Project Report

on

Stability of Steel Columns at Elevated Temperatures: A Finite Element Study

2nd December, 2022

Course Title: Finite Element Method in Structures (CVL757)

Submitted by Group 8:

S Lalit Sagar: *2022CEZ8119* (PhD, 1st year)

Amitesh Shrey: *2019CE10215* (B.Tech. – M.Tech. Dual Degree, 4th year)

Saket Pakhale: *2019CE10297* (B.Tech. – M.Tech. Dual Degree, 4th year)

Sarthak Shrivastav: *2019CE11076* (B.Tech., 4th year)

Under the guidance of:

Dr. Vasant Matsagar,

Professor, Dept. of Civil Engineering,

IIT Delhi

INDIAN INSTITUTE OF TECHNOLOGY DELHI

DEPARTMENT OF CIVIL ENGINEERING

Hauz Khas, New Delhi – 110016

CONTENTS

1. Introduction to stability of columns

Buckling of columns:

In the design of columns, in addition to strength and serviceability criteria, stability criteria has to be satisfied. Stability is defined in terms of the change of configuration of the structure under a given loading.

Euler Buckling Theory:

Leonhard Euler in 1757 gave a theory of buckling of columns. Buckling is the change in shape or loss of stability of equilibrium configuration without fracture or separation of material in transverse direction due to axial loading (Cook, 2002). Euler's critical load for buckling for a column pinned at both ends is given as follows:

$$P_{cr} = \frac{\pi^2 EI}{{L_e}^2} \tag{1.1}$$

Where,

E is Youngs Modulus of column, I is the moment of inertial of column cross section and L_e is the effective length of the column.

Further, the stress corresponding to the buckling load is given by:

$$\sigma_{cr} = \frac{\pi^2 E}{(L/r)^2} \tag{1.2}$$

Where, r is the radius of gyration.

Extension of Euler's Buckling Load for other end conditions:

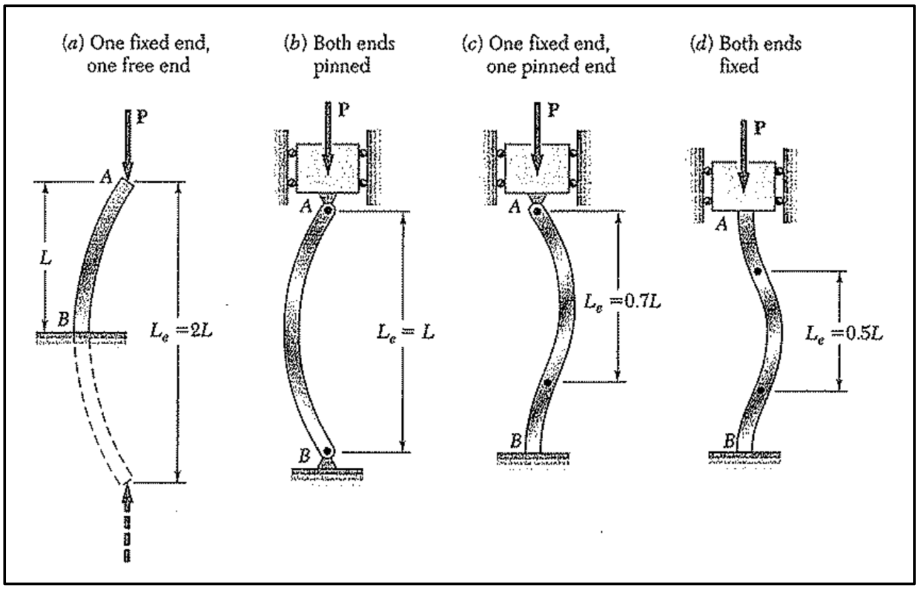


Figure 1: Effective length of columns for basic end conditions (Beer & Johnston, 2007)

The Euler's formula in Eq. (1.1) was derived assuming that the column was pinned at both ends. The formula has been extended to other end conditions as shown in Figure 1.

Assumptions used for deriving Euler's load:

1. Initially the column is perfectly straight, homogeneous and isotropic and obeys Hooke's law.
2. Cross section of the column is uniform (prismatic) throughout the length.
3. The load is axial and passes throughout the centroid.
4. The weight of the column is neglected.
5. The direct stress is small as compared to bending stresses.
6. Ends of the columns are frictionless.

The main limitation of the Euler's formula is that it is applicable only for simplistic cases, for columns having uniform geometric and material properties with standard boundary conditions. Thus, the need for Finite Element (FE) formulation to address the stability problem arises.

2. The need for Finite Element analysis

FE analysis is considered for the following reasons:

1. To reduce the computational complexity of large problem. To obtain numerical solutions with minimal error, where a closed-form solution is cumbersome to obtain.
2. Modelling geometric non-linearities. There is availability of many finite elements for domain discretization. Hence, the modelling of complicated geometries and asymmetrical forms is made simpler.
3. Account for material non-linearities and simulate different kinds of material properties.
4. Attaining better convergence through higher order elements (p-refinement) and more number of elements (h-refinement).
5. Applied loads can consist of pressures, concentrated forces, nonzero prescribed displacements, and/or thermal loading.
6. Analysing buckling mode at failure by formulating it as an eigen value problem.
7. Post-buckling analysis with unstable response, accounting for softening (Example: Stability of columns under elevated temperatures due to unintended fire).

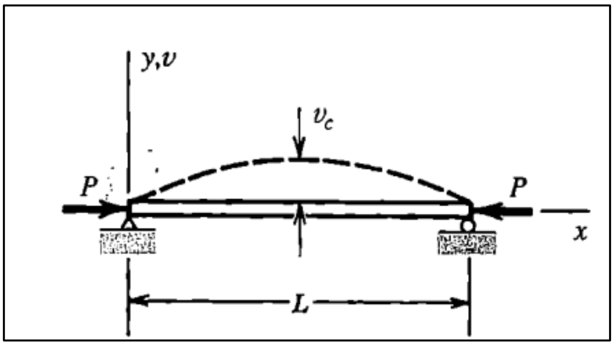
3. Stress stiffness and buckling

Stress stiffness is taken into consideration in formulating the FE problem for buckling analysis. Stress stiffening refers to the phenomenon of membrane forces influencing the lateral deflection that results from the bending of beams, plates, and shells. Here, the membrane forces acts along the axis of the beam, bar or tangent to the mid surface of plate or shell. Due to the membrane effect, when structure is in compression the resistance to bending deformation is reduced, and increased when a member is in axial tension.

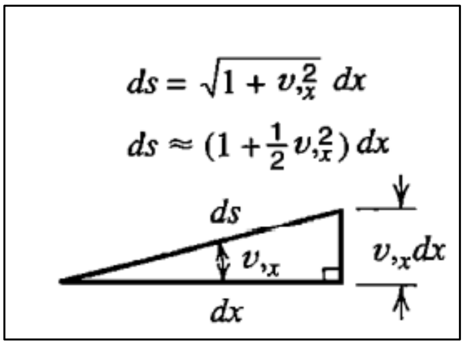
This results in buckling, in which the membrane strain energy gets transformed into bending strain energy without any change in the external load. In slender member with higher membrane stiffness, significant strain energy is stored in small deformations. Thus, when these slender members buckle, they undergo large deformation to accommodate the release of membrane strain energy (Cook, 2002).

In finite element method to take care of stress stiffening effect we have $[k_\sigma]$ matrix which augment the conventional stiffness matrix [k]. $[k_\sigma]$ matrix is applicable regardless of material anisotropy and yielding because it depends only on the geometry, displacement field, and the state of membrane stress.

Analysis of Beam-Column element for buckling load (Variational approach):



(a)



(b)

Figure 2: (a) A uniform beam simply supported; (b) Geometric relations for deformation of the differential element of length *dx* (Cook, 2002).

For small lateral displacement, strain energy in bending is given by:

$$U_b = \frac{1}{2}\int_0^L EI_z v_{,xx}^2\, dx = \frac{\pi^2 EI_z}{4L^3} v_c^2 \tag{3.1}$$

Strain in beam can be expressed as:

$$\varepsilon_m = \frac{ds - dx}{dx} = \frac{ds}{dx} - 1 \tag{3.2}$$

Which implies $$\varepsilon_m \approx \frac{1}{2} v_{,x}^2$$

(3.3)

For constant axial force P, membrane energy is given by:

$$U_m = -\int_0^L P\varepsilon_x \, dx = -\frac{\pi^2 P}{4L} v_C^2$$

(3.4)

Total potential energy is: $\Pi_p = U_b + U_m$

(3.5)

The equilibrium state is given by: $\partial \Pi_P / \partial v = 0$;

$$(k + k_\sigma) v_c = 0$$

(3.6)

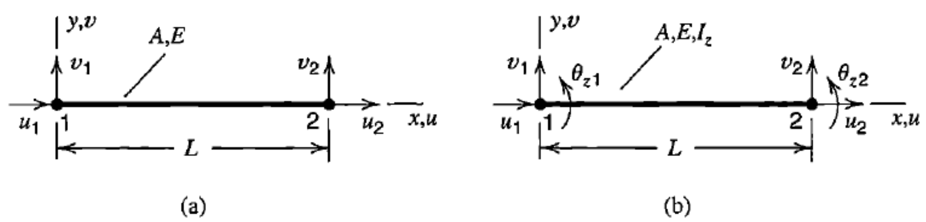
$$\text{where } k = \frac{\pi^4 E I_z}{2L^3} \text{ and } k_\sigma = -\frac{\pi^2 P}{2L}$$

Therefore, since the displacement v_c is non-zero, we get critical load as

$$P_{cr} = \frac{\pi^2 E I_z}{L^2}$$

(3.7)

<u>Derivation of Stress Stiffness Matrix for Bar and Beam element:</u>



<u>Figure 3:</u> Finite element and the corresponding DOF: (a) Bar element; (b) Beam element (Cook, 2002).

Let the nodal displacements be {d}, lateral displacement v and rotation be $v_{,x}$. Thus

$$v = [N]\{d\} \quad \text{and} \quad v_{,x} = [G]\{d\} \qquad \text{where} \quad [G] = \frac{d}{dx}[N]$$

(3.8)

Therefore, we can write membrane strain energy as:

$$U_m = \frac{1}{2}\int_0^L v_{,x}\, P v_{,x}\, dx = \frac{1}{2}\{d\}^T[k_\sigma]\{d\}] \tag{3.9}$$

Where, $[k_\sigma]$ is the stress stiffness matrix given as:

$$[k_\sigma] = \int_0^L [G]^T[G] P\, dx \tag{3.10}$$

<u>Bar element</u>: For displacement v_1 *and* v_2 , the stress stiffness matrix is derived as:

$$[k_\sigma] = \frac{P}{L}\begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{3.11}$$

<u>Beam Element</u>: For displacement $v_1,\ \theta_1,\ v_2,\ and\ \theta_2$ the stress stiffness is given as:

$$[k_\sigma] = \frac{P}{30}\begin{bmatrix} 36 & 3L & -36 & 3L \\ 3L & 4L^2 & -3L & -L^2 \\ -36 & -3L & 36 & -3L \\ 3L & -L^2 & -3L & 4L^2 \end{bmatrix} \tag{3.12}$$

4. <u>Motivation</u>

Columns are integral to the stability of structures. Design of a column is crucial because when the limit state of stress is reached, it leads to failure of the structure, without giving any warning. Large building fires have led to structural failure and collapse in the past (Fischer et al. 2019). Critical strength of a column reduces drastically when subjected to elevated temperatures through accidental fire.

The objective of the *structural fire engineering* is to determine the design strength of various building components such as axial force members, flexural members as well as connections under elevated temperatures. The outcomes of such studies may be incorporated into building codes and construction manuals. In addition to the practice of fire-resistant design of structures, protective materials can be used in the areas vulnerable to excessive thermal stresses.

5. Objectives

Stability of column using finite element approach under elevated temperatures has been studied in the following parts:

(a) Critical load of the column, formulated as an eigen value problem, and the study of mesh convergence.
(b) Stress analysis of columns under elevated temperatures, and a comparative study with critical stresses provided in the literature and design manuals.
(c) Non-linear analysis of columns using Modified Riks approach in Abaqus.

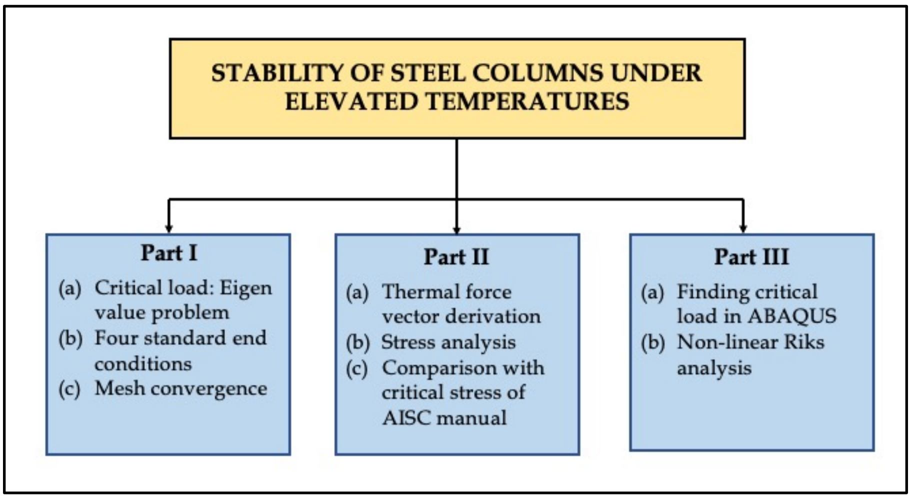


Figure 4: Framework of the FE study undertaken

6. Determination of critical load of column

Procedure to extract critical load (P_{cr}):

In the finite element formulation, stress stiffness matrix is considered in the evaluation of critical load. Linear bifurcation buckling is considered to be the mode of failure. Cook (2002) has derived the expression shown in Equation 5.1 to determine the (smallest root) critical load corresponding to linear bifurcation buckling.

While Equation 5.1 is an eigen value problem, {δD} corresponds to the eigen-vector associated with the buckling load λ_{cr}.

$$[\,[K] + \lambda_{cr}[[K]_{\sigma}]_{ref}\,]\ \{\delta D\} = \{0\} \tag{6.1}$$

The physical interpretation of the Equation 5.1 is as follows: The term in the parentheses which represents the net stiffness is reduced to zero, at buckling mode corresponding to the mode shape {δD}.

Mathematically the matrix in the parentheses is singular, and its determinant is zero.

Formulation of eigen value problem and mesh convergence study:

Column of the following geometric and material properties, and the initial axial load were considered for the buckling analysis:

- Length, L = 1.2 m
- Diameter, d = 0.05 m
- Modulus of elasticity, E = 2 x 10^5 MPa
- Axial load, P = 100 kN

Using the procedure described in the earlier section, the buckling load is determined by considering one, two, four and eight elements. The results are represented in Table 1 and Figure 5.

Table 1: Critical load for various end conditions and discretization (P_{cr}, kN)

Boundary Conditions	One Element	Two Elements	Four Elements	Eight Elements
BC 1: Fixed-Fixed	-	1704.4231	1694.8485	1683.0582
BC 2: Fixed-Pin	1278.3173	882.4139	862.1057	860.4524
BC 3: Pinned-Pinned	511.3269	423.7111	420.7624	420.5621
BC 4: Fixed-Free	105.9256	105.1885	105.1373	105.1373

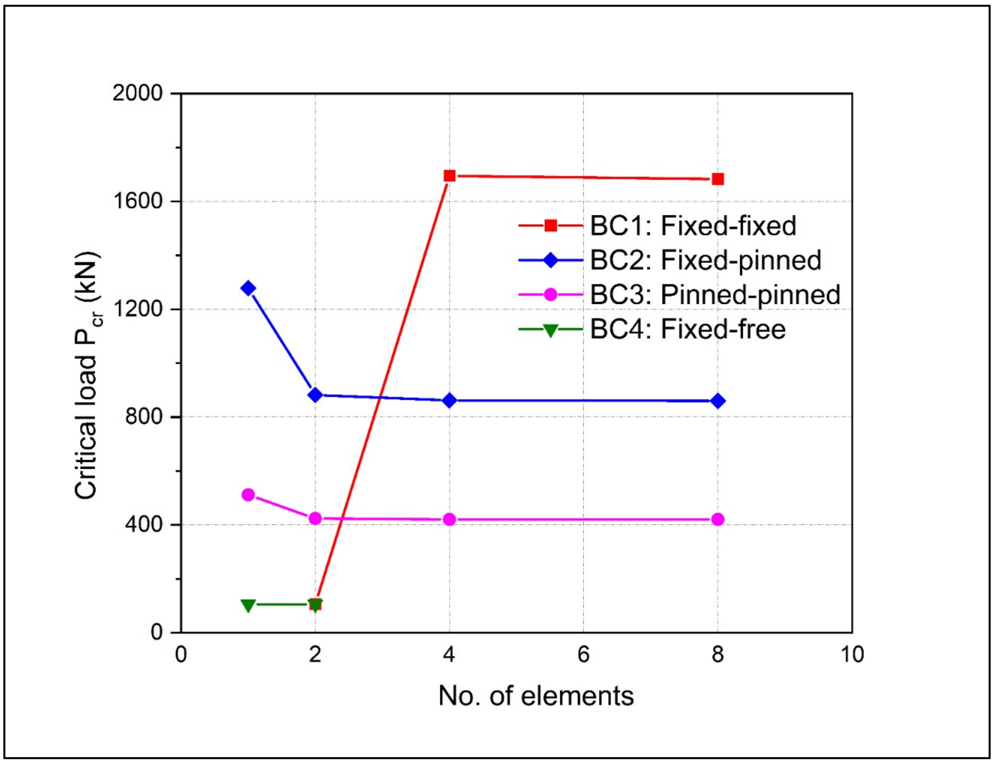


Figure 5: Critical load and mesh convergence

6. Stress analysis of columns under elevated temperatures

Thermal stresses are induced in structures due to temperature rise and constraints to the free expansion of the structure. Temperature changes in a structure can result in large stresses if not considered properly in design. For example, in bridges, unnecessary constraint of beams and slabs can result in large compressive stresses and resulting buckling failures due to temperature changes. In statically indeterminate trusses, members subjected to large temperature changes can result in large thermal stresses in truss members.

Thus, temperature rise is accounted for in the structural analysis through thermal force vector, similar to other types of loading such as: concentrated force, distributed force, and displacement based loads. In the present discussion, thermal force vector is determined using the variational approach by the minimization of potential energy (Logan, 2007).

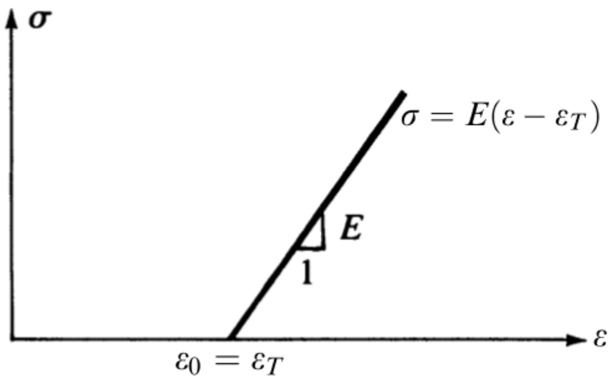


Figure 6: Stress-strain behaviour of a material under thermal strain

The strain energy per unit volume (called strain energy density) is the area under the σ & ε diagram in Figure 4 and is given by:

$$u_0 = \frac{1}{2}\sigma(\varepsilon - \varepsilon_T)$$

(6.1)

The total strain energy in the body is given by:

$$U = \int_V u_0 \, dV$$

(6.2)

Substituting Equation 6.1 in 6.2 and using the strain displacement relation, $u = B\{d\}$, get:

$$U = \frac{1}{2}\int_V (Bd - \varepsilon_T)^T\, D\, (Bd - \varepsilon_T) dV \tag{6.3}$$

Simplifying Equation 6.3 we get:

$$U = \frac{1}{2}\int_V (d^T B^T D B d - d^T B^T D\, {\varepsilon_T}^T - \varepsilon_T^T D B d + \varepsilon_T^T D \varepsilon_T\,)\, dV \tag{6.4}$$

In Equation 6.4, the first term U_L corresponds to strain energy due to mechanical loading and the second and third terms U_T corresponds to strain energy due to thermal loading.

Using the principle of minimum potential energy, minimization of U_T leads to the thermal force vector.

$$\frac{\partial U_T}{\partial d} = \int_V (B^T D \varepsilon_T)\, dV = \{f_T\} \tag{6.5}$$

Using the generalised Equation 6.5 for deriving the thermal force vector, the vectors for the bar and beam element are obtained as follows:

$$\textbf{\textit{Bar element:}}\ \{f_T\} = EA\alpha(\Delta T)\begin{Bmatrix} -1 \\ +1 \end{Bmatrix}\begin{matrix} u_1 \\ u_2 \end{matrix}$$

$$\textbf{\textit{Beam element:}}\ \{f_T\} = EI\alpha(\Delta T)\begin{Bmatrix} 0 \\ -1 \\ 0 \\ +1 \end{Bmatrix}\begin{matrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{matrix} \tag{6.6}$$

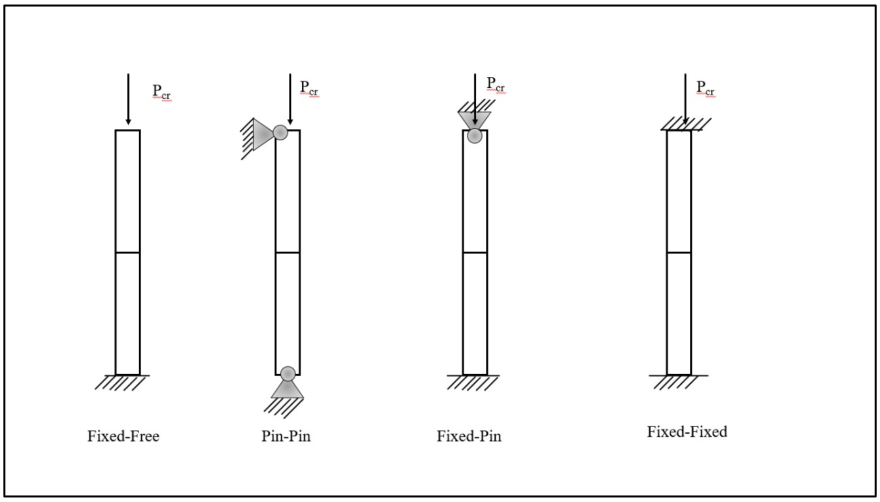


Figure 7: Finite element formulation for different end conditions

Formulation of the thermal stress problem and comparison with critical stress in the literature:

In this part, columns with four standard boundary conditions were considered for stress analysis at elevated temperatures.

Column was discretized into two six-noded beam elements. The same data from the previous section was adopted for the analysis:

- Length, L = 1.2 m
- Diameter, d = 0.05 m
- Modulus of elasticity, E = 2 x 10^5 MPa
- Axial load, P = 100 kN

The columns were axially restrained at both the ends for the development of thermal stresses. Further, the columns were subjected to a uniform temperature rise of 10° C, 20° C, 30° C, 40° C, and 50° C. The values of maximum stresses have been presented in Tables 2 (a)–(d), and Figures 8 (a)-(d).

Table 2: Variation of maximum stresses (σ_{cr}) with end conditions and temperature rise

Eccentricity of loading (%)	Change in temperature (° C)					
	0	10	20	30	40	50
0.000	25.465	50.065	74.665	99.265	123.865	148.465
0.075	40.744	65.344	89.944	114.544	139.144	163.744
0.150	56.023	80.623	105.223	129.823	154.423	179.023
0.225	71.301	95.901	120.501	145.101	169.701	194.301
0.300	86.580	111.180	135.780	160.380	184.980	209.580

(a) BC 1: Fixed-Fixed

Eccentricity of loading (%)	Change in temperature (° C)					
	0	10	20	30	40	50
0.000	25.465	49.465	73.465	97.465	121.465	145.465
0.075	42.310	66.310	90.310	114.310	138.310	162.310
0.150	59.156	83.156	107.156	131.156	155.156	179.156
0.225	76.001	100.001	124.001	148.001	172.001	196.001
0.300	92.846	116.846	140.846	164.846	188.846	212.846

(b) BC 2: Fixed-Pinned

Eccentricity of loading (%)	Change in temperature (° C)					
	0	10	20	30	40	50
0.000	25.465	49.465	73.465	97.465	121.465	145.465
0.075	40.744	64.744	88.744	112.744	136.744	160.744
0.150	56.023	80.023	104.023	128.023	152.023	176.023
0.225	71.301	95.301	119.301	143.301	167.301	191.301
0.300	86.580	110.580	134.580	158.580	182.580	206.580

(c) BC 3: Pinned-Pinned

Eccentricity of loading (%)	Change in temperature (° C)					
	0	10	20	30	40	50
0.000	39.789	64.111	88.434	112.757	137.080	161.402
0.075	59.323	83.646	107.969	132.291	156.614	180.937
0.150	78.858	103.181	127.503	151.826	176.149	200.472
0.225	98.393	122.715	147.038	171.361	195.683	220.006
0.300	117.927	142.250	166.573	190.895	215.218	239.541

(d) BC 4: Fixed-Free

American Institute of Steel Construction (AISC) handbook 360 (2016) provides the expression for design strength of compression members as shown in Equation 6.7.

$$F_{cr}(T) = (0.42)^{\sqrt{\frac{F_y(T)}{F_e(T)}}} F_y(T) \tag{6.7}$$

$\boldsymbol{F_y(T)}$ is the yield stress at elevated temperature to be taken from Table A- 4.2.1 from the AISC 360 handbook.

$\boldsymbol{F_e(T)}$ is the critical elastic stress, calculated from Equation 3-4 in the handbook.

However, the code does not consider reduction in the design strength critical elastic stress below a temperature rise of 200° C. This undermines the stresses induced in the members by elevated temperatures.

Therefore, Fischer et al. (2019) have presented equation for critical elastic stress for columns given in Eq. 6.8 below. The value of $F_e(T)$ has been superimposed on the results obtained, which gives an indirect estimate of the temperature at which buckling occurs, for the given material and geometric properties.

$$F_e(T) = \frac{\pi^2 E(T)}{\lambda_{eff}^2} \tag{6.8}$$

$$\text{Slenderness ratio, } \lambda_{eff} = \left(1 - \frac{T}{4000}\right)\lambda - \left(\frac{35}{4000}\right)T \tag{6.9}$$

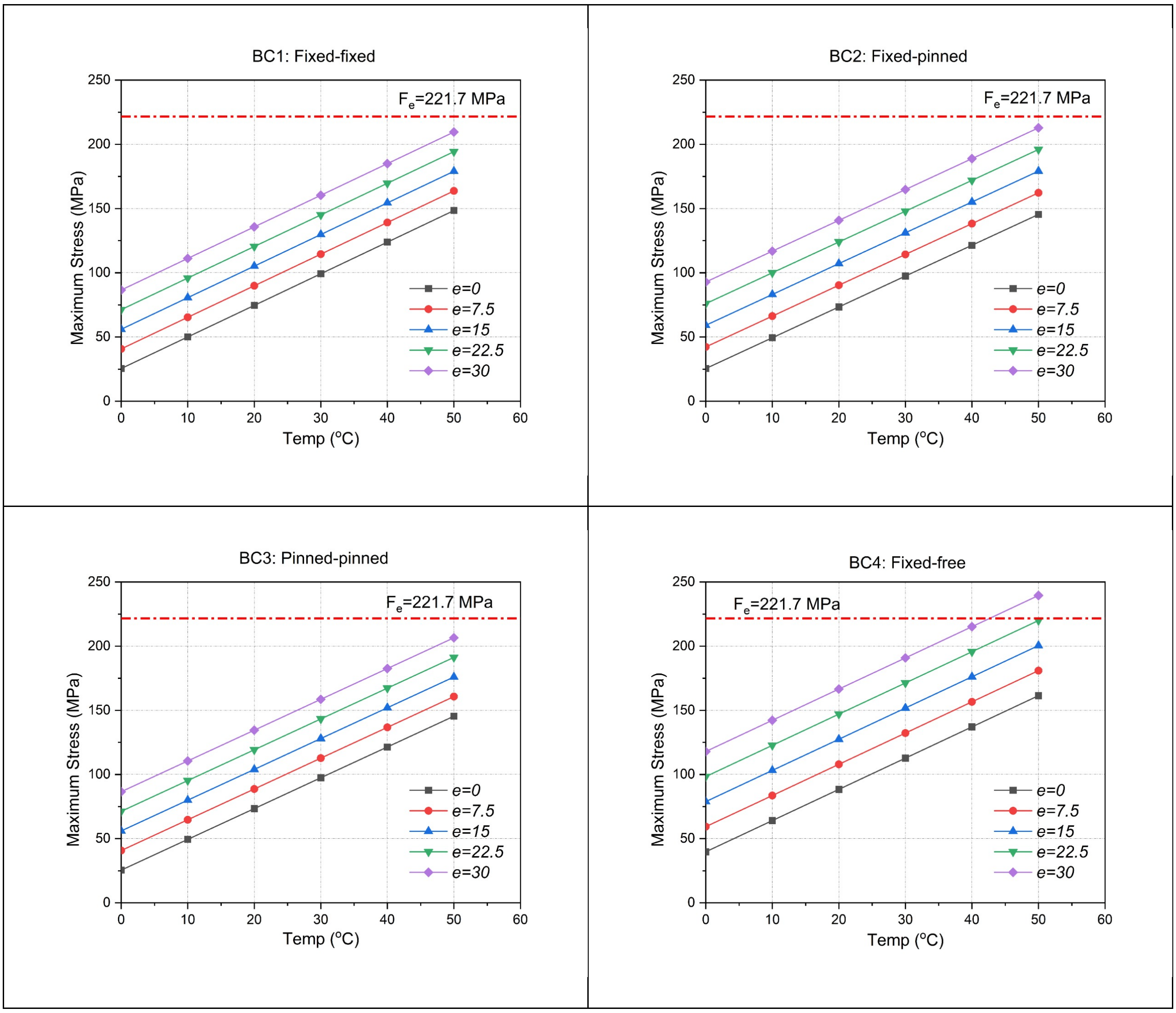


Figure 8: Variation of maximum stress due to mechanical and thermal loading

Thus, the results show that the columns with BC 4 (Fixed - Free) end conditions were susceptible to buckling with a temperature rise of 50° C. However, this method of analysis is indirect, and hence does not evaluate the temperature rise at which buckling occurs. Non-linear analysis is essential to not only determine the buckling temperature, but also in conducting the ***post-buckling analysis*** under ***thermo-mechanical*** loading.

7. Non-linear analysis of columns using Modified Riks approach

Background information of ABAQUS:

ABAQUS is a computer aided engineering (CAE) software that is based on finite element analysis principles. It helps in both modelling and analysis of mechanical elements and in visualization of finite element analysis result. ABAQUS finds use in various industries like aerospace, automotive and complex structure engineering.

After modelling is completed and the model is ready to be analysed, we will proceed to analyse the model by subjecting it to an analysis history. An analysis history is defined as:

- Dividing the problem history into steps.
- Specifying a procedure for analysis of each step;
- Prescribing loads, boundary conditions and output requests for each step.

The most elementary concept in ABAQUS is the division of the problem into steps. A step can be as simple as a static analysis of a load change from one magnitude to other, can also take the form of a creep hold, thermal transient or a dynamic transient.

Analysis steps in which the non-linearities (geometric, material, boundary) present in the model can be included is called a general analysis step. The starting condition of each step in general analysis is taken as the ending of the previous step, i.e. the model continuously evolves during the analysis. The analysis step in which the response can only be linear is called ***Linear Perturbation Analysis*** step. The response in linear perturbation analysis is taken about the base state. ***Eigenvalue buckling analysis*** is an example of a linear perturbation process.

Eigenvalue Buckling Analysis:

Eigenvalue buckling analysis is a linear perturbation procedure used to estimate the critical load of stiff structures. Euler beam under compression is an example of a stiff structure where it doesn't show much geometric deformation until the critical load is reached, after which it buckles and shows large deflection and lowered stiffness.

ABAQUS allows user to use 2 types of eigenvalue extractors. ***Lanczos*** and ***subspace iteration***, out of which the Lanczos method is faster when more eigenvalues are required. Subspace iteration is followed by default.

While analysing, an incremental loading pattern Q^N is defined in the eigenvalue buckling step, This load is scaled by the load multipliers λ_i, and the eigenvalue problem is formulated:

$$(K_0^{NM} + \lambda_i K_\Delta^{NM})v_i^M = 0 \tag{7.1}$$

Where,

$\boldsymbol{K_0^{NM}}$ is the stiffness matrix corresponding to the base state, including preloads
$\boldsymbol{K_\Delta^{NM}}$ is the differential initial stress and load stiffness matrix due to the incremental loading pattern, $\mathbf{Q^N}$
$\boldsymbol{\lambda_i}$ are the eigenvalues,
and $\mathbf{v_i^M}$ are the buckling mode shapes (eigenvectors)

Internally during eigenvalue prediction, ABAQUS first does a static perturbation analysis to determine the incremental stress $\Delta\sigma$, caused by Q^N. The stiffness matrix K_{Δ}^{NM} corresponding to $\Delta\sigma$ and Q^N is formed. During eigenvalue extraction, the stiffness matrix K_0^{NM} corresponding to the base state is formed and the eigenvalue problem is solved iteratively. Input file template for the eigen value problem has been included in the Appendix.

If the live load produces significant geometric change, a nonlinear collapse (Riks) analysis must be performed.

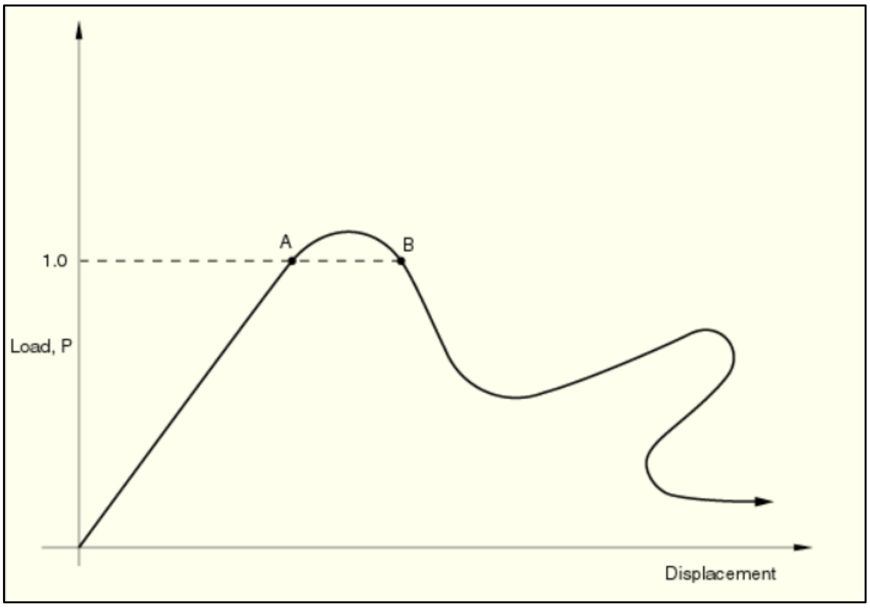


Figure 9: Non-linear (unstable) load-displacement response due to geometric non-linearity captured using ABAQUS

Modified Riks Analysis:

Linear Eigenvalue Analysis is sufficient for most structural engineering design evaluation. But in cases where there is a concern of non-linearities (material or geometric) prior to buckling or an expected unstable response after buckling, a load deflection (Riks) analysis must be performed. Riks method is also useful for solving ill-conditioned problems such as limit load problems or almost unstable problems that exhibit softening.

The Riks method works by assigning load magnitude as an additional unknown and solves simultaneously for displacements and loads. As the load value is an unknown, a quantity called "arc length" is used to quantify the progress of the solution. Riks analysis will return a solution regardless of the stability of the response (Figure 5). A load whose magnitude is defined in the Riks step is referred to as *Reference Load,* and all prescribed loads are ramped up from the initial value (before the Riks analysis step) to this reference value.

The current load P_{Total} is given by:

$$P_{Total} = P_0 + \lambda(P_{ref} - P_0) \tag{7.2}$$

Where, P_0 is the dead load, P_{ref} is the reference load and λ is the load proportionality factor.

ABAQUS uses Newton's Method to solve system of nonlinear equilibrium equations. The Riks procedure uses an extrapolation of 1% of the strain increment. The user provides the initial increment in arc length along the static equilibrium path ($\Delta\lambda_{in}$) and the initial load proportionality factor is calculated as:

$$\Delta\lambda_{in} = \frac{\Delta l_{in}}{lp} \tag{7.3}$$

Where l_p is the total arc length (set to 1 conventionally).

This value of $\Delta\lambda_{in}$ is used in the first iteration of analysis. Subsequent values of load proportionality factor are calculated automatically, implying the user has no control over the load magnitude. The values of λ are a part of the solution.

End the Riks analysis: To end a *Riks analysis step* we can specify a maximum value of λ, or a maximum displacement value at a specified degree of freedom. The Riks analysis will stop once the specified value is crossed. If neither of the conditions is crossed, the analysis will continue for the number of increments specified in the step definition.

Any temperature differences applied between nodes in thermal boundary conditions will cause a thermal expansion and strains if the thermal coefficient is defined for the material. The loads produced by these thermal strains are added to the reference load specified for analysis and is ramped up with the load proportionality factor. Thus, the Riks procedure can analyse *post-buckling* and *collapse due to thermal straining*.

Input file template for the Riks analysis has been included in the Appendix.

8. Discussion and scope for future work

A three-part study was conducted to study the behaviour of columns under elevated temperatures, and with emphasis on determining their stability. The following observations were made based on the study:

(i) Eigen value problem to determine the buckling load through FE formulation showed convergence with Euler's formula with 2- or 4-element discretization.
However, the objective of determining the critical temperature rise was not fulfilled as the buckling load is independent of thermal stresses.

(ii) Stresses were evaluated under thermo-mechanical loading for various end conditions of the column. The analysis involved evaluation of stresses at elevated temperatures.

However, the indirect method of comparing the stresses thus induced with the available literature is not a convincing method of evaluating the critical temperature rise for buckling.

(iii) In Part III, the procedure for non-linear analysis has been described which is known as the Riks method. This not only provides the value of critical thermo-mechanical stresses, but also facilitates the performance of post-buckling analysis until collapse.

Therefore, based on the observations mentioned above, the scope for future work has been identified as follows to further the study in structural fire engineering:

Short-term objectives:

(a) Implementation of Riks method for buckling problem in Part II.
(b) Extending the work to study real-life structures.

Long-term objective:

- Comprehensive study on inelastic buckling of columns for performance-based fire resistant design of steel structures.

A1. <u>References</u>

1. ABACUS Version 6.6 User Documentation.

2. Eagan, N. & Hauser, G. Newton's Method on System of Non Linear Equations. Carnegie Mellon University.

3. Modified Riks Method Section 2.3.2, ABAQUS Version 6.6 User Documentation

4. Logan, D. L. (2014, January 1). A First Course in the Finite Element Method, Textbook by Daryl L. Logan. CL Engineering.

5. Cook, R. D., Malkus, D. S., Plesha, M. E., & Witt, R. J. (2001, October 15). Concepts and Applications of Finite Element Analysis. John Wiley & Sons. https://doi.org/10.1604/9780471356059

6. Fischer, E.C., Agarwal, A., Varma, A.H. (2019). "Performance-Based Structural Fire Engineering of Steel Building Structures: Design-Basis Compartment Fires". ASCE Journal of Structural Engineering. Vol. 145, Issue 9.

7. AISC "Steel Construction Manual," 13th Edition, American Institute of Steel Construction, Chicago, 2005.

8. Beer , F. B., Johnston, E. J., DeWolf, J. D., & Mazurek, D. M. (2014, February 16). Mechanics of Materials. McGraw-Hill Education.

A2: ABAQUS input file templates for analyses

```
*HEADING
…
*BOUNDARY
Data lines to specify zero-valued boundary conditions contributing to the base state
**
*STEP, NLGEOM
The load stiffness terms will be included in the eigenvalue buckling steps
since the NLGEOM parameter is used in this (optional) preload step
*STATIC
Data line to control incrementation
*BOUNDARY
Data lines to specify nonzero boundary conditions (dead loads)
*CLOAD and/or *DLOAD and/or *TEMPERATURE
Data lines to specify dead loads, $P^N$
*END STEP
**
*STEP
*BUCKLE
Data line to request the desired number of symmetric modes
*CLOAD and/or *DLOAD and/or *TEMPERATURE
Data lines to specify perturbation loading, $Q^N$
*END STEP
**
*STEP
*BUCKLE
Data line to request the desired number of antisymmetric modes
*CLOAD and/or *DLOAD and/or *TEMPERATURE
Data lines to specify perturbation loading, $Q^N$
*BOUNDARY, LOAD CASE=1
Data lines to specify all boundary conditions for perturbation loading
*BOUNDARY, LOAD CASE=2, OP=NEW
Data lines to specify all antisymmetric boundary conditions for eigenvalue extraction
*END STEP
```

Figure A2.1: Input File Template for Eigen value buckling analysis

Input file template

```
*HEADING
…
*INITIAL CONDITIONS
Data lines to define initial conditions
*BOUNDARY
Data lines to specify zero-valued boundary conditions
**
*STEP, NLGEOM
*STATIC
*CLOAD and/or *DLOAD and/or *TEMPERATURE
Data lines to specify preload (dead load), $P_0$
*END STEP
**
*STEP, NLGEOM
*STATIC, RIKS
Data line to define incrementation and stopping criteria
*CLOAD and/or *DLOAD and/or *TEMPERATURE
Data lines to specify reference loading, $P_{ref}$
*END STEP
```

Figure A2.2: Input File Template for Riks Analysis