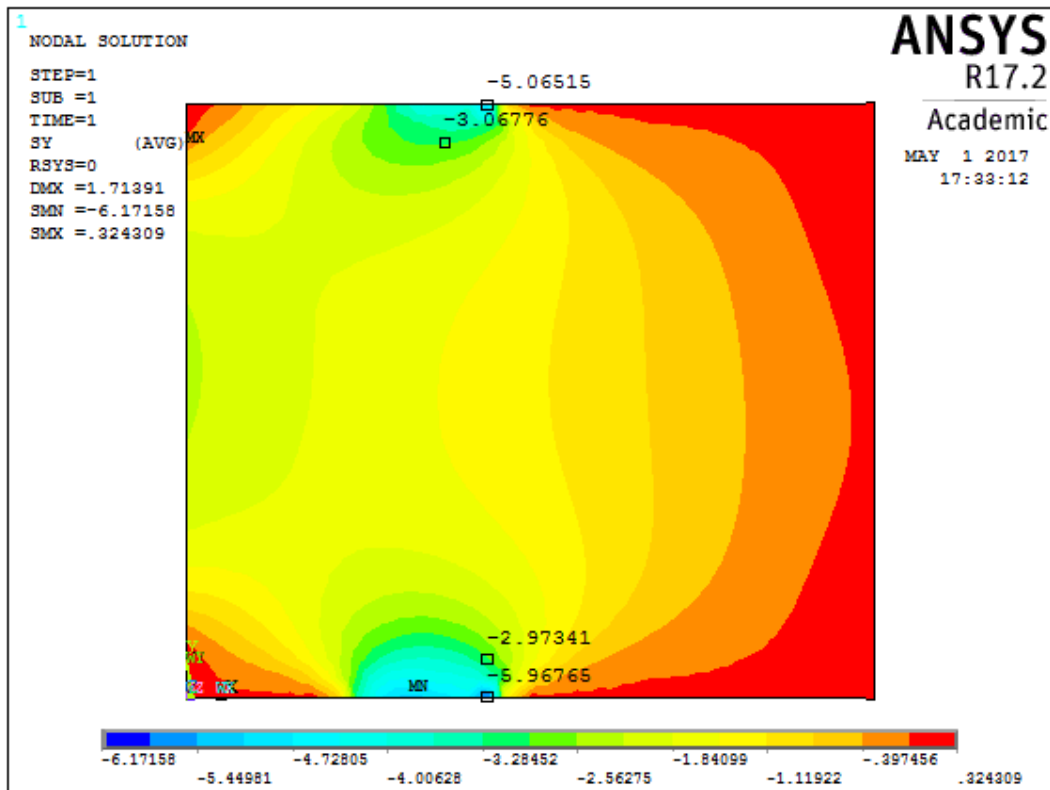


Figure 5.39 Stress perpendicular to the glulam beam at the first load step in case 7

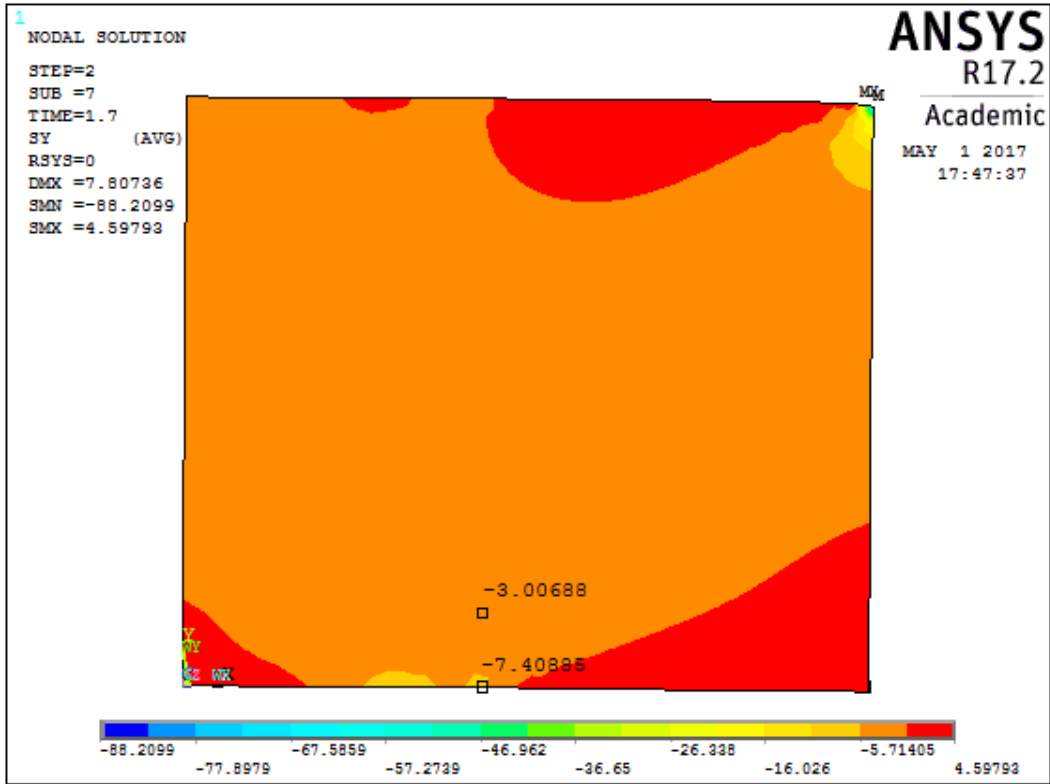


Figure 5.40 Stress perpendicular to the glulam beam at the second load step in case 7

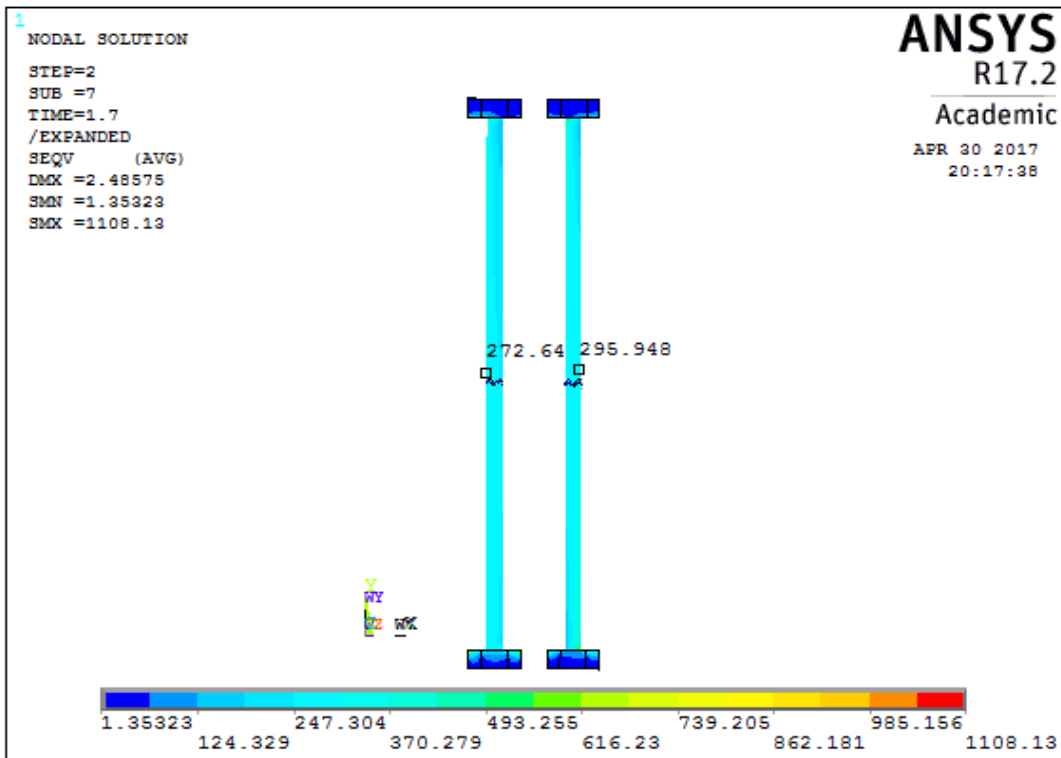


Figure 5.41 von Mises stress in the vertical bolts in case 7

5.8 Initial stiffness of connections

Table 25

Initial stiffness of connections

Case	$S_{j,in}$ (KNm/mrad)	$0.5 E I_b / L_b$ (KNm/mrad)	$8 E I_b / L_b$ (KNm/mrad)
1	5,470	1,17	18,76
2	5,478	1,17	18,76
3	5,481	1,17	18,76
6	5,46	1,17	18,76

Cases 4 and 5 has no change in the initial rotational stiffness. The evaluation of case 7 with stiffener is done with a simplification. Case 7 has 53.6% higher rotational stiffness than case 1, but it has a lower rotational stiffness than case 3. So the initial rotational stiffness for case 7 is between 5.47KNm/mrad-5.48KNm/mrad.

6 Discussion

Before running the analyses of the cases some results are expected according to engineering sense. For instance for every case involving increasing of the angle thickness, or adding stiffeners, the rotational stiffness of the connection is expected to rise due to the increased thickness. The question is the amount of the increase for the different cases and the effect of each parameter modification. Von Mises stresses are expected to decrease in the top and seat angle due to the addition steel. The amount of the decrease however needs to be investigated.

For each case the main discussion is related to the rotational stiffness of the connection and von Mises stress in the top angle. The seat angle, the glulam beam and the bolts connected to the beam are also discussed but not at the same extent. All case studies have been compared to case 1 which has been used as the base case study during the analysis.

6.1 Effect of changing the angle thickness

By varying the thickness of the top and the seat angles, the rotational stiffness of the connection changes significantly. The original connection had angles with a thickness of 15 mm and these were changed to 18 and 20 mm. As seen in figure 5.7 the moment resistance occurred in case 2 increases with 73% compared to the original connection. In case 3, a dramatic increase of 137% in rotational stiffness occurred in the connection as seen in figure 5.13. This indicates that by changing the thickness of the angles even with a small amount, a huge effect on the overall rotational stiffness for the connection is obtained. As expected the connection becomes stiffer when the thickness of the angles are increased. However the increase in the amount of steel in the angles are 21% and 35.2% for case 2 and 3, respectively. von Mises stress in the top angle is decreased with 12.2% in case 2 as seen in table 10 while as seen in table 13 a reduction of 15.5 % in von Mises stress was observed in the top angle in case 3. Since the volume of the angles becomes greater, the stresses distributed on them decreases compared to case 1. The reduction of von Mises stress between cases 2 and 3 compared to case 1 are minimal compared to the increment of the volume of steel for the two cases

6.2 Effect of changing the diameter of the bolts connected to the beam

When changing the diameter of the bolts as in cases 4 and 5, there were no increase observed in the rotational stiffness of the connection as seen in figure 5.24. This implies that changing the diameter of the bolts, are insignificant in this matter.

The reduction of von Mises stress in the top angle are presented in table 16 and table 19 for cases 4 and 5, respectively. In case 4 it is observed a reduction of 20.7 %, while in case 5 a reduction of 32.7% was observed. The volume of bolts however increases significantly. The decrease in von Mises stress in the top angles makes it possible for the angles to be subjected to some normal and shear stresses as well.

6.3 Effect of reducing the width of the angles

When reducing the width of the angle by 20 mm, a reduction of the rotational stiffness in the connection by 14.15 % was observed and the volume of steel in the angles was reduced by 14.28 %. As expected the stiffness of the connection is reduced. Von Mises stress in the top angle is increased with 11.8 % as seen in table 22. Generally this indicates that whenever the reduction of the top and seat angle are done the rotational stiffness decreases, while the stresses increases.

6.4 Effect of using a stiffener in the top angle

In case 7 the rotational stiffness of the connection increases with 53.6 % compared to case 1. The comparison of von Mises stress between the two cases is not accurate, and from figure 5.38 it is seen that some locations have values over the elastic limit. However a reduction of 10.4% in the volume of steel in the top angle is observed in case 7.

6.5 Seat angles

Effect of changing the angle thickness

In the cases 1, 2 and 3 the stresses that occurs in the seat angle reduces as the thickness of the angle increases. For cases 4 and 5 a reduction of von Mises stress was also observed. However in case study 6 the reduction of the width of the angle leads to higher von Mises stress. This is due to the amount of steel that the von Mises stress are distributed amongst. In case 7 at the top-end of the stiffener in the corner, high values are observed. These are most likely singularities, but high values are also observed in the seat angle in the middle of the angle under the bolt heads. This is because when the horizontal leg of the seat angle bends, pressure is applied on the seat angle from the bolt heads and causing these locally high stresses. The pretension in the first load step clamps the top and seat angle together towards the center of the bolts. Since the top angle are much stiffer than the seat angle, a higher deformation is done in the seat angle. This results in higher stresses in a few locations in the seat angle than for the top angle.

6.6 Wood failure

In all cases wood failure occurs in the glulam beam. When the pretension are applied to the bolts, the wood crushes because of compression of the top and seat angle. Then when the external load is applied to the beam the compression increases in the lower part of the beam when it is pushed towards the seat angle, causing even more crushing. Wood crushing in the top angle does not increase because the beam is bent away from the angle when the external load is applied. The beam has a capacity of only 3.6 N/mm² perpendicular to the grain, while in the longitudinal direction it has a capacity of 33 N/mm². The highest stress perpendicular to the grain was observed in case 3 and is approximately 8.28 N/mm². The high stresses however do only penetrate the beam section by a small amount. The stress perpendicular to the grain in case 2 increases compared to case 1, but not as extensively as for case 3. This indicates that the stresses perpendicular to the grain that occurs in the beam becomes larger when the angles are stiffer. This is most likely because whenever an angle is flexible, the beam deflects with the angle. On the other hand when the angle is very stiff the beam deflects mostly alone, resulting in a higher pressure from the angle. However

solving the problem with wood failure is not the main goal of this thesis. A presentation of the wood failure is only due to show what level of perpendicular stress that occurs, and the reason for this. Solving this problem may be taken as a continuation of this thesis, if desirable.

6.7 Vertical bolts

For cases 1, 2 and 3 the changes in the von Mises stress in the bolts are negligible. In cases 4 and 5 however the pretension and the bolt diameter change. While the diameter for the bolts increases, the pretension load decreases. Because the force is lower and distributed is on a smaller area than the previous cases the bolts are subjected to lower von Mises stress. It is only when adding a stiffener that a large increase of the von Mises stress are found in the bolts. When the pretension is applied, the stiffener at the top angle resist the clamping from the bolts much more than for the cases without stiffeners. This causes von Mises stress in the bolt to increase.

6.8 Initial rotational stiffness

For all cases the theoretical rotational stiffness are calculated. For cases 4 and 5 the initial rotational stiffness remains unchanged while some minor differences were found between the other cases. By using the criteria described in chapter 2.3.3, all the cases are analytically defined as semi rigid as seen in table 25.

After reviewing the results, the following observations is done.

- The rotational stiffness of the connection is at its highest for case 2, 3 and 7.
- Von Mises stress in the top angle is reduced the most in case 5.
- Changing the diameter of the bolts have no influence over the rotational stiffness of the connection
- All connections are defined as semi-rigid

Recommendation

To make an optimal connection, a combination between cases 3 and 5 is suggested. Combining the rotational stiffness of the connection in case 3 with the reduction in von Mises stress in the top angle with 20.2% gives an overall good connection. The difference in rotational stiffness of the connection between cases 3 and 2 is 67%. Even though the amount of steel in the angles for case 3 increases with 35.2% compared to case 1, it only increases 15.2% compared to case 2. This increase in steel volume is insignificant compared to the increase in rotational stiffness of the connection and the decreasing of von Mises stress. Therefore the combination between case 3 and 5 are highly recommended.

Table 26

Von Mises comparison between case 1 and the combined case

Reference points	S.EQV (N/mm ²)		% Reduction
	Case 1	Combined case	
1	401	237	40,9
2	267	174	34,8
3	138	112	18,8
4	329	220	33,2
5	153	138	9,9
6	34	63	-84,0
7	195	24	87,7
Average in reduction			20,2

7 Conclusion

The main conclusions that can be extracted from the analyses are:

- The behavior of the hybrid connection is predicted by using a numerical model. The use of a script based model allows engineers to easily change several parameters in order to investigate different characteristics of the connection
- By changing the thickness of the top and seat angles to 18 and 20 mm, the rotational stiffness of the connections increased by 73% and 137 % respectively. Von Mises stress in the top angle is reduced by 12.2% when using 18 mm angles and by 15.5% when using 20 mm angles.
- The rotational stiffness of the connections using different bolt diameters remains unchanged. In case 4, von Mises stress in the top angle is reduced by 20.7%, while the highest reduction of von Mises stress in the top angle was observed in case 5, with a reduction of 32.7%
- In case 6, a reduction of the rotational stiffness in the connection by 14.15% is observed. Von Mises stress in the top angle was increased by 11.8%.
- By analyzing the initial rotational stiffness, all cases are defined as semi-rigid.
- Wood failure due to compression from the top and seat angle is evident in all cases, but the von Mises stress does only penetrate the beam surface by a small amount. It is further proven that the more stiff a connection is, the higher the amount of crushing is observed.
- By combining cases 3 and 5 the optimal connection is constructed. By using top and seat angles with a thickness of 20 mm combined with 20 mm bolts connected to the beam, the best result has been monitored. An increment of the rotational stiffness of the connection by 137 %, as well as, a reduction in von Mises stress in the top angle by 20.2% is observed. Therefore, a combination between cases 3 and 5 is the recommended solution and this is the main contribution of this thesis to the research of hybrid connection between a tubular steel column, and a glulam beam.

Further work suggestion

Here some suggestions of continuations of the work done in this thesis are presented.

- The connection was constructed with elastic material properties and it was investigated in the elastic zone. Modelling the hybrid connection with plastic properties and further analyzing it in the plastic zone, is highly suggested.
- Failure of the wood has been observed and this problem needs to be solved. Modelling the connection with different plate-solutions that distributes the forces over a larger area of the beam should be investigated.
- Since this thesis have investigated the connection using standard bolts with nuts, modelling the connection with blind bolts connected to the tubular steel column to find the exact behavior of the blind bolt is suggested.

BIBLIOGRAPHY

- [1] Ansys. Ansys Mechanical, release 17.2, Help System, Basic Analysis Guide
- [2] Ansys. Ansys Mechanical, release 17.2, Help system, Contact Technology Guide
- [3] Ansys. Ansys Mechanical, release 17.2, Help System, Element reference Library
- [4] Ansys. Ansys Mechanical, release 17.2, Help System, Theory Reference
- [5] Bathe, Klaus-Jurgen. (2014). Finite Element Procedures. 2. Edition. Massachusetts, USA: Prentice Hall, Pearson Education, Inc.
- [6] CEN.EN 1993-1-8. Eurocode 3: Design of steel structures, Part 1-8: Design of joints; 2005
- [7] CEN.EN 1995-1-1. Eurocode 5: Design of timber structures, Part 1-1: General-common rules and rules for buildings; 2004
- [8] David Brown, David Iles, Mary Brittle and Abdul Malik. (2013). Moment-Resisting Joints to Eurocode 3 (P398). 2 Edition. London: The Steel Construction Institute and The British Constructional steelwork Association
- [9] Esdep-wg 02, Applied Metallurgy, available from <http://fgg-web.fgg.uni-lj.si/~pmoze/ESDEP/master/wg02> (read March, 2017)
- [10] Esdep-wg 11, Connection Design: Static loading, available from <http://fgg-web.fgg.uni-lj.si/~pmoze/ESDEP/master/wg11> (read March, 2017)
- [11] Esdep-wg 14, Structural System: Buildings, available from <http://fgg-web.fgg.uni-lj.si/~pmoze/ESDEP/master/wg14> (read April, 2017)
- [12] HANDBOOK 1 Timber Structures, available from http://fast10.vsb.cz/temtis/documents/handbook1_final.pdf (read April.2017)
- [13] Jack Porteous, Abdy Kermani. (2013). Structural Timber design to Eurocode 5. 2 Edition. Edinburgh: Blackwell
- [14] Karagiannis V, Malaga-Chuquitaype C, Elghazouli AY. (2016). Behaviour of hybrid timber beam-to-column moment connections. Engineering structures 2017; 131; 243-263
- [15] Lee Sang-Sup, Moon Tae-Sup. Moment rotation model of semi rigid connection with angles. Engineering structures 2002; 24; 227-237
- [16] Malaga-Chuquitaype C, Elghazouli AY. Component-based mechanical models for blind-bolted angle connections. Engineering structures 2010; 32; 3048-67
- [17] Anwar Naveed, Najam Fawad Ahmad. (2017). Structural Cross-Sections: Analysis and Design. 1. Edition. Asian Institute of Technology, Thailand: Elsevier Inc.
- [17] Torque and tension in bolts, available from http://www.efunda.com/designstandards/screws/fasteners_intro.cfm (read 3.4.2017)

APPENDIX A

!-----The Original Model with 15mm Angle (Case 1)-----

FINISH

/clear all

/PREP7

!-----

! properties

!-----

ET,1,solid187

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC

ET,3,Targe170 ! TARGE170 ELEMENTS

mp,ex,1,210000 ! For steel

mp,prxy,1,.3

MP,MU,1,.7

mp,prxy,2,.2

MP,MU,2,.7

!*

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDE,NUXY,2

MPDE,NUYZ,2

MPDE,NUXZ,2

MPDE,PRXY,2

MPDE,PRYZ,2

MPDE,PRXZ,2

MPDE,EX,2

MPDATA,EX,2,,12100 ! For glulam

MPDATA,EY,2,,430

MPDATA,EZ,2,,430

MPDATA,PRXY,2,,0.2

MPDATA,PRYZ,2,,0.2

MPDATA,PRXZ,2,,0.2

MPDATA,GXY,2,,917

MPDATA,GYZ,2,,917

MPDATA,GXZ,2,,917

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,2,,431

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,1,,7800

!-----

! DATA INPUT

!-----

X0=0

X1=15

X11=99

X12=159

X2=215

X3=1875/4 ! 1/4 of beam length is taken

X_1=-10

X_2=-130

X_3=-150

Y0=405

Y1=15+Y0

Y2=131+Y0

Y3=165+Y0

Y4=500+Y0

Y_1=0

Z0=0

```

Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2          !upper PART OF THE ANGLE BRACKET
blc4,X0+15,Y0+15,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-165,Z2          !lower PART OF THE ANGLE BRACKET
blc4,X0+15,Y_1-15,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1              !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-3, , , ,-Y1-15
vsbv,all,2

CYL4,0,0,D0-4, , , ,-Y1-15    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-3, , , ,-Y1-15
vsbv,all,1

CYL4,0,0,D0-4, , , ,-Y1-15    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vsel,s, , , 1
vsel,a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3    !creating column
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (column)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1, , , ,-25        !making hole
vsbv,all,1
CYL4,0,0,D0-2, , , ,-25        !bolt with head
RPR4,6,0,0,20,0,15

```

wpoffs,,,-25
RPR4,6,0,0,28,0,-15
vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

wpoffs,,-(Y0+2*131)
CYL4,0,0,D0-1,,, ,25
vsbv,all,1
CYL4,0,0,D0-2 ,,,, ,25
RPR4,6,0,0,28,0,-15
wpoffs,,,25
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all

!making hol

!bolt with head

!.....
! mesh
!.....

ALLSEL,ALL !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

ALLSEL,ALL !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !column
vsel,s,,,2
vatt,1,,1
ESIZE,15,0,
MSHAPE,1,3D

MSHKEY,0
VMESH,all
ALLSEL,ALL

Asel,s,area,,89 !top and bottom area of A_bracket
nsla,s,1
NSEL,R,LOC,y,420,430
NREFINE,all,,2,0,1,1
ALLSEL,ALL
Asel,s,area,,56
nsla,s,1
NSEL,R,LOC,x,15,25
NREFINE,all,,2,0,1,1
ALLSEL,ALL

! _____ CONTACT PAIRS _____
! _____ 1 _____
ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170 ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

TYPE,2 !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48 !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,3 !Target element
REAL,2
mat,1
ALLSEL,ALL
Asel,s,area,,56 !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF

! _____
! _____ 2 _____

ET,4,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170 ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5

TYPE,4 !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47 !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,5 !Target element
REAL,4
mat,1
ALLSEL,ALL
Asel,s,area,,60 !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25


```

ESLN,S
ESURF
!
!_____3_____
ET,6,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170      ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6              !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72     !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7              !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56     !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____4_____
ET,8,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170      ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8              !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71     !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9              !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60     !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF
!
!_____5_____
ET,10,conta174     ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170     ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10             !contact element
REAL,10

```

```

mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!_____6_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS
KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!_____7_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF

```

```

!
!_____8_____
ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,17                !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF

!
!_____9_____
ET,18,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170         ! TARGE170 ELEMENTS

TYPE,18                !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all

!
!_____10_____
ET,20,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170         ! TARGE170 ELEMENTS

TYPE,20                !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,21                !Target element

```

```

REAL,20
mat,1
ALLSEL,ALL
Asel,s,area,,59      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____11_____
ET,22,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170      ! TARGE170 ELEMENTS
allsel,all
TYPE,22              !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23              !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,24,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170      ! TARGE170 ELEMENTS

TYPE,24              !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25              !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!_____13_____
ET,26,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170      ! TARGE170 ELEMENTS

TYPE,26              !contact element
REAL,26

mat,1
allsel,all
Asel,s,area,,110     !top and bottom area of A bracket

```

```

nsla,s,1      !bolt head upper
ESLN,S
ESURF

TYPE,27      !Target element
REAL,26
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems      !pretension meshing
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!_____
finish
/solut
antype,0
ngeom,1

time,1      ! Sets time at end of run to 1 sec
autots,on      ! Auto time-stepping on
nsubst,1,1,1      ! Number of sub-steps
outres,all,1      ! Write all output
neqit,300

nsel,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsel,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all      ! constrain
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all      ! making symmetry
asel,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 27000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 39000,1,2
*ENDDO

lswrite,1

time,1      ! Sets time at end of run to 1 sec

```

```
autots,on          ! Auto time-stepping on
nsubst,10,10,10   ! Number of sub-steps
outres,all,1      ! Write all output
neqit,300
```

```
ALLSEL,ALL
nset,s,loc,x,X3   ! Apply load
nset,r,loc,y,Y0
D,all,-10,,,,UY,,,,
!f,all,fy,-10000
allsel,all
!swrite,2
!ssolve,1,2,1
```

APPENDIX B

```
!-----The Model with 18mm Angle(Case 2)-----
FINISH
/clear all
/PREP7
!-----
!          properties
!-----
ET,1,solid187
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170        ! TARGE170 ELEMENTS
mp,ex,1,210000      !For steel
mp,prxy,1,.3
MP,MU,1,.7
mp,prxy,2,.2
MP,MU,2,.7
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDE,NUXY,2
MPDE,NUYZ,2
MPDE,NUXZ,2
MPDE,PRXY,2
MPDE,PRYZ,2
MPDE,PRXZ,2
MPDE,EX,2
MPDATA,EX,2,,12100 !For glulam
MPDATA,EY,2,,430
MPDATA,EZ,2,,430
MPDATA,PRXY,2,,0.2
MPDATA,PRYZ,2,,0.2
MPDATA,PRXZ,2,,0.2
MPDATA,GXY,2,,917
MPDATA,GYZ,2,,917
MPDATA,GXZ,2,,917
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,431
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,7800

!-----
!          DATA INPUT
!-----
X0=0
X1=18
X11=99
X12=159
X2=218
X3=1875/4! 1/4 of beam length is taken
X_1=-10
X_2=-130
X_3=-150

Y0=405
Y1=18+Y0
Y2=131+3+Y0
Y3=165+3+Y0
Y4=500+Y0

Y_1=0
```

```

Z0=0
Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2          !upper PART OF THE ANGLE BRACKET
blc4,X0+18,Y0+18,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-165,Z2        !lower PART OF THE ANGLE BRACKET
blc4,X0+18,Y_1-18,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1              !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-3, , , ,-Y1-18
vsbv,all,2

CYL4,0,0,D0-4, , , ,-Y1-18    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-18
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-3, , , ,-Y1-18
vsbv,all,1

CYL4,0,0,D0-4, , , ,-Y1-18    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-18
RPR4,6,0,0,20,0,-15
vsel, , , , 1
vsel, a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3    !creating column
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (colum)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1, , , ,-28        !making hole
vsbv,all,1

```



```

CYL4,0,0,D0-2 ,,, ,-28      !bolt with head
RPR4,6,0,0,20,0,15
wpoffs,,,-28
RPR4,6,0,0,28,0,-15
vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

```

```

wpoffs,,-(Y0+2*134)
CYL4,0,0,D0-1 ,,, ,28      !making hol
vsbv,all,1
CYL4,0,0,D0-2 ,,, ,28      !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,28
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all
!.....
!                mesh
!.....

```

```

ALLSEL,ALL      !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

```

```

ALLSEL,ALL      !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !column
vsel,s,,,2
vatt,1,,1

```

```

ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL
Asel,s,area,,89
nsla,s,1
nset,r,loc,y,420+3,430+3
nrefine,all,,,2,0,1,1
allsel,all
Asel,s,area,,56
nsla,s,1
nset,r,loc,x,15+3,25+3
nrefine,all,,,2,0,1,1
allsel,all

```

```

! _____ CONTACT PAIRS _____
! _____ 2 _____
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170         ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

```

```

TYPE,2                !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48      !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

```

```

TYPE,3                !Target element
REAL,2
mat,1
ALLSEL,ALL
Asel,s,area,,56      !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF

```

```

! _____
! _____ 3 _____

```

```

ET,4,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170         ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5

```

```

TYPE,4                !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47      !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

```

```

TYPE,5                !Target element
REAL,4
mat,1
ALLSEL,ALL
Asel,s,area,,60      !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25

```

```

ESLN,S
ESURF
!
!_____4_____
ET,6,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170     ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6             !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72   !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7             !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56   !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____5_____
ET,8,conta174     ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170    ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8             !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71   !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9             !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60   !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF
!
!_____6_____
ET,10,conta174    ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170   ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10            !contact element
REAL,10

```

```

mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____7_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____8_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156

```

```

ESLN,S
ESURF
!
!_____9_____
ET,16,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170     ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16             !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123    !top and bottom area of A_bracket
nsla,s,1            !bolt head upper
ESLN,S
ESURF

TYPE,17             !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,   !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____10_____
ET,18,conta174     ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170    ! TARGE170 ELEMENTS

TYPE,18             !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51     !top and bottom area of A_bracket
nsla,s,1            !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19             !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55     !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____11_____
ET,20,conta174     ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170    ! TARGE170 ELEMENTS

TYPE,20             !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52     !top and bottom area of A_bracket
nsla,s,1            !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

```

```

TYPE,21                !Target element
REAL,20
mat,1
ALLSEL,ALL
Asel,s,area,,59       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170        ! TARGE170 ELEMENTS
allsel,all
TYPE,22                !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23                !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170        ! TARGE170 ELEMENTS

TYPE,24                !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5        !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25                !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!
!_____14_____
ET,26,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170        ! TARGE170 ELEMENTS

TYPE,26                !contact element
REAL,26

mat,1

```

```

allsel,all
Ase1,s,area,,110      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,27              !Target element
REAL,26
mat,1
ALLSEL,ALL
Ase1,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!
finish
/solut
antype,0
ngeom,1

time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,1,1,1         ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
asel,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,LOCK,FORC, 27000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,LOCK,FORC, 39000,1,2
*ENDDO

```

```
lswrite,1

time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,10,10,10      ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

ALLSEL,ALL
nset,s,loc,x,X3      ! Apply load
nset,r,loc,y,Y0
D,all,,-10,,,,UY,,,,
!f,all,fy,-10000
allsel,all
lswrite,2
lssolve,1,2,1
```


APPENDIX c

```
!-----The Model with 20mm Angle (Case 3)-----
FINISH
/clear all
/PREP7
!-----
!          properties
!-----
ET,1,solid187
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170        ! TARGE170 ELEMENTS
mp,ex,1,210000      !For steel
mp,prxy,1,.3
MP,MU,1,.7
mp,prxy,2,.2
MP,MU,2,.7
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDE,NUXY,2
MPDE,NUYZ,2
MPDE,NUXZ,2
MPDE,PRXY,2
MPDE,PRYZ,2
MPDE,PRXZ,2
MPDE,EX,2
MPDATA,EX,2,,12100      !For glulam
MPDATA,EY,2,,430
MPDATA,EZ,2,,430
MPDATA,PRXY,2,,0.2
MPDATA,PRYZ,2,,0.2
MPDATA,PRXZ,2,,0.2
MPDATA,GXY,2,,917
MPDATA,GYZ,2,,917
MPDATA,GXZ,2,,917
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,431
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,7800

!-----
!          DATA INPUT
!-----
X0=0
X1=20
X11=99
X12=159
X2=215
X3=1875/4! 1/4 of beam length is taken
X_1=-10
X_2=-130
X_3=-150

Y0=405
Y1=20+Y0
Y2=131+5+Y0
Y3=165+5+Y0
Y4=500+Y0

Y_1=0
```

```

Z0=0
Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2          !upper PART OF THE ANGLE BRACKET
blc4,X0+20,Y0+20,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-165,Z2        !lower PART OF THE ANGLE BRACKET
blc4,X0+20,Y_1-20,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1              !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-3, , , ,-Y1-20
vsbv,all,2

CYL4,0,0,D0-4, , , ,-Y1-20    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-20
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-3, , , ,-Y1-20
vsbv,all,1

CYL4,0,0,D0-4, , , ,-Y1-20    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-20
RPR4,6,0,0,20,0,-15
vsel, , , , 1
vsel, a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3    !creating column
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (column)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1, , , ,-30        !making hole
vsbv,all,1

```

```

CYL4,0,0,D0-2 ,,, ,-30      !bolt with head
RPR4,6,0,0,20,0,15
wpoffs,,,-30
RPR4,6,0,0,28,0,-15
vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

```

```

wpoffs,,-(Y0+2*134)
CYL4,0,0,D0-1,,, ,30      !making hol
vsbv,all,1
CYL4,0,0,D0-2 ,,, ,30      !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,30
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all
!.....
!                mesh
!.....

```

```

ALLSEL,ALL      !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

```

```

ALLSEL,ALL      !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !column
vsel,s,,,2
vatt,1,,1

```

ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

Asel,s,area,,89 !top and bottom area of A_bracket

nsla,s,1
NSEL,R,LOC,y,420,430
NREFINE,all,,2,0,1,1
ALLSEL,ALL
Asel,s,area,,56
nsla,s,1
NSEL,R,LOC,x,15,25
NREFINE,all,,2,0,1,1
ALLSEL,ALL

! _____ CONTACT PAIRS _____
! _____ 2 _____

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170 ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

TYPE,2 !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48 !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,3 !Target element
REAL,2
mat,1
ALLSEL,ALL
Asel,s,area,,56 !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF

! _____
! _____ 3 _____

ET,4,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170 ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5

TYPE,4 !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47 !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,5 !Target element
REAL,4
mat,1
ALLSEL,ALL
Asel,s,area,,60 !bottom area of lower A_bracket
nsla,s,1

```

NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
!
!_____
!_____4_____
ET,6,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170          ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6                  !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72        !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7                  !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56        !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____
!_____5_____
ET,8,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170          ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8                  !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71        !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9                  !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60        !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF

!
!_____
!_____6_____
ET,10,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170        ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10                 !contact element

```

```

REAL,10
mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____7_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____8_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,      !top area of A_bracket
nsla,s,1

```

```

NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____9_____
ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,17                !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____10_____
ET,18,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170       ! TARGE170 ELEMENTS

TYPE,18                !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____11_____
ET,20,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170       ! TARGE170 ELEMENTS

TYPE,20                !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S

```

```

ESURF

TYPE,21                !Target element
REAL,20
mat,1
ALLSEL,ALL
Asel,s,area,,59       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170        ! TARGE170 ELEMENTS
allsel,all
TYPE,22                !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23                !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170        ! TARGE170 ELEMENTS

TYPE,24                !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5        !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25                !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!
!_____14_____
ET,26,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170        ! TARGE170 ELEMENTS

TYPE,26                !contact element
REAL,26

```



```

mat,1
allsel,all
Asel,s,area,,110      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,27              !Target element
REAL,26
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
! _____
finish
/solut
antype,0
nlgeom,1

time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,1,1,1         ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
asel,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 27000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 39000,1,2
*ENDDO
lswrite,1

```

```
time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,10,10,10      ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

ALLSEL,ALL
nset,s,loc,x,X3      ! Apply load
nset,r,loc,y,Y0
D,all,,-10,,,,UY,,,,
allsel,all
lswrite,2
lssolve,1,2,1
```

APPENDIX D

!-----The Model with 15mm Angle and 16mm vertical bolt (Case 4)-----

FINISH

/clear all

/PREP7

!-----

! properties

!-----

ET,1,solid187

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC

ET,3,Targe170 ! TARGE170 ELEMENTS

mp,ex,1,210000 !For steel

mp,prxy,1,.3

MP,MU,1,.7

mp,prxy,2,.2

MP,MU,2,.7

!*

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDE,NUXY,2

MPDE,NUYZ,2

MPDE,NUXZ,2

MPDE,PRXY,2

MPDE,PRYZ,2

MPDE,PRXZ,2

MPDE,EX,2

MPDATA,EX,2,,12100 !For glulam

MPDATA,EY,2,,430

MPDATA,EZ,2,,430

MPDATA,PRXY,2,,0.2

MPDATA,PRYZ,2,,0.2

MPDATA,PRXZ,2,,0.2

MPDATA,GXY,2,,917

MPDATA,GYZ,2,,917

MPDATA,GXZ,2,,917

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,2,,431

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,1,,7800

!-----

! DATA INPUT

!-----

X0=0

X1=15

X11=99

X12=159

X2=215

X3=1875/4 ! 1/4 of beam is taken

X_1=-10

X_2=-130

X_3=-150

Y0=405

Y1=15+Y0

Y2=131+Y0

Y3=165+Y0

Y4=500+Y0

```

Y_1=0

Z0=0
Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!           GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2           !upper PART OF THE ANGLE BRACKET
blc4,X0+15,Y0+15,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2           !BEAM

blc4,X0,Y_1,X2,-165,Z2         !lower PART OF THE ANGLE BRACKET
blc4,X0+15,Y_1-15,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1               !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-1, , , ,-Y1-15
vsbv,all,2

CYL4,0,0,D0-2, , , ,-Y1-15     !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoofs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1               !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-1, , , ,-Y1-15
vsbv,all,1

CYL4,0,0,D0-2, , , ,-Y1-15     !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoofs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vsel, s, , , 1
vsel, a, , , 4, 6, 2
vadd, all
allsel, all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3     !creating column
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (column)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002

```

```

CYL4,0,0,D0-1 , , , -25      !making hole
vsbv,all,1
CYL4,0,0,D0-2 , , , -25      !bolt with head
RPR4,6,0,0,20,0,15
wpoffs, , , -25
RPR4,6,0,0,28,0, -15
vsel, s, , , 1
vsel, a, , , 3,6,3
vadd,all
allsell,all

wpoffs, , -(Y0+2*131)
CYL4,0,0,D0-1, , , , 25      !making hol
vsbv,all,1
CYL4,0,0,D0-2 , , , ,25      !bolt with head
RPR4,6,0,0,28,0, -15
wpoffs, , , 25
RPR4,6,0,0,20,0,15
vsel, s, , , 1
vsel, a, , , 4,8,4
vadd,all
allsell,all
numcmp,all
!.....
!                mesh
!.....

ALLSEL,ALL      !A_Bracket
vsel, s, , , 4,6,2
vatt,1, , 1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

Asel, s, area, , 89      !top and bottom area of A_bracket
nsla, s, 1
NSEL, R, LOC, y, 420, 430
NREFINE, all, , , 2, 0, 1, 1
ALLSEL, ALL
Asel, s, area, , 56      !top and bottom area of A_bracket
nsla, s, 1
NSEL, R, LOC, x, 15, 25
NREFINE, all, , , 2, 0, 1, 1
ALLSEL, ALL

ALLSEL, ALL      !Beam
vsel, s, , , 1
vatt, 2, , 1
ESIZE, 15, 0,
MSHAPE, 1, 3D
MSHKEY, 0
VMESH, all
ALLSEL, ALL

ALLSEL, ALL      !Bolt
vsel, s, , , 3, 5, 2
!vsel, a, , , 7, 8
vatt, 1, , 1
ESIZE, 5, 0,
MSHAPE, 1, 3D
MSHKEY, 0
VMESH, all
ALLSEL, ALL

```

```

ALLSEL,ALL          !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL          !column
vsel,s,,,2
vatt,1,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

! _____ CONTACT PAIRS _____
! _____ 2 _____
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170          ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

TYPE,2                !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48        !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,3                !Target element
REAL,2
mat,1
ALLSEL,ALL
Asel,s,area,,56        !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
! _____
! _____ 3 _____

ET,4,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170          ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5
TYPE,4                !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47        !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,5                !Target element
REAL,4
mat,1

```

```

ALLSEL,ALL
Asel,s,area,,60          !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
!
!_____
!_____4_____
ET,6,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170          ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6                  !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72        !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7                  !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56        !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____
!_____5_____
ET,8,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170          ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8                  !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71        !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9                  !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60        !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF

!
!_____
!_____6_____
ET,10,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170        ! TARGE170 ELEMENTS

KEYOPT,10,2,2

```

```

KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10                !contact element
REAL,10
mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____
!_____7_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____
!_____8_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1

```



```

ALLSEL,ALL
Asel,s,area,,111,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____9_____
ET,16,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170      ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16              !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,17              !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____10_____
ET,18,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170      ! TARGE170 ELEMENTS

TYPE,18              !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19              !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____11_____
ET,20,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170      ! TARGE170 ELEMENTS

TYPE,20              !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52       !top and bottom area of A_bracket

```

```

nsla,s,1      !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,21      !Target element
REAL,20
mat,1
ALLSEL,ALL
Asel,s,area,,59      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170      ! TARGE170 ELEMENTS
allsel,all
TYPE,22      !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88      !top and bottom area of A_bracket
nsla,s,1      !bolt head upper
ESLN,S
ESURF

TYPE,23      !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170      ! TARGE170 ELEMENTS

TYPE,24      !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5      !top and bottom area of A_bracket
nsla,s,1      !bolt head upper
ESLN,S
ESURF

TYPE,25      !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!_____14_____
ET,26,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170      ! TARGE170 ELEMENTS

```

```

TYPE,26                !contact element
REAL,26

mat,1
allsel,all
Asel,s,area,,110      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,27                !Target element
REAL,26
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!_____
finish
/solut
antype,0
nlgeom,1

time,1                 ! Sets time at end of run to 1 sec
autots,on              ! Auto time-stepping on
nsubst,1,1,1          ! Number of sub-steps
outres,all,1          ! Write all output
neqit,300

nsl,s,loc,x,X_3        ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3        ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
asel,s,loc,z,0
DA,all,SYMM
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 20000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 39000,1,2

```

```
*ENDDO

lswrite,1

time,1                ! Sets time at end of run to 1 sec
autots,on            ! Auto time-stepping on
nsubst,10,10,10     ! Number of sub-steps
outres,all,1        ! Write all output
neqit,300

ALLSEL,ALL
nset,s,loc,x,X3      ! Apply load
nset,r,loc,y,Y0
D,all,,-10,,,,UY,,,,
allsel,all
lswrite,2
lssolve,1,2,1
```

APPENDIX E

!-----The Model with 15mm Angle and 20mmØ vertical bolt (Case 5)-----

```
FINISH
/clear all
/PREP7
!-----
!           properties
!-----
ET,1,solid187
ET,2,conta174           ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170         ! TARGE170 ELEMENTS
mp,ex,1,210000       !steel
mp,prxy,1,.3
MP,MU,1,.7
mp,prxy,2,.2
MP,MU,2,.7
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDE,NUXY,2
MPDE,NUYZ,2
MPDE,NUXZ,2
MPDE,PRXY,2
MPDE,PRYZ,2
MPDE,PRXZ,2
MPDE,EX,2
MPDATA,EX,2,,12100           !glulam
MPDATA,EY,2,,430
MPDATA,EZ,2,,430
MPDATA,PRXY,2,,0.2
MPDATA,PRYZ,2,,0.2
MPDATA,PRXZ,2,,0.2
MPDATA,GXY,2,,917
MPDATA,GYZ,2,,917
MPDATA,GXZ,2,,917
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,431
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,7800
!-----
!           DATA INPUT
!-----
X0=0
X1=15
X11=99
X12=159
X2=215
X3=1875/4! 1/2 of beam is taken
X_1=-10
X_2=-130
X_3=-150

Y0=405
Y1=15+Y0
Y2=131+Y0
Y3=165+Y0
Y4=500+Y0           !HEIGHT OF THE COLUM

Y_1=0

Z0=0
Z1=-34
Z1_2=-65
```

```

Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2          !upper PART OF THE ANGLE BRACKET
blc4,X0+15,Y0+15,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-165,Z2        !lower PART OF THE ANGLE BRACKET
blc4,X0+15,Y_1-15,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1              !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0+1, , , ,-Y1-15
vsbv,all,2

CYL4,0,0,D0, , , ,-Y1-15     !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0+1, , , ,-Y1-15
vsbv,all,1

CYL4,0,0,D0, , , ,-Y1-15     !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vsel,s, , , 1
vsel,a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1              !center of hole (column)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1, , , ,-25      !making hole
vsbv,all,1
CYL4,0,0,D0-2, , , ,-25      !bolt with head
RPR4,6,0,0,20,0,15
wpoffs, , , ,-25
RPR4,6,0,0,28,0,-15

```

```

vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

wpoffs,-(Y0+2*131)
CYL4,0,0,D0-1,,,,,25      !making hol
vsbv,all,1
CYL4,0,0,D0-2,,,,,25      !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,,25
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all
!.....
!              mesh
!.....

ALLSEL,ALL      !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

ALLSEL,ALL      !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL      !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL      !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL      !column
vsel,s,,,2
vatt,1,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all

```

ALLSEL,ALL

Asel,s,area,,89 !top and bottom area of A_bracket

nsla,s,1

NSEL,R,LOC,y,420,430

NREFINE,all,,2,0,1,1

ALLSEL,ALL

Asel,s,area,,56 !top and bottom area of A_bracket

nsla,s,1

NSEL,R,LOC,x,15,25

NREFINE,all,,2,0,1,1

ALLSEL,ALL

! _____ CONTACT PAIRS _____
! _____ 2 _____

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC

ET,3,Targe170 ! TARGE170 ELEMENTS

KEYOPT,2,2,2

KEYOPT,2,4,2

KEYOPT,2,12,5

TYPE,2 !contact element

REAL,2

mat,1

allsel,all

asel,s,area,,48 !outer bolt head and upper A_bracket

nsla,s,1

ESLN,S

ESURF

TYPE,3 !Target element

REAL,2

mat,1

ALLSEL,ALL

Asel,s,area,,56 !top area of upper A_bracket

nsla,s,1

NSEL,R,LOC,X,X12-25,X12+25

ESLN,S

ESURF

! _____
! _____ 3 _____

ET,4,conta174 ! CONTA173 ELEMENTS for bonded MPC

ET,5,Targe170 ! TARGE170 ELEMENTS

KEYOPT,4,2,2

KEYOPT,4,4,2

KEYOPT,4,12,5

TYPE,4 !contact element

REAL,4

mat,1

allsel,all

asel,s,area,,47 !outer bolt head lower A_bracket

nsla,s,1

ESLN,S

ESURF

TYPE,5 !Target element

REAL,4

mat,1

ALLSEL,ALL

Asel,s,area,,60 !bottom area of lower A_bracket

nsla,s,1

NSEL,R,LOC,X,X12-25,X12+25

ESLN,S

ESURF

! _____


```

! _____ 4 _____
ET,6,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170         ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6                 !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72       !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7                 !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56       !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
! _____
! _____ 5 _____
ET,8,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170         ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8                 !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71       !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9                 !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60       !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF

! _____
! _____ 6 _____
ET,10,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170        ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10                !contact element
REAL,10
mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket

```

```

nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____7_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102     !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,    !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____8_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124     !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,    !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____

```

```

! _____ 9 _____
ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                 !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
ESLN,S
ESURF

TYPE,17                 !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
! _____
! _____ 10 _____
ET,18,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170         ! TARGE170 ELEMENTS

TYPE,18                 !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                 !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
! _____
! _____ 11 _____
ET,20,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170         ! TARGE170 ELEMENTS

TYPE,20                 !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,21                 !Target element
REAL,20

```

```

mat,1
ALLSEL,ALL
Asel,s,area,,59      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170      ! TARGE170 ELEMENTS
allsel,all
TYPE,22              !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23              !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170      ! TARGE170 ELEMENTS

TYPE,24              !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25              !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!_____14_____
ET,26,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170      ! TARGE170 ELEMENTS

TYPE,26              !contact element
REAL,26

mat,1
allsel,all
Asel,s,area,,110     !top and bottom area of A_bracket
nsla,s,1              !bolt head upper

```

```

ESLN,S
ESURF

TYPE,27          !Target element
REAL,26
mat,1
ALLSEL,ALL
Asel,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!
finish
/solut
antype,0
nlgeom,1

time,1           ! Sets time at end of run to 1 sec
autots,on       ! Auto time-stepping on
nsubst,1,1,1    ! Number of sub-steps
outres,all,1    ! Write all output
neqit,300

nsl,s,loc,x,X_3  ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3  ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
asel,s,loc,z,0
DA,all,SYMM
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC,16000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC,39000,1,2
*ENDDO
lswrite,1

time,1           ! Sets time at end of run to 1 sec
autots,on       ! Auto time-stepping on
nsubst,10,10,10 ! Number of sub-steps
outres,all,1    ! Write all output

```

neqit,300

```
ALLSEL,ALL
nset,s,loc,x,X3      ! Apply load
nset,r,loc,y,Y0
D,all,-10,,,,UY,,,,
allsel,all
lswrite,2
lssolve,1,2,1
```

APPENDIX F

```
!-----The Model with reduced angle width(Case 6)-----
FINISH
/clear all
/PREP7
!-----
!          properties
!-----
ET,1,solid187
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170         ! TARGE170 ELEMENTS
mp,ex,1,210000        !steel
mp,prxy,1,.3
MP,MU,1,.7
mp,prxy,2,.2
MP,MU,2,.7
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDE,NUXY,2
MPDE,NUYZ,2
MPDE,NUXZ,2
MPDE,PRXY,2
MPDE,PRYZ,2
MPDE,PRXZ,2
MPDE,EX,2
MPDATA,EX,2,,12100          !steel
MPDATA,EY,2,,430
MPDATA,EZ,2,,430
MPDATA,PRXY,2,,0.2
MPDATA,PRYZ,2,,0.2
MPDATA,PRXZ,2,,0.2
MPDATA,GXY,2,,917
MPDATA,GYZ,2,,917
MPDATA,GXZ,2,,917
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,431
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,7800
!-----
!          DATA INPUT
!-----
X0=0
X1=15
X11=99
X12=159
X2=215
X3=1875/4          ! 1/4 of beam is taken
X_1=-10
X_2=-130
X_3=-150

Y0=405
Y1=15+Y0
Y2=131+Y0
Y3=165+Y0
Y4=500+Y0

Y_1=0

Z0=0
```

```

Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2+10          !upper PART OF THE ANGLE BRACKET
blc4,X0+15,Y0+15,X2,165,Z2+10
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-165,Z2+10          !lower PART OF THE ANGLE BRACKET
blc4,X0+15,Y_1-15,X2,-165,Z2+10
vsbv,2,4

k,2000,X12,Y1,Z1          !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-3, , , ,-Y1-15
vsbv,all,2

CYL4,0,0,D0-4, , , ,-Y1-15          !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1          !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-3, , , ,-Y1-15
vsbv,all,1

CYL4,0,0,D0-4, , , ,-Y1-15          !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-15
RPR4,6,0,0,20,0,-15
vsel, s, , , 1
vsel, a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3          !creating column
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1          !center of hole (colum)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1 , , , ,-25          !making hole
vsbv,all,1

```



```

CYL4,0,0,D0-2 ,,, ,-25      !bolt with head
RPR4,6,0,0,20,0,15
wpoffs,,,-25
RPR4,6,0,0,28,0,-15
vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

```

```

wpoffs,,-(Y0+2*131)
CYL4,0,0,D0-1,,, ,25      !making hol
vsbv,all,1
CYL4,0,0,D0-2 ,,, ,25      !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,25
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all
!.....
!                mesh
!.....

```

```

ALLSEL,ALL      !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

```

```

ALLSEL,ALL      !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !column
vsel,s,,,2
vatt,1,,1

```

ESIZE,15,0,
 MSHAPE,1,3D
 MSHKEY,0
 VMESH,all
 ALLSEL,ALL

Asel,s,area,,89 !top and bottom area of A_bracket
 nsla,s,1
 NSEL,R,LOC,y,420,430
 NREFINE,all,,2,0,1,1
 ALLSEL,ALL
 Asel,s,area,,56 !top and bottom area of A_bracket
 nsla,s,1
 NSEL,R,LOC,x,15,25
 NREFINE,all,,2,0,1,1
 ALLSEL,ALL

! _____ CONTACT PAIRS _____
 ! _____ 2 _____

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC
 ET,3,Targe170 ! TARGE170 ELEMENTS
 KEYOPT,2,2,2
 KEYOPT,2,4,2
 KEYOPT,2,12,5

TYPE,2 !contact element
 REAL,2
 mat,1
 allsel,all
 asel,s,area,,48 !outer bolt head and upper A_bracket
 nsla,s,1
 ESLN,S
 ESURF

TYPE,3 !Target element
 REAL,2
 mat,1
 ALLSEL,ALL
 Asel,s,area,,56 !top area of upper A_bracket
 nsla,s,1
 NSEL,R,LOC,X,X12-25,X12+25
 ESLN,S
 ESURF

! _____
 ! _____ 3 _____

ET,4,conta174 ! CONTA173 ELEMENTS for bonded MPC
 ET,5,Targe170 ! TARGE170 ELEMENTS
 KEYOPT,4,2,2
 KEYOPT,4,4,2
 KEYOPT,4,12,5

TYPE,4 !contact element
 REAL,4
 mat,1
 allsel,all
 asel,s,area,,47 !outer bolt head lower A_bracket
 nsla,s,1
 ESLN,S
 ESURF

TYPE,5 !Target element
 REAL,4
 mat,1
 ALLSEL,ALL
 Asel,s,area,,60 !bottom area of lower A_bracket

```

nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
!
!_____
!_____4_____
ET,6,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170         ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6                !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72      !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7                !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56      !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____
!_____5_____
ET,8,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170         ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8                !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71      !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9                !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60      !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF
!
!_____
!_____6_____
ET,10,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170       ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5

```

```

TYPE,10                !contact element
REAL,10
mat,1
allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____7_____
ET,12,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____8_____
ET,14,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,      !top area of A_bracket

```

```

nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____
!_____9_____
ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                 !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
ESLN,S
ESURF

TYPE,17                 !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____
!_____10_____
ET,18,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170        ! TARGE170 ELEMENTS

TYPE,18                 !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                 !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____
!_____11_____
ET,20,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170        ! TARGE170 ELEMENTS

TYPE,20                 !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52       !top and bottom area of A_bracket
nsla,s,1               !bolt head upper
NSEL,R,LOC,x,0,X2+10

```

```

ESLN,S
ESURF

TYPE,21                !Target element
REAL,20
mat,1
ALLSEL,ALL
Ase1,s,area,,59       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170        ! TARGE170 ELEMENTS
allsel,all
TYPE,22                !contact element
REAL,22
mat,1
allsel,all
Ase1,s,area,,88       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23                !Target element
REAL,22
mat,1
ALLSEL,ALL
Ase1,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170        ! TARGE170 ELEMENTS

TYPE,24                !contact element
REAL,24
mat,2
allsel,all
Ase1,s,area,,5        !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25                !Target element
REAL,24
mat,1
ALLSEL,ALL
Ase1,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!
!_____14_____
ET,26,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170        ! TARGE170 ELEMENTS

TYPE,26                !contact element
REAL,26

```

```

mat,1
allsel,all
Ase1,s,area,,110      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,27              !Target element
REAL,26
mat,1
ALLSEL,ALL
Ase1,s,area,,106      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!
finish
/solut
antype,0
nlgeom,1

time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,1,1,1         ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3      ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
ase1,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
ase1,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 27000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 39000,1,2
*ENDDO

```

```
lswrite,1

time,1                ! Sets time at end of run to 1 sec
autots,on            ! Auto time-stepping on
nsubst,10,10,10     ! Number of sub-steps
outres,all,1        ! Write all output
neqit,300

ALLSEL,ALL
nset,s,loc,x,X3     ! Apply load
nset,r,loc,y,Y0
D,all,,-10,,,,UY,,,,
allsel,all
lswrite,2
lssolve,1,2,1
```


APPENDIX G

```
!-----The Model with 12mm Angle and stiffner(Case 7)-----
FINISH
/clear all
/PREP7
!-----
!          properties
!-----
ET,1,solid187
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170         ! TARGE170 ELEMENTS
mp,ex,1,210000        !steel
mp,prxy,1,.3
MP,MU,1,.7
mp,prxy,2,.2
MP,MU,2,.7
!*
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDE,NUXY,2
MPDE,NUYZ,2
MPDE,NUXZ,2
MPDE,PRXY,2
MPDE,PRYZ,2
MPDE,PRXZ,2
MPDE,EX,2
MPDATA,EX,2,,12100          !glulam
MPDATA,EY,2,,430
MPDATA,EZ,2,,430
MPDATA,PRXY,2,,0.2
MPDATA,PRYZ,2,,0.2
MPDATA,PRXZ,2,,0.2
MPDATA,GXY,2,,917
MPDATA,GYZ,2,,917
MPDATA,GXZ,2,,917
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,2,,431
MPTEMP,,,,,,,,
MPTEMP,1,0
MPDATA,DENS,1,,7800

!-----
!          DATA INPUT
!-----
X0=0
X1=12
X11=99
X12=159
X2=212
X3=1875/4! 1/4 of beam is taken
X_1=-10
X_2=-130
X_3=-150

Y0=405
Y1=15+Y0-3
Y2=131+Y0-3
Y3=165+Y0-3
Y4=500+Y0

Y_1=0
```

```

Z0=0
Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!          GEOMETRY
!-----

blc4,X0,Y0,X2,162,Z2          !upper PART OF THE ANGLE BRACKET
blc4,X0+12,Y0+12,X2,162,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2          !BEAM

blc4,X0,Y_1,X2,-162,Z2        !lower PART OF THE ANGLE BRACKET
blc4,X0+12,Y_1-12,X2,-162,Z2
vsbv,2,4
k,2000,X12,Y1,Z1              !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0-3, , , ,Y1-12
vsbv,all,2

CYL4,0,0,D0-4, , , ,Y1-12    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , ,Y1-12
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0-3, , , ,Y1-12
vsbv,all,1

CYL4,0,0,D0-4, , , ,Y1-12    !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , ,Y1-12
RPR4,6,0,0,20,0,-15
vsel,s, , ,1
vsel,a, , ,4,6,2
vadd,all
allsell,all
WPCSYS,-1,0
blc4,X0,-700,X_3,2*700+Y0,Z3
blc4,X_1,-700,X_3+20,2*700+Y0,Z1_2
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (colum)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1
KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0 , , , , -22      !making hole
vsbv,all,1
CYL4,0,0,D0-1 , , , , -22    !bolt with head
RPR4,6,0,0,20,0,15
wpoffs, , , -22
RPR4,6,0,0,28,0,-15

```

```

vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsell,all

wpoffs,-(Y0+2*128)
CYL4,0,0,D0-1 ,,, ,22      !making hol
vsbv,all,1
CYL4,0,0,D0-2 ,,, ,22      !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,,22
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsell,all
WPCSYS,-1,0
k,4000,X1,Y1,Z0-7          !center of hole (column)
k,4001,X1+150,Y1,Z0-7
k,4002,X1,Y1+150,Z0-7
l,4000,4001
l,4001,4002
l,4002,4000
al,333,334,335
VOFFST,4,7, ,
VADD,1,9

```

```

numcmp,all
!.....
!                mesh
!.....

```

```

ALLSEL,ALL      !A_Bracket
vsel,s,,,3,5,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

```

```

ALLSEL,ALL      !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
vsel,s,,,4,6,2

```

```

vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

```

```

ALLSEL,ALL      !Bolt
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D

```

```

MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL          !column
vsel,s,,,2
vatt,1,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL
!_____CONTACT PAIRS_____
!_____2_____
ET,2,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170          ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

TYPE,2                !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48       !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,3                !Target element
REAL,2
mat,1
ALLSEL,ALL
Ase1,s,area,,127      !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
!_____
!_____3_____

ET,4,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170          ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5
TYPE,4                !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47       !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,5                !Target element
REAL,4
mat,1
ALLSEL,ALL
Ase1,s,area,,59       !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF
!_____
!_____4_____

```

```

ET,6,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170         ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6                !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,71      !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7                !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,127     !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
! _____
! _____ 5 _____

ET,8,conta174        ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170       ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8                !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,70     !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9                !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,59     !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF

! _____
! _____ 6 _____

ET,10,conta174       ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170      ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10              !contact element
REAL,10
mat,1
allsel,all
Asel,s,area,,101    !top and bottom area of A_bracket
nsla,s,1

```

```

ESLN,S
ESURF

TYPE,11                !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,126,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____7_____
ET,12,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170        ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12                !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,100      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13                !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,105,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____8_____
ET,14,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170        ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14                !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,123      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,15                !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,109,      !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____9_____

```

```

ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,122      !top and bottom area of A bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,17                !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,105,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____
!_____10_____
ET,18,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170         ! TARGE170 ELEMENTS

TYPE,18                !contact element
REAL,18
mat,1
allsel,all
Asel,s,area,,51      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55      !top area of A bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____
!_____11_____
ET,20,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170         ! TARGE170 ELEMENTS

TYPE,20                !contact element
REAL,20
mat,1
allsel,all
Asel,s,area,,52      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,21                !Target element
REAL,20
mat,1

```

```

ALLSEL,ALL
Asel,s,area,,58      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____12_____
ET,22,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170      ! TARGE170 ELEMENTS
allsel,all
TYPE,22              !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,87      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,23              !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,104     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____13_____
ET,24,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170      ! TARGE170 ELEMENTS

TYPE,24              !contact element
REAL,24
mat,1
allsel,all
Asel,s,area,,7       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,25              !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,104     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!
!_____14_____
ET,26,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170      ! TARGE170 ELEMENTS

TYPE,26              !contact element
REAL,26

mat,1
allsel,all
Asel,s,area,,108     !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S

```



```

ESURF

TYPE,27          !Target element
REAL,26
mat,1
ALLSEL,ALL
Ase1,s,area,,104    !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,4,0,y,Y0/2,,,,elems
psmesh,,name,,volu,6,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!_____
finish
/solut
antype,0
nlgeom,1

time,1           ! Sets time at end of run to 1 sec
autots,on       ! Auto time-stepping on
nsubst,1,1,1    ! Number of sub-steps
outres,all,all  ! Write all output
neqit,300

nse1,s,loc,x,X_3    ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nse1,s,loc,x,X_3    ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
ase1,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
ase1,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 27000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC, 39000,1,2
*ENDDO

lswrite,1

time,1           ! Sets time at end of run to 1 sec
autots,on       ! Auto time-stepping on
nsubst,10,10,10 ! Number of sub-steps

```

```
outres,all,1      ! Write all output
neqit,300
```

```
ALLSEL,ALL
nset,s,loc,x,X3   ! Apply load
nset,r,loc,y,Y0
D,all,,-10,,,,UY,,,,,
```

```
allsel,all
lswrite,2
lssolve,1,2,1
```

APPENDIX H

!-----The Model with combined Case3 and 5 -----

FINISH

/clear all

/PREP7

!-----

! properties

!-----

ET,1,solid187

ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC

ET,3,Targe170 ! TARGE170 ELEMENTS

mp,ex,1,210000 !steel

mp,prxy,1,.3

MP,MU,1,.7

mp,prxy,2,.2

MP,MU,2,.7

!*

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDE,NUXY,2

MPDE,NUYZ,2

MPDE,NUXZ,2

MPDE,PRXY,2

MPDE,PRYZ,2

MPDE,PRXZ,2

MPDE,EX,2

MPDATA,EX,2,,12100

MPDATA,EY,2,,430

MPDATA,EZ,2,,430

MPDATA,PRXY,2,,0.2

MPDATA,PRYZ,2,,0.2

MPDATA,PRXZ,2,,0.2

MPDATA,GXY,2,,917

MPDATA,GYZ,2,,917

MPDATA,GXZ,2,,917

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,2,,431

MPTEMP,,,,,,,,

MPTEMP,1,0

MPDATA,DENS,1,,7800

!-----

! DATA INPUT

!-----

X0=0

X1=20

X11=99

X12=159

X2=215

X3=1875/4! 1/2 of beam is taken

X_1=-10

X_2=-130

X_3=-150

Y0=405

Y1=20+Y0

Y2=131+5+Y0

Y3=165+5+Y0

Y4=500+Y0 !HEIGHT OF THE COLUM

Y_1=0

Z0=0

```

Z1=-34
Z1_2=-65
Z2=-70
Z3=-75
D0=10
!-----
!           GEOMETRY
!-----

blc4,X0,Y0,X2,165,Z2           !upper PART OF THE ANGLE BRACKET
blc4,X0+20,Y0+20,X2,165,Z2
vsbv,all,2

blc4,X0,Y_1,X3,Y0,Z2           !BEAM

blc4,X0,Y_1,X2,-165,Z2         !lower PART OF THE ANGLE BRACKET
blc4,X0+20,Y_1-20,X2,-165,Z2
vsbv,2,4

k,2000,X12,Y1,Z1               !center of outer hole (Beam)
k,2001,X2,Y1,Z1
k,2002,X12,Y1,Z2

KWPLAN,-1,2000,2001,2002
CYL4,0,0,D0+1, , , ,-Y1-20
vsbv,all,2

CYL4,0,0,D0, , , ,-Y1-20      !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-20
RPR4,6,0,0,20,0,-15
vadd,1,2,3

k,1000,X11,Y1,Z1              !center of inner hole (Beam)
k,1001,X2,Y1,Z1
k,1002,X11,Y1,Z2

KWPLAN,-1,1000,1001,1002
CYL4,0,0,D0+1, , , ,-Y1-20
vsbv,all,1

CYL4,0,0,D0, , , ,-Y1-20      !creating bolt thread and head 1&2
RPR4,6,0,0,20,0,15

wpoffs, , , ,-Y1-20
RPR4,6,0,0,20,0,-15
vsel,s, , , 1
vsel,a, , , 4,6,2
vadd,all
allsell,all

WPCSYS,-1,0
blc4,X0,-700,X_3,700*2+Y0,Z3    !-Y4 må byttes -200
blc4,X_1,-700,X_3+20,700*2+Y0,Z1_2 !2*(Y0+500)med400+Y0
vsbv,1,4

k,3000,X1,Y2,Z1                !center of hole (colum)
k,3001,X1,Y2,Z2
k,3002,X1,Y3,Z1

KWPLAN,-1,3000,3001,3002
CYL4,0,0,D0-1, , , ,-30        !making hole
vsbv,all,1
CYL4,0,0,D0-2, , , ,-30        !bolt with head
RPR4,6,0,0,20,0,15

```

wpoffs,,,-30
RPR4,6,0,0,28,0,-15
vsel,s,,,1
vsel,a,,,3,6,3
vadd,all
allsel,all

wpoffs,,-(Y0+2*134)
CYL4,0,0,D0-1,,, ,30 !making hol
vsbv,all,1
CYL4,0,0,D0-2 ,,,, ,30 !bolt with head
RPR4,6,0,0,28,0,-15
wpoffs,,,30
RPR4,6,0,0,20,0,15
vsel,s,,,1
vsel,a,,,4,8,4
vadd,all
allsel,all
numcmp,all
!.....
! mesh
!.....

ALLSEL,ALL !A_Bracket
vsel,s,,,4,6,2
vatt,1,,1
ESIZE,10,0
MSHAPE,1,3D
MSHKEY,0
VMESH,all

ALLSEL,ALL !Beam
vsel,s,,,1
vatt,2,,1
ESIZE,15,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !Bolt
vsel,s,,,3,5,2
!vsel,a,,,7,8
vatt,1,,1
ESIZE,5,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !Bolt
!vsel,s,,,3,5,2
vsel,s,,,7,8
vatt,1,,1
ESIZE,8,0,
MSHAPE,1,3D
MSHKEY,0
VMESH,all
ALLSEL,ALL

ALLSEL,ALL !column
vsel,s,,,2
vatt,1,,1
ESIZE,15,0,
MSHAPE,1,3D

MSHKEY,0
VMESH,all
ALLSEL,ALL

Asel,s,area,,89 !top and bottom area of A_bracket
nsla,s,1
NSEL,R,LOC,y,420,430
NREFINE,all,,2,0,1,1
ALLSEL,ALL
Asel,s,area,,56 !top and bottom area of A_bracket
nsla,s,1
NSEL,R,LOC,x,15,25
NREFINE,all,,2,0,1,1
ALLSEL,ALL

! _____ CONTACT PAIRS _____
! _____ 2 _____
ET,2,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,3,Targe170 ! TARGE170 ELEMENTS
KEYOPT,2,2,2
KEYOPT,2,4,2
KEYOPT,2,12,5

TYPE,2 !contact element
REAL,2
mat,1
allsel,all
asel,s,area,,48 !outer bolt head and upper A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,3 !Target element
REAL,2
mat,1
ALLSEL,ALL
Asel,s,area,,56 !top area of upper A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S
ESURF

! _____
! _____ 3 _____

ET,4,conta174 ! CONTA173 ELEMENTS for bonded MPC
ET,5,Targe170 ! TARGE170 ELEMENTS
KEYOPT,4,2,2
KEYOPT,4,4,2
KEYOPT,4,12,5
TYPE,4 !contact element
REAL,4
mat,1
allsel,all
asel,s,area,,47 !outer bolt head lower A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,5 !Target element
REAL,4
mat,1
ALLSEL,ALL
Asel,s,area,,60 !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X12-25,X12+25
ESLN,S

```

ESURF
!
!_____4_____
ET,6,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,7,Targe170     ! TARGE170 ELEMENTS

KEYOPT,6,2,2
KEYOPT,6,4,2
KEYOPT,6,12,5
TYPE,6             !contact element
REAL,6
mat,1
allsel,all
asel,s,area,,72   !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,7             !Target element
REAL,6
mat,1
ALLSEL,ALL
Asel,s,area,,56   !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25
ESLN,S
ESURF
!
!_____5_____
ET,8,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,9,Targe170     ! TARGE170 ELEMENTS

KEYOPT,8,2,2
KEYOPT,8,4,2
KEYOPT,8,12,5
TYPE,8             !contact element
REAL,8
mat,1
allsel,all
asel,s,area,,71   !bolt head upper
nsla,s,1
ESLN,S
ESURF

TYPE,9             !Target element
REAL,8
mat,1
ALLSEL,ALL
Asel,s,area,,60   !bottom area of lower A_bracket
nsla,s,1
NSEL,R,LOC,X,X11-25,X11+25

ESLN,S
ESURF

!
!_____6_____
ET,10,conta174     ! CONTA173 ELEMENTS for bonded MPC
ET,11,Targe170    ! TARGE170 ELEMENTS

KEYOPT,10,2,2
KEYOPT,10,4,2
KEYOPT,10,12,5
TYPE,10            !contact element
REAL,10
mat,1

```

```

allsel,all
Asel,s,area,,103      !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,11              !Target element
REAL,10
mat,1
ALLSEL,ALL
Asel,s,area,,89,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____
!_____7_____
ET,12,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,13,Targe170     ! TARGE170 ELEMENTS

KEYOPT,12,2,2
KEYOPT,12,4,2
KEYOPT,12,12,5
TYPE,12              !contact element
REAL,12
mat,1
allsel,all
Asel,s,area,,102     !top and bottom area of A_bracket
nsla,s,1
ESLN,S
ESURF

TYPE,13              !Target element
REAL,12
mat,1
ALLSEL,ALL
Asel,s,area,,107,   !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y2-25,Y2+25
ESLN,S
ESURF
!
!_____
!_____8_____
ET,14,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,15,Targe170     ! TARGE170 ELEMENTS

KEYOPT,14,2,2
KEYOPT,14,4,2
KEYOPT,14,12,5
TYPE,14              !contact element
REAL,14
mat,1
allsel,all
Asel,s,area,,124     !top and bottom area of A_bracket
nsla,s,1             !bolt head upper
ESLN,S
ESURF

TYPE,15              !Target element
REAL,14
mat,1
ALLSEL,ALL
Asel,s,area,,111,   !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S

```



```

ESURF
!
!_____9_____
ET,16,conta174          ! CONTA173 ELEMENTS for bonded MPC
ET,17,Targe170         ! TARGE170 ELEMENTS

KEYOPT,16,2,2
KEYOPT,16,4,2
KEYOPT,16,12,5
TYPE,16                !contact element
REAL,16
mat,1
allsel,all
Asel,s,area,,123      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,17                !Target element
REAL,16
mat,1
ALLSEL,ALL
Asel,s,area,,107,     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-105,-156
ESLN,S
ESURF
!
!_____10_____
ET,18,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,19,Targe170        ! TARGE170 ELEMENTS

TYPE,18                !contact element
REAL,18
mat,2
allsel,all
Asel,s,area,,51       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

TYPE,19                !Target element
REAL,18
mat,1
ALLSEL,ALL
Asel,s,area,,55       !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____11_____
ET,20,conta174         ! CONTA173 ELEMENTS for bonded MPC
ET,21,Targe170        ! TARGE170 ELEMENTS

TYPE,20                !contact element
REAL,20
mat,2
allsel,all
Asel,s,area,,52       !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
NSEL,R,LOC,x,0,X2+10
ESLN,S
ESURF

```

```

TYPE,21                !Target element
REAL,20
mat,1
ALLSEL,ALL
Asel,s,area,,59      !top area of A_bracket
nsla,s,1
ESLN,S
ESURF
allsel,all
!
!_____
!                12
ET,22,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,23,Targe170      ! TARGE170 ELEMENTS
allsel,all
TYPE,22                !contact element
REAL,22
mat,1
allsel,all
Asel,s,area,,88      !top and bottom area of A_bracket
nsla,s,1            !bolt head upper
ESLN,S
ESURF

TYPE,23                !Target element
REAL,22
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y0,Y3+20
ESLN,S
ESURF
allsel,all
!
!_____
!                13
ET,24,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,25,Targe170      ! TARGE170 ELEMENTS

TYPE,24                !contact element
REAL,24
mat,2
allsel,all
Asel,s,area,,5       !top and bottom area of A_bracket
nsla,s,1            !bolt head upper
ESLN,S
ESURF

TYPE,25                !Target element
REAL,24
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,Y_1,Y0,
ESLN,S
ESURF
allsel,all!
!
!_____
!                14
ET,26,conta174      ! CONTA173 ELEMENTS for bonded MPC
ET,27,Targe170      ! TARGE170 ELEMENTS

TYPE,26                !contact element
REAL,26

mat,1
allsel,all

```

```

Asel,s,area,,110      !top and bottom area of A_bracket
nsla,s,1              !bolt head upper
ESLN,S
ESURF

TYPE,27              !Target element
REAL,26
mat,1
ALLSEL,ALL
Asel,s,area,,106     !top area of A_bracket
nsla,s,1
NSEL,R,LOC,y,-180,Y_1
ESLN,S
ESURF
allsel,all

psmesh,,name,,volu,3,0,y,Y0/2,,,,elems
psmesh,,name,,volu,5,0,y,Y0/2,,,,elems
psmesh,,name,,volu,7,0,x,X0,,,,elems
psmesh,,name,,volu,8,0,x,X0,,,,elems
!
finish
/solut
antype,0
nlggeom,1

time,1                ! Sets time at end of run to 1 sec
autots,on             ! Auto time-stepping on
nsubst,1,1,1         ! Number of sub-steps
outres,all,1         ! Write all output
neqit,300

nsl,s,loc,x,X_3       ! constrain
NSEL,r,LOC,y,940,960
d,all,all
allsel,all
nsl,s,loc,x,X_3       ! constrain
NSEL,r,LOC,y,-560,-540
d,all,all

allsel,all
asel,s,area,,1,
nsla,s,1
D,all, , , , ,UZ, , , ,

Allsel,all
asel,s,loc,z,0
DA,all,SYMM

*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),1,2
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC,16000,1,2
*ENDDO
*DEL,_z
*DIM,_z,ARRAY,2,1,1
*SET,_z(1),3,4
*DO,_z1,1,2
SLOAD,_z(_z1),9,tiny,FORC,39000,1,2
*ENDDO
lswrite,1

time,1                ! Sets time at end of run to 1 sec

```

```
autots,on           ! Auto time-stepping on
nsubst,10,10,10    ! Number of sub-steps
outres,all,1       ! Write all output
neqit,300
```

```
ALLSEL,ALL
nset,s,loc,x,X3     ! Apply load
nset,r,loc,y,Y0
D,all,-10,,,,UY,,,,
!f,all,fy,-10000
allsel,all
!swrite,2
!ssolve,1,2,1
```




Norges miljø- og biovitenskapelig universitet
Noregs miljø- og biovitenskapelige universitet
Norwegian University of Life Sciences

Postboks 5003
NO-1432 Ås
Norway