Structural health monitoring of splice joint in a steel beam

Aastha¹ and RR Singh²

¹ Ph.D. Student, Civil Engineering Department Punjab Engineering College (Deemed to Be University) Chandigarh, India
² Professor & Head, Department of Civil Engineering Punjab Engineering College (Deemed to Be University), Chandigarh, India

E-mail: aastha.aastha.civ09@gmail.com

Abstract. Nowadays, steel sections are extensively used in various constructions because of its lesser erection time and higher reliability. These members are connected either by welding or bolts. As time goes on, the structure degrades due to various reasons. In our study, the degradation of joints with bolted connections is presented. In reality, joints are damaged frequently when compared to the other parts of the structure. As bridges are generally of long spans, splicing is done to ensure the continuity of the spans and this splicing is done with the help of welding or bolts. Loosening of bolts is the most common damage that occurs in bolts, making the joint act like a semi-rigid joint. We have considered a bridge model consisting of two beams connected by splicing with bolts’ help, and this entire modelling is done in ABAQUS 6.13. Our intention to study the behaviour of the joint in both damaged and undamaged case. Loosening of bolts is considered as damage, and deflection of the joint is considered as the extent of the damage. We have performed a static analysis of the model. Theoretical validation is also done by developing an elemental stiffness matrix with semi-rigid connections using fixity factors.

Keywords: Splice joint, Bolts, ABAQUS 6.13, Static analysis, Stiffness matrix

1. Introduction

Structural Health Monitoring (SHM) aims to diagnose the condition of the whole structure or different parts of the structure at every instance during the life of the structure. SHM is an improved or modified way to make a Non-Destructive Evaluation. But the truth is, it is much more than Non-Destructive Evaluation. It involves in the integration of sensors, data transmission, computational ability and processing ability inside the structure. The primary objective of the users, manufacturers, and maintenance teams is to know about the in-service structure's structural integrity.

The effects of SHM are as follows

• The optimal use of structure.
• It avoids catastrophic failure of the structure.
• Provide sufficient time to improve the structure.
• Improvement of safety and reliability by minimizing human errors, downtime and labour.

After we encountered some accidents as a result of unsatisfactory maintenance, for example, the aeronautical field, the accident of Aloha airlines or in civil engineering field the collapse of Mianus River Bridge and ill controlled manufacturing process, for example, the Injaka bridge collapse, safety has given significant importance in Structural Health Monitoring. In both cases, the reason for damage or collapse was the ageing of structures. The design philosophy followed by the bridges constructed several years ago (maybe a century ago) were outdated, and the consistent increase in the traffic pressure along with the loss of the strength of structure due to its aging makes the structure vulnerable to damage, if not monitored may lead to collapse. So it is necessary to monitor the performance of the structure to perform its intended use. Figure 1 shows the benefits of SHM are constant reliability and maintenance costs instead of decreasing reliability and increasing maintenance costs.

Figure 1. Benefits of SHM [1]

Further, with the advance in technology, it is now possible to build more extensive and more complex structures that are much lighter in weight. Hence, composite materials are increasing use as primary structural components in many modern applications due to their high strength-to-weight ratio and other properties that make them preferable to metals and other conventional engineering materials. The literature on various damage detection methods are being reviewed in the coming chapters. We have plenty of literature on building frames, bridges and trusses, but on the contrary, we have significantly less amount of study on bolted joints. Therefore, the present project work is carried out with the following objective

• Development of model and analyse statically using ABAQUS CAE 6.13 for both undamaged and damaged case. Find out the displacements in both the case and verify the existence of damage.
2. Stiffness matrix formulation

2.1 Introduction

We assume the connections as fully rigid or frictionless pinned in the conventional design and analysis of steel structures. But in practice, we can’t achieve complete rigidity. So, all the connections in practice are semi-rigid. The design of semi-rigid connections are more economical and practical when compared to our actual assumption of rigid connections, but the design process is very tedious and time taking even for a simple structure.

2.1.1 Element derivation.

We generally take fixed-fixed boundary condition for the development of the elemental matrices. Now a days bolted connections widely used in joint connections and these bolts have a tendency to fail mostly by loosening. This loosening of bolts introduces the semi rigidity in a joint. Therefore, to introduce the semi-rigidity at the joint, the element matrices are developed based on the elastic boundary condition. This elasticity of the boundary is provided by the two rotational springs placed at the two ends of an element. The boundary conditions are as follows.

![Figure 2 (a). Degrees of freedom of an element][2]

Deflected shape of the element which consists of two springs at the end to consider the end flexibility is shown in figure 2 (a). Therefore, a relative rotation will take place in the inner and outer side of the spring. In inner side, the rotations are $L_1\theta_1$, $R_1\theta_1$ and in outer side, these are $L_2\theta_2$, $R_2\theta_2$ respectively, shown in figure 2(b).

![Figure 2 (b). Beam element with semi rigid connections][2]
According to[3]
\[
\begin{align*}
\frac{RM}{R_c} &= R\theta_2 - R\theta_1 \\
\frac{LM}{L_c} &= L\theta_2 - L\theta_1
\end{align*}
\] (1)

In the above equations (1) and (2) \(LM, RM, L_c\) and \(R_c\) are the applied moment and rotational spring constants at the left and right side of the element, respectively. The elemental stiffness and mass matrices are derived as per[2] formulation. The formulation is described as below

The displacement field of an element can be written as
\[
\begin{align*}
\begin{bmatrix}
\frac{1}{g_1} \\
\frac{1}{g_2} \\
\end{bmatrix}
\end{align*}
\] (3)

As that the element is the assemblage of one straight element in which boundary conditions are
\[
\begin{align*}
\frac{1}{g_1} \\
\frac{1}{g_2}
\end{align*}
\] (4)

And two rotational springs in which boundary conditions for the left and right spring are
\[
\begin{align*}
\frac{1}{g_3} \\
\frac{1}{g_3}
\end{align*}
\] (5)

Displacement due to rotation is
\[
\begin{align*}
\frac{1}{g_4} \\
\frac{1}{g_4}
\end{align*}
\] (7)

Therefore, total displacement is
\[
\begin{align*}
\begin{bmatrix}
\frac{1}{g_1} \\
\frac{1}{g_1}
\end{bmatrix}
\end{align*}
\] (8)

Where, \(L_1, L_2, R_1, R_2\) are determined by the following ways. Stiffness matrix for left rotational spring is
\[
\begin{align*}
\begin{bmatrix}
L_c & -L_c \\
-L_c & L_c
\end{bmatrix}
\end{align*}
\] (9)

Similarly, for the right rotational spring, \[
\begin{align*}
\begin{bmatrix}
R_c & -R_c \\
-R_c & R_c
\end{bmatrix}
\end{align*}
\] (10)

Rotational component of the member stiffness matrix is
\[
\begin{align*}
\begin{bmatrix}
4E_1 & 2EI \\
2EI & 4E_1
\end{bmatrix}
\end{align*}
\] (11)

Assembled matrix for the rotation is-
Stiffness and mass matrix of a beam element for fixed-fixed boundary conditions is given in equation 14 and then to equation 8, the displacement field or shape function of the element is derived. The shape function is as follows

\( y = [p_1 p_2 L \quad p_1 p_2 L] \left( \begin{bmatrix} 1 & 1 & -1 & 0 \\ 1 & 0 & -1 & 1 \end{bmatrix} \right) + v_1 p_1 + v_2 p_2 \)  

Substituting the value equation 15 in equation 14 and then to equation 8, the displacement field or shape function of the element is derived. The shape function is as follows

\( y = \left[ \begin{bmatrix} 1 & 1 & -1 & 0 \\ 1 & 0 & -1 & 1 \end{bmatrix} \right] \left( \begin{bmatrix} \theta_1 \\ \phi_1 \\ \theta_2 \\ \phi_2 \end{bmatrix} \right) + v_1 p_1 + v_2 p_2 \)  

Stiffness and mass matrix can be determined by the standard procedure.

\( k = \int_0^L EI[N'(x)]^T [N'(x)] \, dx \)  

\( m = \int_0^L \bar{m}[N(x)]^T [N(x)] \, dx \)
2.1.2 Fixity factors

According to [4] Stiffness matrices of a member with elastic restraint at the ends are presented in the form of the stiffness matrices for members with rigid connections modified by a correction matrix whose elements are functions of two parameters, designated as “fixity factors” of the members.

An approximate linear relationship exists within a specific region between the applied moment and the relative rotation of the beam and column (also for beam and beam) as shown in the figure 2 (c). For practical purpose, \( \phi = MA \) may be considered as the acceptable relationship in the design of frames with semi-rigid connections. The inverse of the slope of the assumed straight-line connection factor, \( \lambda \) and its magnitude depends on the type of connection of the members.

![Figure 2. (c) Relation between applied moment and relative rotation [4]](image)

Fixity factors of an element of length \( L \) having two nodes, \( i \) as tail node and \( j \) as head node, are

\[
\gamma_i = \frac{L}{L + 3EI\lambda_i} \tag{19}
\]

\[
\gamma_j = \frac{L}{L + 3EI\lambda_j} \tag{20}
\]

\( \lambda_i \) & \( \lambda_j \) are flexibility (inverse of stiffness) at \( i^{th} \) and \( j^{th} \) node respectively. In derivation of stiffness matrix for semi-rigid joint we used \( L_r \), which is stiffness at left node of an element and \( R_r \) is the stiffness at right node of the element.

The dimensionless parameters, \( \gamma_i \) & \( \gamma_j \), are designated as “fixity factors” of the member. The value of \( \gamma \) depends on the known semi-rigid connection factor and the geometrical and elastic properties of the member. It varies from zero for a frictionless pin connection to unity for a perfectly rigid connection.

According to EURO code 3

| Fixity factor connection          |
|----------------------------------|
| 1 - 0.891                       | rigid |
Deflection at the centre of a beam due to point load at centre is

\[ \delta_p = \frac{PL^3}{48EI} \]  \hspace{1cm} (21)

Here, \( P = 5000 \text{N} \)
\( L=3030 \text{mm} \)
\( E=200000 \text{N/mm}^2 \)
\( I=2490208.33 \text{mm}^4 \)

Therefore deflection at the centre is, \( \delta_p = 5.818 \text{mm} \)

**Theoretical approach with fixity factor:**
Entire length of the beam is divided into 4 elements each of length 0.7575m shown in figure 2(d)

![Figure 2 (d). Discretisation of the beam](image)

At every node there will be two degrees of freedom (Rotational and translational) except at supports which will be having one degree of freedom (Rotational) figure 2(e)

![Figure 2 (e). Degrees of freedom](image)

Fixity factor is used to represent rigidity of a connection. It is assumed that each element is connected to another element by means of a spring. Each element has two nodes and a fixity factor is assigned to two nodes of every element. Considering 0.143 fixity factor at supports and 0.9999 for remaining nodes and global stiffness matrix is formulated from elemental stiffness matrices.

For the above loading and geometric properties deflection values by using exact formula and using fixity factors are tabulated in Table 1.
Table 1. Comparison of deflections

| Location of deflection (mm) | Deflection by exact formula (mm) | Deflection by fixity factor (mm) |
|----------------------------|---------------------------------|---------------------------------|
| L/4 = 757.5                | 4                               | 4                               |
| L/2 = 1515                 | 5.818                           | 5.819                           |
| 3L/4 = 2272.5              | 4                               | 4                               |

3. Details of the model

3.1 Introduction

Splices are therefore most often used when structural elements are required in longer lengths than the available material. The most common form of the splice joint is the half lap splice, which is common in building construction, where it is used to join shorter lengths of timber into longer beams (Wikipedia). These splices must be capable of transferring the shear and moment effectively in the beam at the point of splice. There are four types of splice joints

- Half lap splice
- Bevel lap splice
- Tabled splice
- Tapered finger splice

Often bolts are used to do splicing, where we assume the joints to be rigid. But as the bolts get loosened or any damage to the bolts contradicts our assumption. Then the joint will no longer be rigid, it becomes semi-rigid. From the studies we came to know that half of the failure of steel structures is caused by failure of bolts. There are many ways a bolted joint can get damaged but most failure of the bolted joints is caused by loosening of bolts.

3.1.1 Details of the model

We have considered a double T cross-section beam which was joined by splicing the webs and bolts are used as the connectors. Two double T cross-sections beams are spliced using a total of four splice plates and four bolts. The isometric view and cross-sectional view are shown in figure 3(a) and figure 3 (b) respectively.
The total span of the model is 3030mm, which includes two beams each of length 1500mm with a gap of 30mm between them. These two beams are simply supported at the ends and are connected by splice plates at the mid-span. A vertically downward load of 5000N is applied at the centre of the whole structure.

Flange dimensions: $300 \times 5 \times 1500$ (mm)

Web dimensions: $5 \times 100 \times 1500$ (mm)
3.1.2 Splice plate

Consider 26mm diameter bolts of 10.9 grade, then from this we can decide the edge distance and pitch of the bolts for the splice plates.

Edge/end distance = 1.7×d = 1.7×26 = 44.2mm ≈ 45mm
Pitch = 2.5×d = 2.5×26 = 65mm

From this we can get the dimensions of splice plate shown in figure 3(c): 6 × 90 × 250 (mm)

![Figure 3(c). Splice plate details](image)

3.1.3 Material properties

Grade of all plates: Mild Steel (yield strength = 250MPa)
Grade of bolts: 10.9 (ultimate strength = 1000MPa and yield strength = 900MPa)
Young’s Modulus of Elasticity = E = 200000MPa
Poisson’s ratio = μ = 0.3
Density of steel = 7800Kg/m³

3.1.4 Checks for safety of the structure

3.1.4.1 Beam

**Flexure:**

Geometric properties of the beam are as follows:

Moment of inertia, I = 2490208.33 mm⁴
Area of cross-section, A = 2500 mm²
Centroid from top and bottom fibre, \(y_{\text{top}} = 23.5\text{mm} \& y_{\text{bottom}} = 81.5\text{mm}\)

From Euler – Bernoulli furof flexure, we have
Where, \( M \) = Moment acting the section considered  
\( I \) = Moment of cross-section

\[ f = \frac{M}{I} \]  (22)

\[ f_{\text{max}} = \frac{M_{\text{max}}}{I} \times y_{\text{max}} \]  (23)

\( f \) = Flexural stress  
\( y \) = distance from neutral axis

Bending moment will be maximum at the centre for a simply supported beam with a point load at the centre,

\[ M_{\text{max}} = \frac{P \times L}{4} \]  (24)

\[ = \frac{5000 \times 3000}{4} = 3.7875 \text{ KN-m} \]

Now from substituting all the values in equation 23, we get

\[ f_{\text{max}} = \frac{3.7875 \times 10^6}{2490208.33} \times 81.5 \]

\[ = 123.958 \text{ N/mm}^2 \]

As the maximum flexural stress is less than the yield stress, we can say that the beam is safe against flexure.

Design moment:

Design moment in a beam is

\[ M_d = \frac{1.2 \times \frac{L}{2} \times f_y}{1.1} \]  (25)

\[ = \frac{1.2 \times 2490208.33 \times 250}{1.1} \]

\[ = 8.333 \text{ KN-m} \]

As the design moment is more than that of applied moment \( (M_d>M_{\text{applied}}) \), the structure or model considered is safe.

3.1.4.2 Splice plate

Flexure:

The maximum bending is distributed to the webs first and from them to splice plates. So, webs take half of the applied moment (as there are two webs) and splice plates take quarter of the applied moment (as there are two plates for each web).

From Euler-Bernoulli equation of flexure, we have
\[
\frac{M}{I} = \frac{f}{y}
\]
\[
f_{\text{max}} = \frac{M_{\text{max}}}{I} \times y_{\text{max}}
\]
\[
f_{\text{max}} = \frac{0.946875 \times 10^6}{\left(\frac{9 \times 90^3}{12}\right)} \times 45
\]
\[
= 140.278 \text{ N/mm}^2 \text{ (< 250 N/mm}^2\text{)}
\]

As the maximum stress is not exceeding yield stress of the plate, splice plate is also safe against flexure

3.1.4.3 Bolts

Bearing:

The total moment applied on one side of the web is \(M/2\), i.e. 1.89375KN-m. The total tensile or compressive force is calculated as,

\[
\text{Tensile/Compressive force} = \frac{\text{Moment}}{\text{Leverarm}}
\]
\[
= \frac{1.89375 \times 10^6}{130}
\]
\[
= 14.596 \text{KN}
\]

Bearing strength of the bolts is given as

\[
BS = 2.5 \times K_b \times d \times t \times f_{tu}
\]
\[
= \min \left(\frac{250}{1000}, 1\right)
\]
\[
= 0.25
\]

\(d = \text{diameter of the bolt} = 26\text{mm}\)

\(t = \text{min (thickness of web, sum of thickness of two splice plates)}\)

\(= \text{min (5, 5+5)}\)

\(= 5\text{mm}\)

From equation 26 we get the value of bearing strength as,

\[
BS = \frac{2.5 \times 0.25 \times 26 \times 5 \times 1000}{1.25}
\]
\[
= 65\text{KN}
\]

Bearing strength is more than the tensile/compressive force. So, OK
Shearing:

Shearing strength of the bolts is given as

\[ \frac{f_{ul}}{\sqrt{3}} \times \frac{(n_n \times A_{nb} + n_s \times A_{sb})}{1.25} \]  

\( n_n \) = Number of shear planes with the threads intercepting the shear plane in a bolted connection  
\( n_s \) = Number of shear planes without threads intercepting the shear plane in a bolted connection  
\( A_{sb} \) = Shank gross cross-sectional area (nominal area) of the bolt  
\( A_{nb} \) = Net tensile cross-sectional area of the bolt  
\( d \) = Diameter of the bolt  
\( f_{ul} \) = Ultimate shear strength of the material

\[ n_n = 2 \]  
\[ A_{nb} = \pi \left( \frac{d}{4} \right)^2 = \pi \left( \frac{2}{4} \times 26^2 \right) = 416.991 \text{mm}^2 \]  
\[ n_s = 0 \]  
\[ A_{sb} = 416.991 \text{mm}^2 \]

Therefore, Shear strength = \[ \frac{1000}{\sqrt{3}} \times \frac{(2 \times 416.991 + 0)}{1.25} \]  
= 385.2KN

Friction:

The design shear capacity of the bolt as governed by slip for friction type connection is given as,

\[ V_{dsf} = \frac{V_{nsf}}{\lambda_{nsf}} \]  
(28)

Where, \( V_{nsf} = \mu_f \times n_e \times K_b \times F_0 \) (\( \mu_f = 0.55 \), \( n_e = 2 \), \( K_b = 0.7 \))

\( F_0 = A_{sb} \times f_o \)
\( f_o = 0.7 \times f_{ub} \)

Therefore, from equation 28 we get

\[ V_{nsf} = 0.55 \times 2 \times 0.7 \times 416.991 \times 0.7 \times 1000 \]
= 224.758KN

\[ V_{dsf} = \frac{224.758}{1.25} \]
= 179.806KN
4. Modelling in ABAQUS

The whole assembly is created using ABAQUS 6.13 and is analysed statically to obtain the desired results of deflection at the centre.

4.1 Parts

Parts are the models which are used to define the individual elements of the entire assembly. In the model we considered there are four parts

- Beam
- Splice plate
- Bolt
- Dummy

4.1.1 Beam

Beam is modelled as a 3-dimensional modelling space of deformable type and solid shape of extrusion type. The structure considered is made up of two beams with a gap of 30mm between them. So, length of each beam is 1500mm and the cross-sectional properties were discussed in the above chapter. To draw the cross-section sketch (in X-Y axes) we use ‘Create lines-connected’ option, where we need to enter the co-ordinates of the points in a sequential order in order to get the desired sketch figure 4(a) and figure 4(b). After the sketch is created or drawn it is extruded in Z-direction by providing the extruding depth as 1500. This gives us the 3-d model of the beam. As we are connecting the beams with the help of bolted connections, both the webs should have holes of diameter 26mm. in order to provide holes we need to select ‘Create cut-extrude’ option and then select the plane for the extruded cut. After selecting the plane, the face of web will be displayed on which hole has to be made and sketch of the hole is made with the help of ‘Create circle-centre and perimeter’ option. Once the sketch of hole is made on the web select ‘through all’ option that appears in the dialog box so as to get the holes on both the webs.

![Figure 4(a) Side view of beam with sketch of holes](image-url)
4.1.2 Splice plate

Splice plate is also modelled as a 3-dimensional modelling space of deformable type and solid shape of extrusion type Figure 4(c) and figure 4 (d). It is oriented in such a way that its length is along the longitudinal direction of the beam. So, the dimensions of the plate in X-Y plane are $5 \times 90$ (mm) and is extruded 250mm (length of the plate) in Z-direction. As we are using these plates for connecting the two beams, holes are made with the same edge distance (45mm) and the centre to centre distance between holes will be 130mm ($50+50+30$).
4.1.3 Bolt

Bolt is modelled as a 3-dimensional modelling space of deformable type and solid shape of revolution type figure 4(e) and figure 4(f). As it is revolution type, sketch with half dimensions were made in X-Y plane and is revolved about Y-axis by 360°. Thickness of the bolt is considered as 5mm and the length of the shank is thickness of the web plus twice the thickness of the splice plate which turns out to be 15mm (5+5+5). So, the total length of the bolt will be 25mm with 15mm shank length and bolt head diameter is taken as 40mm.
The bolt is partitioned at the centre by using ‘Create partition’ option and then selecting define cutting plane where we again select point and normal for the partition. Partitioning is required because we apply pretension with the help of bolt load at the interior surface of the bolts.

![3-D bolt](image)

**Figure 4(f).** 3-D bolt

### 4.1.4 Dummy

Dummy is used to cover the gap between the two beams which are to be connected by splicing. This part does not have any structural importance. It is modelled as a 3-dimensional modelling space of deformable type and solid shape of extrusion type shown in figure 4 (g). The dimensions of dummy in X-Y plane are 300 × 105 (mm) and is extruded 30mm in Z-direction.

![Extruded view of dummy](image)

**Figure 4(g).** Extruded view of dummy
4.2 Material and Section Properties

Steel is the material which is used for the analysis. So we need to give the properties of steel in property module by clicking on create material. In ABAQUS all dimensions must be maintained same so that errors won’t arise at the time of calculations. We take load in Newton and dimension in millimetres. Hence Young’s Modulus of Elasticity is given as 200000 and Poisson’s ratio as 0.3 figure 4(h).

![Figure 4(h). Defining elastic properties](image)

Section is also defined in property module and a solid homogenous section is chosen for analysis figure4(i) and figure 4(j). After creating the section, we need to assign the section to every part and we need to be sure that we have selected the property and section properly. Assignment of the section can be done by clicking on ‘Assign section’ in the property module.
4.3 Assembly

The sole purpose of this module is to assemble all the parts together and position them according to the requirement. We create our assembly with the help of two beam parts, four splice parts, four bolts and a dummy part. For assembling the parts, we need to click on ‘Create instance’ and then we need to select the parts that are to be assembled. First, we take beam part and a dummy part and the dummy part is attached to the beam part by using translate option. After that we need to translate and rotate the other beam part in such a way that 30mm gap should be maintained between the two beams figure 4(k). When this setup is done we can remove the dummy part as its purpose has been met. As setting up of the beams is done, now we need to translate the splice plates in such a way that
the holes of the plates and are concentric to each other. As the connection is given by bolts, we need to rotate the bolt first by 90° about Z-axis so that the orientation of their longitudinal axis is along X-axis. After rotating the bolts they are translated in such a way that the centre of the bolt exactly coincides with the centre of the holes assembly.

Figure 4(k). Complete assembly

4.4 Interactions

Interactions are defined to surfaces which are in contact with each other. In the present model we defined two different type of interactions, ‘Surface-to-surface contact (standard)’ and ‘Tie constraint’. Tie constraint is defined for the shank area of the bolt and inner surface of the holes of web and splice plates so that there won’t be any relative displacement between them. Surface to surface contact is defined for ‘bolt head & splice plates’ and ‘web of the beam & splice plates’.

Before defining interactions we need to know about master surface and slave surface. Each node of slave surface attempts to find the closest point on the master surface and the master surface normal passes through the slave node. The interaction is then discretized between the point on the master surface and the slave surface. Slave surface must have a finer mesh whereas master surface can have a coarser mesh.

4.4.1 Surface-to-surface contact

In the interaction module first we need to define the contact property using ‘Create Interaction Property’ option. Here we define the tangential and normal behaviour of the surfaces. In tangential behaviour we use penalty option to enter the coefficient of friction value as 0.5 (coefficient of friction between steel surfaces varies from 0.5 to 0.7) whereas normal behaviour is defined by the default properties. After defining the contact property, we define the interactions between web & splice plates and bolt head & splice plates by choosing ‘Create Interaction’ wherein we choose ‘Surface-to-surface contact (standard)’ option. After selecting the above said option software prompts us to choose the master surface and slave surfaces.

Interactions should be defined individually. For web & splice plates, there will be four interactions, one for each face. Master surfaces will be the faces of two webs and slave surface will be the face of splice plate in contact with those two webs. After selecting master and slave surfaces ‘Edit Interactions’ dialog box will appear in which we need to choose small sliding as sliding formulation, surface to surface as the discretization method, adjust to remove over-closure as slave
adjustment and finally we need to select the contact property defined earlier. The same is repeated for the rest three interactions.

For bolt head & splice plates also there will be four interactions. Master surfaces will be the heads of the bolts in contact with one face of a single splice plate i.e. four bolt head surfaces and the slave surface will be the face of the splice plate which is in contact with all the master surfaces selected. We define the same properties as above in edit interaction dialog box.

4.4.2 Tie constraint

Tie is defined from the constraints option in the interaction module. In this also we have four constraints as we have four bolts. Master surface will be the shank area of the bolt and the slave surfaces will be the surfaces of the holes which are in contact with the shank area of the bolt. In the edit constraint dialog box we don’t do any changes.

4.5 Step and loading

We need to define load steps in step module prior to the application of load. As we are going to give two different type of loads i.e. transverse load and bolt load we need to define two different load steps. First we need to define bolt load step then the transverse load step is defined.

After defining load steps we will open load module and give boundary conditions to the assembled model. We are taking simply supported boundary conditions for the analysis. So we will define the boundary conditions using ‘Create Boundary Condition’ option and choose Zasymm in antisymmetric/symmetric type. This condition is only used for both the ends as this gives apt results for simply supported boundary condition.

Pretension to the bolt is given by choosing bolt load option with bolt load step in ‘Create Load’ menu. Every bolt is given pretension individually. The interior surface of the bolt is selected over which the bolt load is applied in Newton. The internal surface of the bolt can be selected with the help of partition done earlier. After selecting the surface of the bolt we need to choose the direction of load application and finally enter the magnitude of the load. This is repeated for all the bolts.

After application of bolt loads, we need to load the beam with a point load. This loading should be done in transverse load step by choosing concentrated force in create load menu. We select the nodes where the load need to be applied and enter the magnitude of the loading in Newton.

4.6 Meshing

All the parts are meshed after assigning the sections as shown in figure 4(l) only which is generally known as dependant type meshing. For bolts and beam parts we used a linear tetrahedron element of default type i.e. C3D4 (4 node linear tetrahedron) but for the splice plate we used hexagonal element of C4D8 type i.e. 8 node linear brick element. Meshing should be done very carefully as it leads to difference in the results. A correct mesh size can be given only based on experience. Before meshing we need to seed the parts and assign the element type.
4.7 Static analysis

After meshing we need to run the jobs for different bolt loads. We kept on increasing the bolt loads and finally we found out that at bolt load of about 400KN we are approximately achieving full tight condition as there is negligible difference in the value of deflection with further increment in load. We have run several jobs and deflections are calculated for different bolt loads.

The initial loading is taken as 1N which approximately represents complete loosening of bolts case. After 1N the results are taken at 50KN load which is increased till 400KN with an increment of 50KN. Deflection of shape can be seen in figure 4(m).
5. Results and discussion

5.1 Results

Results are shown in Table 2 (a) and Table 2 (b) for various bolt loads. Graphical presentation is shown in figure 5(a), figure 5(b), figure 5(c) and figure (d)

| Bolt load(KN) | Percentage loosening | Deflection at the centre (mm) |
|---------------|----------------------|-------------------------------|
| 0.001         | 99.99975             | 9.245                         |
| 50            | 87.5                 | 8.136                         |
| 100           | 75                   | 7.303                         |
| 150           | 62.5                 | 6.896                         |
| 200           | 50                   | 6.621                         |
| 250           | 37.5                 | 6.423                         |
| 300           | 25                   | 6.273                         |
| 350           | 12.5                 | 6.155                         |
| 400           | 0                    | 6.06                          |
| 450           | -                    | 5.98                          |

Table 2(a). Deflection at centre for various bolt loads

![Graph](image-url)

**Figure 5(a).** Deflection vs Bolt load
Table 2(b). Relation between bolt load and fixity factors

| Bolt load (KN) | Fixity Factor |
|---------------|---------------|
| 0.001         | 0.68852       |
| 50            | 0.7536        |
| 100           | 0.81823       |
| 150           | 0.8572        |
| 200           | 0.88724       |
| 250           | 0.9112        |
| 300           | 0.9307        |
| 350           | 0.94725       |
| 400           | 0.961         |

Figure 5(b). Deflection vs percentage loosening

Figure 5(c). Fixity factor vs Bolt load
5.2 Discussion & Future Scope

In static analysis there is a huge difference in the deflection values for damaged and undamaged case. We can find out the location of damage and even the extent of damage only in terms of deflection. But in reality, this won’t be the only load case. So, the future scope of the present study is as follows

- Dynamic analysis of the model could be done and correlation of these results with experimental results will give us a better understanding.
- Parameters sensitive to damage should be found out, other than deflection, may be modal frequencies, modal strain energy etc.

Appendix A

Stiffness and consistent mass matrix of an element with fixed-fixed boundary conditions

\[
K = \begin{bmatrix}
12EI & 6EI & -12EI & 6EI \\
L^2 & L^2 & -L^2 & 2EI \\
6EI & L^2 & -6EI & L \\
L^2 & 2EI & -L^2 & 4EI \\
\end{bmatrix}
\]

\[
M = \frac{mL}{420} \begin{bmatrix}
156 & 22L & 54 & -13L \\
22L & 4L^2 & 13L & -3L^2 \\
54 & 13L & 156 & -22L \\
-13L & -3L^2 & -22L & 4L^2 \\
\end{bmatrix}
\]
Appendix B

Here $r_a$ and $r_b$ are the rotational spring stiffness at the two ends of the element.

Stiffness matrix coefficients and mass matrix coefficients for semi-rigid boundary conditions are

$$k_{11} = \left(12E^3I^3\left(4r_a^2 - 4r_ar_b + 4r_b^2\right) + 12E^3I^2L\left(2r_a^2r_b + 2r_ar_b^2\right) + 12EI_L^2r_a^2r_b^2\right)$$
$$\left(L\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2\right)$$

$$k_{21} = \left(6E^3I^3r_a\left(8r_a - 4r_b\right) + 6EI_L^2r_a^2r_b^2 + 24E^2I^2Lr_a^2r_b\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$k_{22} = \left(4EILr_a^2\left(12E^2I^2 + 6EILr_a + L^2r_a^2\right)\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$k_{31} = -k_{11}$$

$$k_{32} = -k_{12}$$

$$k_{33} = k_{11}$$

$$k_{41} = \left(6EI_L^2r_a^2r_b^2 - 6E^3I^3r_a\left(4r_a - 8r_b\right) + 24E^2I^2Lr_a^2r_b\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$k_{42} = \left(2EILr_ar_a\left(12E^2I^2 - r_ar_bL^2\right)\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$k_{43} = -\left(6EI_L^2r_a^2r_b^2 - 6E^3I^3r_a\left(4r_a - 8r_b\right) + 24E^2I^2Lr_ar_b\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$k_{44} = \left(4EILr_a^2\left(12E^2I^2 + 6EILr_a + L^2r_a^2\right)\right)$$
$$\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2$$

$$M_{11} = \left(Lm_e\left(1680E^4I^4 + 1344E^3I^3Lr_a + 924E^3I^3Lr_b + 272E^2I^2Lr_a^2 + 656E^2I^2Lr_br_a\right)\right)$$
$$\left(35\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2\right)$$

$$M_{21} = \left(Lm_e\left(672E^3I^3 + 288E^2I^2Lr_a + 32EI_L^2r_a^2 + 110EI_L^2r_ar_b + 11r_aL^3r_b\right)\right)$$
$$\left(210\left(12E^2I^2 + L^2r_ar_b + 4EILr_a + 4EILr_b\right)^2\right)$$
\[ M_{31} = M_{12} \]

\[ M_{32} = \frac{\left( L \frac{M r_a}{2} \left( 32 E^2 I^2 + 11 E I L r_a + L^2 r_a^2 \right) \right)}{\left( 105 \left( 12 E^2 I^2 + L^2 r_a r_b + 4 E I L r_a + 4 E I r_b \right)^2 \right)} \]

\[ M_{33} = \left( \frac{3Lm}{2} \left( 560 E^4 I^4 + 364 E^3 I^3 L r_a + 364 E^2 I^2 L^2 r_a^2 + 52 E^2 I^2 L r_a r_b^2 + 216 E^2 I^2 L r_b r_a + 52 E^2 I^2 L^2 r_b^2 + 27 E^2 I L^3 r_a^2 r_b + 27 E I L^3 r_a r_b^2 + 3 L^4 r_a^2 r_b^2 \right) \right) \]

\[ M_{34} = \left( \frac{3Lm}{2} \left( 1176 E^3 I^3 + 684 E^2 I^2 L r_a + 264 E I L^2 r_a^2 + 76 E I L^2 r_a r_b^2 + 76 E I L^2 r_b^2 + 130 r_a L^3 r_b^2 \right) \right) \]

\[ M_{41} = M_{14} \]

\[ M_{42} = \left( \frac{L^3 m r_a}{2} \left( 124 E^2 I^2 + 3 L^2 r_a r_b + 20 E I L r_a + 20 E I r_b \right) \right) \]

\[ M_{43} = \left( \frac{L^3 m r_a}{2} \left( 672 E^3 I^3 + 288 E^2 I^2 L r_a + 32 E I L^2 r_a^2 + 110 E I L^2 r_a r_b^2 + 110 r_a L^3 r_b^2 \right) \right) \]

\[ M_{44} = \left( \frac{L^3 m r_a}{2} \left( 32 E^2 I^2 + 11 E I L r_a + L^2 r_a^2 \right) \right) \]

\[ \left( 105 \left( 12 E^2 I^2 + L^2 r_a r_b + 4 E I L r_a + 4 E I r_b \right)^2 \right) \]
References

[1] D. Balageas 2006, Structural Health Monitoring, london : ISTE Ltd.

[2] S. L. Chan 1994, “Nonlinear Vibration Analysis of Steel Frames with Semirigid Connections,” *ASCE journal of structural engineering*, Vol. 120, no. 4, pp. 1075-1086.

[3] Doebling 1996, “Damage Identification and Health Monitoring of Structural and Mechanical Systems from Changes in Their Vibration Characteristics: A literature review. Los Alamos National Laboratory, LA-13070-MS;,” United States.

[4] G. Monforton 1963, “Matrix analysis of semi-rigidly connected frames,” *ASCE Journal of structural engineering*, Vol. 89, no. 6, pp. 13-42.