CHAPTER: 3

FINITE ELEMENT ANALYSIS (FEA)

3.1 INTRODUCTION TO FEA

The Finite element analysis (FEA) is a numerical method for solving complicated structural systems that may be impossible to be solved in the closed form. The finite element analysis may be viewed as a general structural analysis procedure that allows the computation of stresses and deflections in 2D and 3D.

It acquired by its name based the approach used with in the technique; assembling a finite number of structural components or elements interconnected by a finite number of nodes. Any structure may be idealized as a finite number of elements assembled together in a structural system (i.e., discretizing a continuous system). The name, finite element analysis, arises because there exist only a finite number of elements in any given model to represent an actual continuum with an infinite number of degrees of freedom.

The main advantage of using the finite element analysis can be seen once the model is established. The structural geometry, material properties, boundary/support conditions and the loading conditions which change the response of the structure can be quickly established.

To study the response of the structural system due to these changes through the testing by means of experiments requires the construction and testing of other specimens. In addition, the FEA
model can be subjected to any boundary conditions and the same is very difficult to simulate by means of experimentation.

The other advantage of using the finite element analysis is that the slips between the individual structural components and the stresses in the individual structural components can be easily determined. To get the similar kind of information through testing by means of experiments requires additional instrumentation which may be costly and time consuming to set up and regulate.

The finite element analysis is also having certain drawbacks. Firstly, the quality of the results obtained from FEA depends up on the density of the mesh, element type and the input properties of the element and these modeling aspects usually increases as the structural system becomes complex. For example- 1) The increase in the number and the variety of structural components and its connections. 2) When the geometric non linearity’s and the material non linearity’s cannot be neglected 3) when the modeling shifts from one dimension to three dimensions. Also it is very important to validate the modeling and analysis strategies with the classical theories OR with experimental testing.

The second drawback of the finite element analysis is that the analysis necessitates very powerful software and an individual with strong basics of the finite element theory and the analysis techniques.
There are three basic phases that make up the finite element analysis procedure:

First phase is structural idealization in which the original/actual system is subdivided into assemblage of discrete elements and is a critical aspect in performing an accurate analysis. This is because for the idealization to provide a reasonable and accurate representation of the actual continuum, each element must be established so that it deforms similarly to the deformations that occur in the corresponding domain of the continuum. Otherwise, as load is applied, the elements would distort independently of one another, except at the nodes, and gaps or overlapping would develop along their edges. The idealization would therefore be much more flexible than the continuum. In addition, sharp stress concentrations would develop at each nodal point and the result would be an idealization that poorly resembled the actual structure. Thus, considering the deformation pattern of an element, and ensuring compatibility to adjacent elements’ patterns, is the most important criterion in performing this first phase.

The second phase is the evaluation of the element properties. This is the critical phase of the analysis procedure as it involves the setting up of the force-deflection relationship by use of a flexibility or stiffness matrix. The essential elastic characteristics of an element are represented by this force-displacement relationship which is a means of relating the forces applied at the nodes to the resulting nodal deflections.
The third phase of the finite element analysis procedure is the structural analysis of the element assemblage. As in any analysis, the main problem is to simultaneously satisfy equilibrium, compatibility, and force-deflection relationships. The basic operations for approaching this problem include the use of the displacement method which is easiest for dealing with highly complex structures.

3.2 ABAQUS SOFTWARE

The commercial multipurpose finite element software package ABAQUS (Version-6.6-3) is employed in this research. ABAQUS software has the ability to treat both the geometric and material nonlinearity that may rise in a given structural system.

Finite element tests were carried out using the theoretically and experimentally verified techniques involving the commercially available multipurpose finite element program ABAQUS. The parametric study involved changing the spans, lateral bracing stiffness with the resulting capacities being recorded and analyzed. It provides the user with an extensive library of elements that can model virtually any geometry. It has a wide variety of material models that can simulate the behavior of most typical engineering materials such as metals, composites, reinforced concrete, etc.

For the purpose of performing nonlinear analyses ABAQUS software is capable of automatically choosing appropriate load increments and convergence tolerances as well as continually adjusting them during the analysis to ensure that an accurate solution is obtained efficiently.
Use of the ABAQUS software is split up into three distinct stages: preprocessing in which all aspects of the model are defined through the creation of an input file, simulation in which the program actually solves the numerical problem defined in the model, and the post processing through which the results can be evaluated and analyzed in a variety of ways to assist the user. Assuming all three stages are conducted appropriately, ABAQUS software is capable of providing extremely reliable results for a wide variety of structural problems.

The following convention used for the displacement and rotational degrees of freedom in ABAQUS software and is shown in figure 3.1.

1. Translation in the 1-direction (U1).
2. Translation in the 2-direction (U2).
3. Translation in the 3-direction (U3).
4. Rotation about the 1-direction (UR1).
5. Rotation about the 2-direction (UR2).
6. Rotation about the 3-direction (UR3).

Figure 3.1 Displacement and Rotational degrees of freedom
3.3 SOURCE OF NONLINEARITIES IN STRUCTURAL RESPONSE

In linear analysis, the response is directly proportional to the load. Linearity may be a good representation of reality or may only be inevitable result of assumptions made for the analysis purposes. In linear analysis, the assumption are the displacements and rotations are smaller, stress is directly proportional to strain and the supports do not settle and the loads maintain their original directions as the structural system deforms.

The nonlinearity which presents in a structural system makes the problem more complicated because the equations that describe the solutions must incorporate the conditions not fully known until the solution is known-the actual structural configurations, loading conditions, state of stresses and the support conditions. The solutions cannot be obtained in a single step of analysis and will take several steps, update the tentative solution after each step and repeating until a convergence is satisfied.

There are three basic types of non linearity’s and they are 1) Geometric non linearity 2) Material non linearity and 3) the boundary nonlinearity.

The modeling and the analysis employed for the verification and parametric studies include the geometric non linearity’s and the material non linearity’s.
3.3.1 Geometrical nonlinearity

Geometric nonlinearity arises when the deformations are large enough to significantly alter the way load is applied or the way load is resisted by the structural system.

Geometric nonlinearity’s should be considered, especially when there is a large deformation and small strain case. Ignoring the effects of geometric nonlinearity makes the governing kinematic equations linear and thus it is impossible to capture the behavior such as lateral torsional buckling.

3.3.2 Material Nonlinearity

The stress-strain curve of steel is linearly elastic until some significant point called the yielding point. After the attainment of the yield point, the stress strain curve becomes non linear and the strains become partially irrecoverable. In other words when the material behavior does not fit the elastic model $\sigma = E\varepsilon$ there is a phenomenon of material nonlinearity. Effects due to the constitutive equations (stress-strain relations) that are non linear, are referred to as material nonlinearities.

Material nonlinearity is modeled using ABAQUS standard metal plasticity material model which is based on an incremental plasticity formulation employing associated flow assumptions in conjunction with a Von Mises failure surface whose evolution in stress-strain is governed by a simple isotropic hardening rule.
3.4 NON LINEAR FINITE ELEMENT ANALYSIS

The primary objective of a non linear finite element analysis is to find the state of equilibrium of a structure corresponding to set of applied loads. In such a non linear analysis, by solving a system of linear equations the solutions cannot be obtained. Rather, the load is specified as a function of “time”. (The term "time" is employed in general sense, since the FEA proposed herein are static and accordingly the time is a dummy variable employed to mean the increment of applied load). The “time” is divided into intervals and the same is applied incrementally in small steps in order to trace the non linear equilibrium response. The values of the accumulated time denote the load proportionality factors (LPF).

In the method of incremental analysis, each step is assumed to be linear with the displacement or loading applied in a series of increments. Each time a new displacement increment is evaluated and the result is added to the previous displacement of the structural system and is calculated for each incremental step. These increments in displacement that allow for the observation of changes in the overall model.

The finite element analysis program ABAQUS deal with geometric and material non linearity’s that may occur in modeled structures. ABAQUS software traces the non linear equilibrium path through an iterative approach. In the context of the current research program, the program loads the model with small load increments. ABAQUS software presumes the structural behavior to be linear
within each increment. After each increment loading, a new structural configuration is determined and a new idealized structural behavior (i.e tangent stiffness matrix) is considered with in each of these increments, the linear structural problem is solved for displacement increments using load increment. The incremental displacement results are subsequently are added to previous deformations (as obtained from earlier solution increments).

In ABAQUS software, the load increment is denoted by a load proportionality factor related to the applied load. For example the initial load increment may be 0.01 times the applied load, when 0.01 is the load proportionality factor, and a second load increment factor may be 0.02 times the applied load. The load proportionality factor may increase in size if the solution convergence rate appears to be more and more favorable with each increment. However, as the ultimate load for the structure is approached, the load increments are reduced in size. After each converged increment is obtained a new tangent stiffness matrix is computed using the internal loads and the deformation of the structure at the beginning of the load increment.

In non linear analysis the tangent stiffness matrix, is used as a means for relating changes in loads and the changes in displacement in a linearised fashion with an individual load increment (i.e, between two different LPF’s). The tangent stiffness matrix depends only up on the internal forces and deformation at the beginning of each load increment. This tangent stiffness can be represented by the following equation.
\[
\begin{align*}
\begin{bmatrix} K_T \end{bmatrix} &= \begin{bmatrix} K_0 \end{bmatrix} + \begin{bmatrix} K_p \end{bmatrix} \tag{3.1}
\end{align*}
\]

Where \( \begin{bmatrix} K_0 \end{bmatrix} \) is the usual linear stiffness matrix for uncoupled bending and force behavior and matrix \( \begin{bmatrix} K_p \end{bmatrix} \) is the initial stiffness matrix that depends upon the force at the beginning of each load increment (stress matrix).

### 3.5 Non Linear Equilibrium Equations

The virtual work may be caused by true forces moving through the imaginary displacements or vice versa. The principle of virtual work can be divided into the principle of virtual displacements and the principle of virtual forces.

The principle of virtual displacements is based on the virtual work done by the true forces moving through the virtual displacements. This principle establishes the conditions of equilibrium and the same is used in the displacement models of the finite element methods.

The principle of virtual forces uses the virtual work done by the virtual forces in moving through the true displacements and establishes the compatibility conditions and the same is used in the equilibrium models.

In the virtual displacement context, the principle of virtual work states that a structure is in equilibrium under the action of external forces (due to nodal forces, surface forces, body forces) for arbitrary virtual displacements forms a state of compatible deformations with
compatible strains and accordingly the virtual work is equal to the virtual strain energy.

This is stated as below.

\[ \delta U^{(e)} = \delta V^{(e)} \]  \hspace{1cm} (3.2)

Where \( \delta U^{(e)} \) is the virtual strain energy due to internal stresses, and \( \delta V^{(e)} \) is the virtual work of external forces on the element.

### 3.6 NON LINEAR FINITE ELEMENT SOLUTION TECHNIQUES

In the present study, it is compulsory to trace the non linear equilibrium path of the structural system because of the application of the incremental load. The non linear finite element solution techniques are used to establish the solution to the equation-3.2. The Newton Raphson’s and Riks Wempner are the commonly used incremental solution techniques. The Newton-Raphson method and the Riks-Wempner method are potentially good tools in establishing the nonlinear behavior of the structural system.

The Newton-Raphson solver traces the nonlinear equilibrium path by successively formulating linear tangent stiffness matrix at each load level. The tangent stiffness matrix changes at each load level due to a difference in internal force and applied external load (i.e., as a direct effect of the stress softening effects of \([K_p]\)).

The Newton-Raphson method is advantageous because of its quadratic convergence rate when the approximation at a given iteration is within the radius of convergence. ABAQUS software
usually uses the Newton Raphson’s method for solving the non linear equilibrium equations. The number of iterations needed to find a converged solution for a time increment will vary depending on the degree of non-linearity in the structural system. However, Newton-Raphson’s method fails around the critical points meaning it is unable to negotiate the solution features at the interface between stable and unstable equilibrium conditions, hence the same is not preferable to plot the unloading portion of the structural system in the non linear equilibrium state and the same is not recommended for this investigation. One solution method for tracing the nonlinear equilibrium path that is used in ABAQUS software in such circumstances is Riks-Wempner method. The Riks -Wempner method is also sometimes referred to as the arc length method. In arc-length methods, the solution is constrained to lie either in a plane normal to the tangent of the equilibrium path at the beginning of the increment or on a sphere with radius equal to the length of the tangent. This method allows tracing snap-through as well as snap-back behavior.

Since this study focuses on the inelastic capacity of web tapered I- members in bending in the presence of buckling influences, the solver of choice for this particular study is the modified Riks-Wempner method. Therefore, within the context of the current research, the Riks-Wempner method allows the web tapered I-beams to buckle and unload. This method also provides some of the most efficient use of the computational resources during the nonlinear
response since step size in an increment is tied to the convergence rate from the previous increment.

3.7 RIKS-WEMPNER METHOD (The arc length method)

The Riks Wempner method is usually used for predicting the unstable state, geometrically non linear collapse of the structural system. For investigation of this kind of behavior the load–defection curve is to be established and it is essential to incorporate a initial geometrical imperfection in to the finite element model. The very purpose of incorporating the initial geometrical imperfection is to perturb the finite element model from the condition of perfect geometry. There are basically three ways to incorporate the initial geometrical imperfection and they are :- 1) Superposition of buckling modes from the buckling analysis. 2) Developing the model from the displacements of the static analysis. 3) By applying a small torque. The non linear analysis is carried out by considering the two analysis runs with the same definition of the finite element model. In the first run, a linearised eigen value buckling analysis is carried out on a perfect structural system in order to verify whether the finite element mesh discretizes the modes correctly and to establish the probable collapse modes. During the second run, an initial geometric imperfection is added to the buckling mode under consideration to the perfect geometry by utilizing the” IMPERFECTION” command. The imperfections are scaled to a suitable value and are added to the
perfect geometry. Thus imperfection has the form

$$\Delta x_i = \sum_{i=1}^{m} \omega_i \phi_i,$$

where $\phi_i$ is the $i^{\text{th}}$ mode shape and $\omega_i$ is the associated scale factor. ABAQUS software then performs a geometrically nonlinear load-displacement analysis of the structure containing the imperfection using the Riks-Wempner method. In simple cases, linear eigenvalue analysis may be sufficient for design evaluation, but if there is any concern about material nonlinearity, geometric nonlinearity prior to buckling, or unstable post buckling response, a load-deflection, Riks analysis must be performed to investigate the problem further. The load magnitude is used as an additional unknown in the Riks method. The loads and displacements are solved simultaneously. For this reason, another quantity must be used to measure the progress of the solution. ABAQUS software uses the arc length along the static equilibrium path in load-displacement space. This approach provides solutions regardless of whether the response is stable or unstable. As previously mentioned, when the nonlinear static equilibrium solution for unstable problems is desired, ABAQUS software uses the Riks-Wempner method. In such cases, ABAQUS software allows the effective solution to be determined for situations in which the load and/or the displacement may decrease as the solution evolves. This typical unstable static response is illustrated in figure 3.2.
In the modified Riks method, it is assumed that all load magnitudes vary with a single scalar parameter, the loading is proportional to this parameter. The response is additionally assumed to be reasonable smooth and the sudden bifurcations do not occur. The essence of the method is that the solution is viewed as the discovery of a single equilibrium path in a space defined by the nodal variables and the loading parameter. Tracing this path as far as required allows the development of the solution. It is essential to limit the increment size due to the fact that many of the materials, and possibly loadings of interest will have path-dependent response. The increment size for the modified Riks algorithm is limited by moving a given distance along the tangent line to the current solution point and then searching for equilibrium in the plane that passes through the point thus obtained and that is orthogonal to the same tangent line.
3.8 VON MISES YIELD CRITERION

Richard Von Mises, a German-American Mathematician developed maximum distortion energy criterion. This later came to be known as the Von Mises Yield Criterion. This criterion is based on the determination of the distortional energy in a given material (i.e., the energy associated with the change in the material as opposed to the energy associated with the change in the volume of the same material). According to this criterion, a given structural component is elastic as long as the maximum value of the distortion energy per unit volume required to cause yield, such values may be obtained experimentally.

Von Mises stress is used as a criterion in determining the onset of failure in ductile material such as steel. The failure criterion states that the Von Mises stress should be less than the yield stress of the material.

When developing the yield criterion mathematical model, certain assumptions should be considered and they are as follows: 1) The material may be assumed as isotropic 2) the bauschinger effect may be ignored 3) the uniform hydrostatic compression or tension does not have an effect on yielding.

Thus, the von Mises yield criterion forms a cylinder encompassing the entire hydrostatic axis. The radius of the cylinder represents the deviatoric component (Only stresses that deviate from the hydrostatic stress state, referred to as deviatoric stresses,
influence and cause yielding in a ductile metal) of the strain tensor associated with initiation and propagation of yielding in the material.

A geometrical representation of the yield criterion in principal stress space and for biaxial stress space is shown in figure 3.3 and 3.4, respectively.

Figure 3.3 Yield Surface in Principal Stress Space

Figure 3.4 Yield Surface for biaxial stress state ($\sigma_3 = 0$)
The elastic state of stress is defined as being any point inside the cylinder, and yielding is defined as any state of stress that permits the stress point to lie on the surface of the cylinder.

According to the Von Mises criterion, yielding will occur when the distortional strain energy density of the structure (or at a point) reaches the distortional strain energy density at yield in uniaxial tension or compression.

The distortional energy per unit volume, or the distortional strain energy density, can be obtained from the total strain energy density, \( U_O \). The total strain energy density can be broken into two parts: one part that causes volumetric changes, \( U_V \), and one that causes distortion, \( U_D \), and is defined in the principal stress state.

\[
U_O = U_V + U_D \tag{3.3}
\]

\[
U_V = \frac{(\sigma_1 + \sigma_2 + \sigma_3)^2}{18K} \tag{3.4}
\]

\[
U_D = \frac{(\sigma_1 - \sigma_2)^2 + (\sigma_2 - \sigma_3)^2 + (\sigma_3 - \sigma_1)^2}{12G} \tag{3.5}
\]

\[
(\sigma_1 = \sigma_2 = \sigma_3)
\]

Where \( K = \frac{E}{3(1-2\nu)} \) and \( G = \frac{E}{2(1+\nu)} \)

The first term on the right hand side of the above equation is \( U_V \), the strain energy that is associated with the pure volume change
and can be neglected because it is known that hydrostatic pressure does not have effect on yielding. The second term is the distortional strain energy density and is defined.

Under a uniaxial stress state stress, \( \sigma_1 = \sigma \) and \( \sigma_2 = \sigma_3 = 0 \) at yield,

\[
U_D = U_{DY} = \frac{(\sigma_Y)^2}{6G} \quad \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots (3.6)
\]

Thus, for a multi axial stress state, the distortional energy density criterion states that yielding is initiated when the distortional energy density \( U_D \) given by equation-(3.5) equals, or, failure occurs when the energy of distortion reaches the same energy for yield/failure in uniaxial tension.

\[
U_{DY} = \frac{(\sigma_Y)^2}{6G} \quad \ldots \ldots \ldots \ldots \ldots \ldots \ldots \ldots (3.7)
\]

The ellipse represents the yield surface boundary. The area within the ellipse corresponds to the material behaving elastically and anything outside of the ellipse corresponds to yielding of the material.

After attaining the yield point, many materials such as steel indicates an increase in stress with the increase in strain with a flatter slope than the original elastic slope. Also the steel material seems to have the increase in yield stress after unloading and reloading. This increase in stress is called as strain hardening. The increase in yield point also means a change in the yield surface.
Isotropic hardening means that the yield surface changes its size uniformly in all directions such that the yield stress increases (or decreases) in all stress directions as plastic straining occurs. ABAQUS provides an isotropic hardening model, which is useful for cases involving gross plastic straining or in cases where the straining at each point is essentially in the same direction in strain space throughout the analysis.

The Von Mises yield surface is used to define isotropic yielding in ABAQUS software. It is defined by giving the value of the uniaxial yield stress as a function of uniaxial equivalent plastic strain.

3.9 IMPLEMENTATION OF MATERIAL PLASTICITY IN ABAQUS SOFTWARE

True-stress versus true-strain (logarithmic strain) characteristics of the material is used in nonlinear finite element analysis since nonlinear finite element formulations permit the consideration of updated structural configurations.

Engineering stress-strain response does not give a true indication of the deformation characteristics of a structural steel because it is based entirely on the original dimensions of a given specimen. Ductile materials, such as steel, exhibit localized geometric changes and therefore, the relevant stress and strain measures are different from the measured engineering stress and strain values.

Engineering stress is calculated using the original, undeformed, cross-sectional area of a specimen. The engineering stress for a
uniaxial tensile or compressive test has a magnitude \( \sigma_{\text{eng}} = \frac{P}{A_O} \), where "\( P \)" is the force applied and "\( A_O \)" is the original cross-sectional area. Engineering strain is the change or elongation of a sample over a specified gage length "\( L \)". The engineering strain is equal to \( \varepsilon = \frac{e}{L} \), where "\( e \)" is the elongation of the material over the gage length.

When employing nonlinear finite element modeling strategies considering nonlinear material effects, it is important to use true stress and true strain (logarithmic strain) when characterizing the material response within the finite element environment. True stress and true strain is required by ABAQUS software in cases of geometric nonlinearity because of the nature of the formulation used in the incremental form of the equilibrium equations in ABAQUS software.

The change in the specimen’s cross sectional area may be an important consideration when large deformations occur. Under such circumstances the strain hardening range should be described using the true stress.

The true stress may be presented conceptually by the following equation:

\[ \sigma_t = \frac{P}{A_t} \], where "\( A_t \)" = the actual cross-sectional area of the sample specimen when the load "\( P \)" is acting on it.
Engineering stress and strain data for uniaxial test for isotropic material can be converted into true stress and true strain (logarithmic plastic strain) by using the following equations.

\[
\sigma_t = \sigma_{\text{eng}} \left(1 + \sigma_{\text{eng}}\right) \quad \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdots \cdOTS
The formulation of the engineering strain is shown by below equation.

\[ \varepsilon_t = \frac{L^t}{L} = \ln \left( \frac{L^t}{L} \right) = \ln \left( \frac{L+e}{L} \right) = \ln \left( 1+\sigma_{\text{eng}} \right) \]

\[ \text{…………………… (3.9)} \]

The logarithmic plastic strain may be expressed as follows:

\[ \varepsilon_{\text{ln}}^{\text{pl}} = \ln \left( 1+\sigma_{\text{eng}} \right) - \frac{\sigma_t}{E} \]

Furthermore, Logarithmic strain is a more appropriate strain measure to use in geometrically nonlinear finite element problems.

A yield surface in three-dimensional principal stress space is extrapolated from this information using the Von Mises yield criterion. The input file must ensure that the material is adequately defined for the purpose of the analysis. The material specifications in the input file must include both elastic and plastic properties. The elastic properties are entered into the input file by specifying the Young’s modulus “E” and Poisson’s ratio “\(\nu\)”. For the current study, “E” and “\(\nu\)” are equal to 200,000 MPa (29,000 ksi) and 0.3, respectively. The plastic values are specified as points along the true stress versus true strain plot shown in figure 3.5 and given in table 3.1. The plastic properties for the steel used in all the models are a amalgam of values provided by Salmon and Johnson [3], experimental data, and are based on the research publications. [108, 113, 122]. ABAQUS utilizes the uniaxial material properties to extrapolate a yield surface in 3D principal stress space.
Table 3.1 True Stress versus True Strain (Logarithmic Strain)

<table>
<thead>
<tr>
<th></th>
<th>Logarithmic Plastic Strain</th>
<th>True Stress (Ksi)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Yielding</td>
<td>0.0000</td>
<td>50.000</td>
</tr>
<tr>
<td>Strain Hardening</td>
<td>0.0092</td>
<td>51.345</td>
</tr>
<tr>
<td>Strain Hardening</td>
<td>0.0557</td>
<td>75.000</td>
</tr>
<tr>
<td>Ultimate</td>
<td>0.0900</td>
<td>80.000</td>
</tr>
<tr>
<td>Rupture</td>
<td>-</td>
<td>-</td>
</tr>
</tbody>
</table>

1 Kip = 4.45 KN and 1” = 25.4 mm.

3.10 FINITE ELEMENT MODELING CONSIDERATIONS

The ABAQUS software (FEA) was used for the verification study in which a number of finite element modeling techniques were applied and evaluated. This was done for the purpose of determining the most accurate modeling approach for predicting actual response. The different modeling parameters considered were:

1) Different types of shell elements
2) Mesh density
3) Geometric imperfections
4) Lateral bracing stiffness

The verification of analytical model developed for this study was built using the ABAQUS -S4R shell elements as those used for the parametric study reported here in.

The flexible bracing was modeled through the application of the ABAQUS SPRING1 element to the centre of top flange width (compression flange) at mid span.
3.11 FINITE ELEMENT MODELING DETAILS

The commercially available multipurpose finite element program ABAQUS has been employed in all of the numerical studies reported herein. A nonlinear shell element is chosen for this study so as to be able to explicitly model local buckling deformations and the spread of plasticity effects. A shell element is suitable for “thick” or “thin” shell applications utilizing reduced integration.

Earls and Shah [112] considered both the S4R and S9R5 shell elements from the ABAQUS element library in their verification work. Their verification study demonstrated that the S4R element had a better agreement with the experimental work. Thus, the models considered in the verification study and the parametric studies are constructed from a mesh of S4R shell finite elements.

S4R elements were selected for use in this Verification study and parametric study based on the ability of this shell element to accurately model local buckling deformations, large rotations, shear flexible, reduced integration, finite strains and capable to model torsion, in and out of plane bending as well as shear in the elastic and the plastic regime.

These elements are oriented along the planes of the middle surfaces corresponding to the constituent plate components of the members. The restraint in the out-of-plane direction are enforced at purlin/girts, flange braces locations through the use of ABAQUS’ SPRING1 elements. These elements act as linear springs possessing a
stiffness that is specified in the input file. The stiffness values are varied uniformly as part of the parametric study.

3.12 SHELL ELEMENTS

Shell elements are used in the verification study and the parametric study because of their ability to model the structural system in which the thickness is very small as compared to the other dimensions and the stresses normal to the direction of the thickness are negligible. The S4R elements are selected in this study. The order of numbering the nodes is illustrated in Figure 3.6.

Figure 3.6 Numbering order for a 4-Noded Element

Figure 3.7 S4R –ELEMENT
The S4R element is defined by ABAQUS as a 4-node, doubly curved general purpose shell with reduced integration, finite member strains and hourglass control. The following are the aspects of an element that influence its response (ABAQUS). They are as follows:-

1. The element family
2. Degrees of freedom
3. Number of nodes
4. Formulation
5. Integration

The S4R element is part of the “shell” family. Two types of shell elements are used in general and they are “thick” and “thin”. Thick shell elements are needed in situations where transverse shear flexibility is vital and the second order interpolation is required (ABAQUS).

The thin shell elements are needed in situations where the transverse shear flexibility is insignificant and the Kirchhoff constraint must be satisfied accurately (i.e., the shell normal remains orthogonal to the shell reference surface) (ABAQUS). The S4R is a 4-noded, general purpose element which allows for thickness changes. The S4R uses thick shell theory as the shell thickness increases and become Kirchhoff thin shell elements as the thickness decreases; the transverse shear deformation becomes very small as the shell thickness decreases.

In addition, the S4R is suitable for large-strain analysis involving materials with a non zero effective Poisson’s ratio.
First order (lower order) shear deformation theory is the basis for the formulation of S4R elements, means the shell employs the linear displacement and the rotation interpolation in the context of Mindlin-Reissner theory, but the shear deformations are then obtained directly from a consideration of the nodal deformations. This approach is made to be consistent with the assumption that cross-sections remain plain but not normal to the Gauss surface of the shell.

The degrees of freedom for a shell element are the displacements and rotations at each node. The active S4R degrees of freedom are shown below

$$1, 2, 3, 4, 5, 6 (u_x, u_y, u_z, \phi_x, \phi_y, \phi_z)$$

The S4R element uses reduced integration to form the element stiffness. In the reduced integration technique, the order of in-plane integration is one integration order less than that which would require performing the stiffness matrix integration exactly.

Reduced integration usually provides results that are more accurate (as a means of overcoming some over stiffness elements in the shell, relieves shear locking provided the elements are not distorted or loaded in in-plane bending) and significantly reduces running time, especially in three dimensions (ABAQUS). The S4R element is computationally less expensive since the integration is executed at one Gauss point per element.
The disadvantage of the reduced integration procedure is that it can admit deformation modes that cause no straining at the integration points. The zero energy mode starts propagating through the mesh, leading to cause of an unstable solution or inaccurate solutions. This problem is particularly severe in case of first-order quadrilaterals. However, the ABAQUS software overcomes this difficulty by considering the Hourglass control. The Hourglass control uses an additional artificial stiffness which is added to the elements.

3.13 SPRING ELEMENTS

The flexible lateral bracing was modeled through the application of the ABAQUS SPRING1 element at locations that correspond to the bracing location. The rigid out-of-plane restraints are provided at the supports. These elements act as linear springs possessing a stiffness that is specified in the input file. The stiffness values are varied uniformly as part of the verification study and parametric study. Figure 3.8 shows a schematic of how this bracing was idealized in the finite element model.

Figure 3.8 Idealized Lateral bracing using ABAQUS SPRING 1-element
3.14 Finite Element Mesh

The numerical models proposed for this research are developed by considering a dense mesh using the S4R element type. The density of the mesh is directly proportional to computational time and modeling accuracy. These considerations must be kept in mind so that the model generates relatively good results within the reasonable amount of time.

The density of mesh considered was demonstrated to provide accurate results at the local and global level. Equally sized elements in the top flange, bottom flange and the web of each numerical model make the structural components compatible. This means the mesh of the top flange and bottom flange can be integrated with the mesh of the web. By performing this activity, one can tie the meshes of top flange, bottom flange and web elements together and the same will act like a single component.

3.15 Source of Modeling Uncertainties

There will be always some uncertainties between the experimental testing and the finite element models/analysis. For the experimental testing, material yield stress, stress strain relationship, the geometry of the plates which are used to fabricate the cross sections and the lengths of the structural members, residual stress patterns, mis-measured geometrical imperfections, additional restraints or the slips in the experimental testing do have the affects on the results obtained by the finite element models/analysis. In finite
element modeling the structural engineer/researcher must define the information listed below.

- Initial geometry of the specimen with imperfections
- Boundary conditions
- Mesh density
- Element Type
- Material model with material stress strain properties
- Numerical solution procedure with convergence tolerances.

### 3.16 IMPERFECTION SEED

In modeling studies where inelastic buckling is investigated, it is important that the evolution of the modeling solution will be carefully monitored so that any indication of bifurcation in the equilibrium path is carefully assessed in order to try and ensure that the equilibrium branch being followed corresponds to the lowest energy state of the system [112]. The strategy of seeding the finite element mesh with an initial displacement field is employed in this study for guaranteeing that the lowest energy path is taken. The initial displacement field is obtained by conducting the buckling analysis. It should be noted that the first buckling mode is not always the correct mode to consider. In the present investigation, it was compulsory to look and study each of the buckling modes from the multiple buckling modes which are obtained by the buckling analysis.

From the linearized eigen value buckling analysis, the desired displacement field is selected as a imperfection seed and is scaled to $L/1000$ (i.e., $L_b/500$ during the parametric studies) as a initial
displacement and then superimposed on the finite element model in order to carry out the non linear response of the considered structural system.