IJCRT.ORG ISSN : 2320-2882
INTERNATIONAL JOURNAL OF CREATIVE RESEARCH THOUGHTS (IJCRT)
An International Open Access, Peer-reviewed, Refereed Journal

Modified Formulae For Major Buckling Length Of Steel Rigid Frame With I-Beams Having Circular Duplicated Web Openings (Cellular Beams).

[1] W.M. Sayed, [2] Ali Hammad, [3] Sherif K. Hassan
[1] Research Assistant, [2] Assistant Professor, [3] Professor
Department of Structural Engineering,
[1] Ain Shams University, Cairo, Egypt

Abstract: Most design codes for steel members and frames include specifications for the design of compression columns or beam columns, which involve the effective length factor, K. This factor simplifies the design of framed members by transforming an end-restrained compressive member into an equivalent pinned-ended member. Traditionally, obtaining the effective length factor involves solving complex equations or using alignment charts for different frame cases. However, a new approach using simple equations directly determines the effective length factor based on the rotational resistance at column ends (GA, GB) and how to take the shear deformation of the cellular members by getting an equivalent major elastic second moment of area. The goal is to provide more accurate results than existing steel construction codes. The paper also includes comparisons between the results obtained from the proposed equations and those obtained from exact solutions.

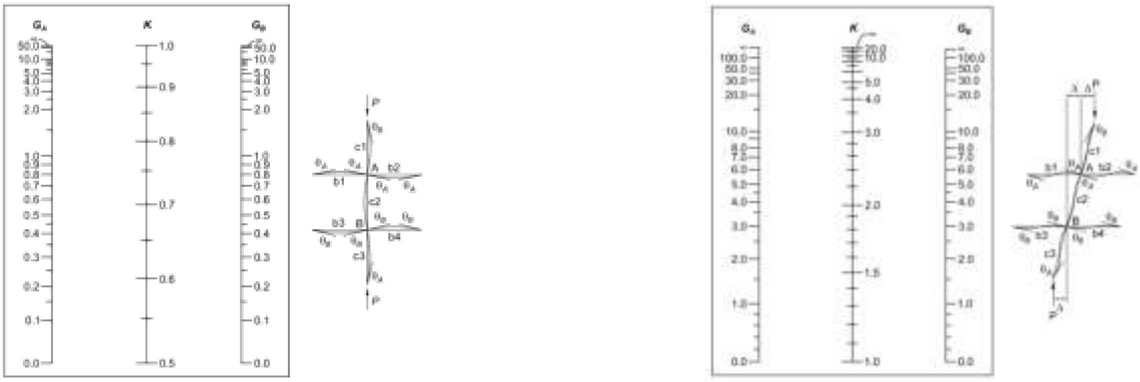
Index Terms: New formula, Cellular, Perforated, Buckling, Stability, Steel.

I. Introduction

This study focuses on rigid steel frames, so all elements are beam-column elements. They will resist bending moments and axial forces. Hence, we need to know the effective lengths in and out of plane. The distance between bracing points or strut points can determine the out-of-plane effective length. The alignment charts [1] can determine the buckling in the plane.

The alignment charts [1]have two types: prevented and permitted to sway.

Moreover, both have elastic rotational resistance at both ends of the element GA and GB values to get the K value and multiply it by the actual length of the elements to get the in-plane buckling.



(a) (b)

Figure 1: Alignment charts[1] **a) sidesway prevented b) sidesway permitted.**

II. THE IN-PLANE BUCKLING FACTOR "K".

The specimen is a rigid steel frame with hinged support at the column ends and a rigid connection between the beam and column. The frame is laterally braced out-of-plane and moves freely in-plane. Figure 2 shows our point's boundary condition; the plane (z-x) is defined as in-plane, and the axis-y is the out-of-plane.

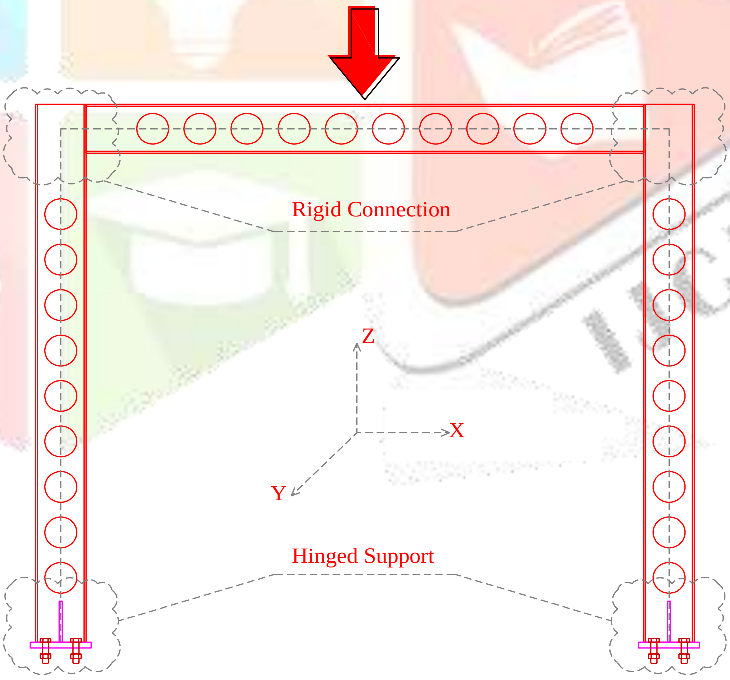


Figure 2: Specimen schematic arrangement.

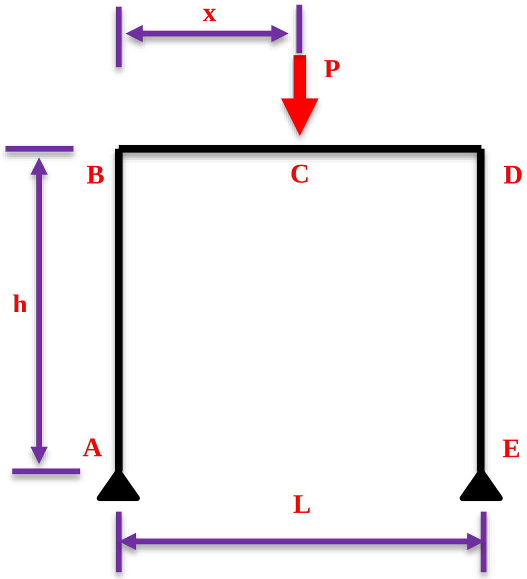


Figure 3: Free body diagram for the frame.

Figures 2 and 3 show the arrangement of the specimen and the free body diagram for it for points A and E, defined as hinged supports, and points B and D, defined as rigid connections, and the load acting on point C midpoint of the member B-D.

Point A is defined as hinged, which means it is free to rotate, so the GA value equals infinity.

Point B is defined as a rigid connection between the beam and column, so the value of GB is expressed as: in Eq. 1 [2].

$$G_B = \frac{\sum\left(\frac{E\,I}{L}\right)_{column}}{\sum\left(\frac{E\,I}{L}\right)_{beam}} = \frac{\text{sum of column stiffnesses meeting at the joint}}{\text{sum of beam stiffnesses meeting at the joint}} \quad \mathbf{1}$$

To solve Eq 1, we need the following:

- Material modulus of elasticity (E).
- The second moment of area (I).
- The length (L) of every member in the frame.
- The assumption used for the equation.

The first three points are given data from the studied case.

For point 4. The assumptions [2] used for the model are:

1. All members are prismatic and behave elastically.
2. Beam-to-column connections behave linearly with identical stiffness parameters on each floor.
3. The axial force in beam members is negligible.
4. All columns in the frame buckle simultaneously.
5. At a joint, the restraining moment provided by the beam is distributed among the columns in proportion to their stiffnesses.
6. At buckling, the rotations at the near and far ends of the beam are equal and in the same direction (i.e., the beams are bent in double curvature).
7. The applied column stress may be well below the yield stress; the presence of residual stresses causes yielding on portions on the cross-section, thereby reducing the effective material stiffness of the cross-section to a value referred to as the tangent stiffness, Et< E.

   Tangent stiffness can be obtained from the stiffness reduction factor τ by dividing Et by E.

We have two main issues before we can get the value of GB:

The first one is that the accuracy of the alignment charts depends on their size and the reader's sharpness of vision. Using an alignment chart throughout calculations in a spreadsheet may delay full automation and create errors.

will be illustrated in part 3.

The second is that, in our case, all elements are web-perforated, which means they have different elastic properties throughout the length of the member.

will be illustrated in part 4.

III. THEORETICAL FORMULA.

A rigid frame depends upon frame action to withstand lateral forces, but its sideways are not prevented. In this case, the K-factor

is never smaller than 1.0. The mathematical equation for the

"Sway permitted" case is [2]:

$$\frac{G_A \; G_B \left[\frac{\pi}{K}\right]^2 - 36}{6(G_A+G_B)} = \frac{\frac{\pi}{K}}{\tan\left[\frac{\pi}{K}\right]} \qquad 2$$

To solve this equation, we must use an advanced tool like Excel VBA to get the Ktrue value.

To validate this equation, we will compare the results with the KFEA.

By conducting finite element model (ANSYS) for a rigid steel frame with different parameters, as shown in Figure 4.

The parameter is used to get the two values Ktrue & KFEA for the column.

- Beam profile and its length.
- Column profile and its length.
- Assumed all elements are solid webbed ex IPE, HEA, ……etc.

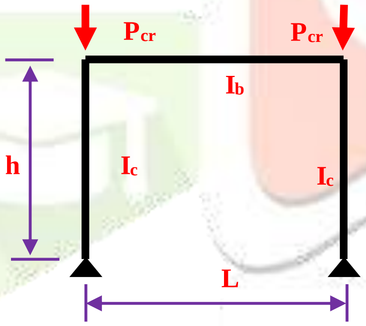


Figure 4: FBD shows the studied parameter to obtain the critical buckling load.

The finite element model is conducted using beam elements, and the boundary condition is that the bases are hinged, and the steel

frame is braced out-of-plane. We assign a unit load at the ends of columns (at joints between beams and columns), as shown in

Figure 4, run a linear buckling analysis, and obtain the critical buckling load. Then, we get the KFEA value by Eq 3 (Euler

Bernoulli), as it is the only unknown parameter.

$$P_{cr} = \frac{\pi^2 E I_c}{[K_{FEA} L]^2} \qquad 3$$

$$K_{FEA} = \sqrt{\frac{\pi^2 E I_c}{P_{cr} L^2}} \qquad 4$$

One hundred forty models were created, and the results were compared. We found that the difference is minor, and we can predict an equation for the buckling in-plane for the column in a hinged steel rigid frame.

For example:

As shown in Figure 5, the steel rigid frame consists of two columns and one beam. The profile for all elements is IPE 600, the distance between the center line of the column is 8 m, and the vertical distance between the base and the center line of the beam is 12 m.

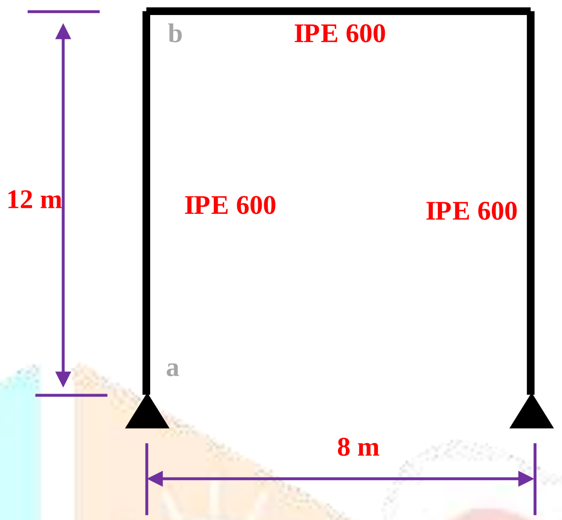


Figure 5: FBD show the frame.

For column a-b Joint a is hinged so GA = ∞ (not applicable) Assume that GA = 109 Joint b is rigid connection, by using Eq 1 we found that,

$$G_B = \frac{L_b}{L_c} = \frac{8}{12} = 0.6667$$

By using Eq 2 we found that,

$$\frac{10^9 x\, 0.6667 \left[\frac{\pi}{K_{true}}\right]^2 - 36}{6\left(10^9 + 0.6667\right)} = \frac{\frac{\pi}{K}}{\tan\left[\frac{\pi}{K_{true}}\right]}$$

By using excel VBA to get the K_{true} Assume

$$LHS = \frac{10^9 x\, 0.6667 \left[\frac{\pi}{K_{true}}\right]^2 - 36}{6\left(10^9 + 0.6667\right)}$$

$$RHS = \frac{\frac{\pi}{K}}{\tan\left[\frac{\pi}{K_{true}}\right]}$$

Assuming Ktrue = 0.0001 and get values of LHS & RHS, Then check on Eq 5

$$\Delta = LHS - RHS \quad (5)$$

If Eq 5 equals zero, that means Ktrue = 0.0001. If not, we will add 0.0001 and try again until the equation equals zero. This try-and-error circulation will be run by the VBA tool in Excel. After that, we will conduct linear buckling analysis using ANSYS software for the same frame as shown in Figure 5 and the same concept as shown in Figure 4. The beam element was used to create the frame to shorten the run time and get the buckling critical load, and Eq 4 can be used to get the KFEA.Then, we found a good agreement between the two values of the K buckling factor in each model as shown in Figure 6.

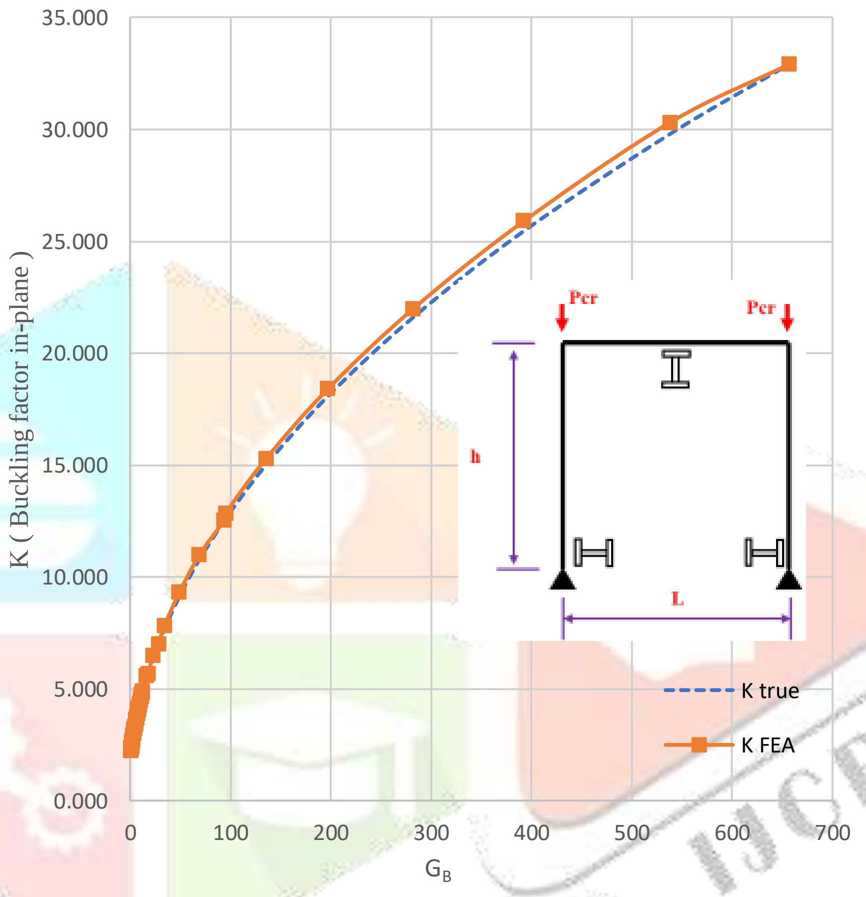


Figure 6: Comparison between K_{true} & K_{FEA}.

We can use a formula extracted from the curve to calculate the buckling in-plan for the column of the two hinged rigid frame.
We analyzed the data by using Python code.

$$K = 1.245 \times \sqrt{G_B} + 0.846 \quad (6)$$

Using Eq 6, the first problem will be solved.

IV. ANALYTICAL AND FEM METHODS.

The main concept for getting the elastic major second moment of area for any element is to get the linear buckling load for this element by creating a model that is moving freely about its major axis and is braced about its minor axis, as shown in Figure 7, the concept model to get the value of the elastic major second moment of area.

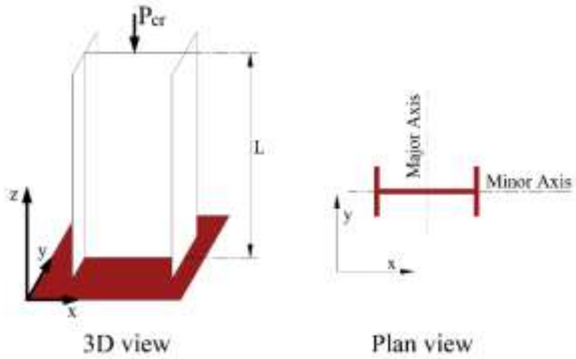


Figure 7 : FBD for a fixed-free column about major axis.

$$P_{cr} = \frac{\pi^2 E I_{FEM}}{[K L]^2} \quad \textbf{7}$$

Where;

E	= Modulus of elasticity,	Mpa
I_{FEM}	= Required elastic major second moment of area,	mm^4
L	= Actual length of the member,	mm
K	= 2 “buckling factor”	
P_{cr}	= The linear buckling load,	N

By using the Eq 1.7 the only unknown is the I_{FEM}

$$I_{FEM} = \frac{P_{cr} (K L)^2}{\pi^2 E} \quad \textbf{8}$$

By using Eq 8 to get directly I_{FEM}

V. MODEL DESCRIPTION.

The typical studied configuration consists of a fixed free column as shown in Figure 8.

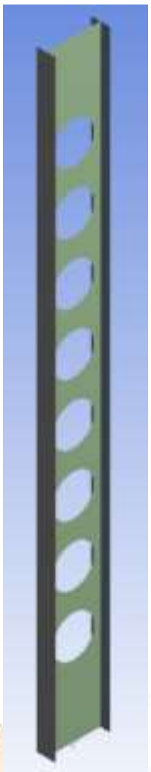


Figure 8: Overview for the 3-D model.

VI. SHELL ELEMENTS.

The model consists of a column with a profile steel I-beam perforated in the web, and its boundary conditions are fixed-free. This column is modelled using SHELL 181, which is used for 3-D modelling of thick shell structures. A four-node element can be defined with six degrees of freedom at each node, enabling translations and rotations about the cartesian axes. If the membrane option is selected, the element will only have translational degrees of freedom. The degenerate triangular option should exclusively be employed as an auxiliary component in mesh production. SHELL181 is highly suitable for linear, big rotation, and large strain nonlinear applications. Changes in shell thickness are accounted for in nonlinear analyses. In the element domain, both full and reduced integration schemes are supported. SHELL181 accounts for the effects of distributed pressures on the follower (load stiffness). SHELL181 can be used for layered applications to model composite shells or sandwich construction. The first-order shear-deformation theory, also referred to as the Mindlin-Reissner shell theory, governs the accuracy of modelling composite shells. The formulation of the element was obtained from logarithmic strain and real stress measurements. The kinematics of the element permit the estimation of finite strains in the membrane, which means stretching. However, it is anticipated that the changes in curvature over a specific amount of time will be low.

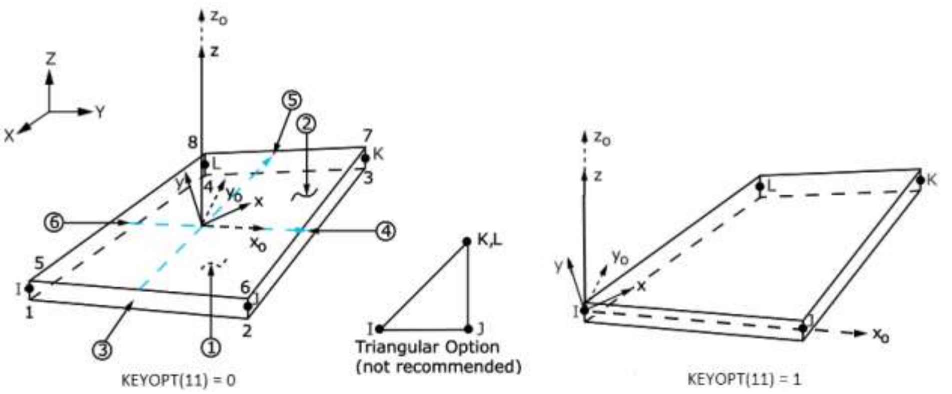
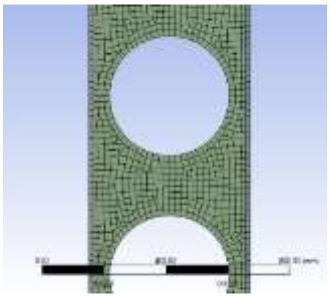


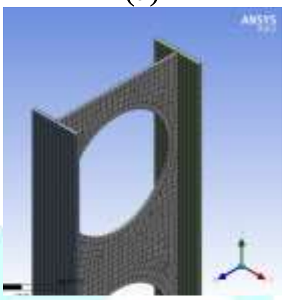
Figure 9: Shell element SHELL181 in ANSYS.

VII. MESHING OF FINITE ELEMENT MODEL.

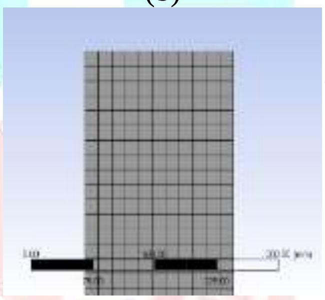
Meshing is one of the most important processes in modeling since the accuracy of the results largely depends on it. A fine mesh can be used in areas of high stress and deformation. In the remaining areas, a coarser mesh can be used. Different sizes were tried, and the best size for these elements was chosen to get close results and avoid errors. Element size is used as a ratio of its element's width; for the flange, it is taken as 0.1 from the flange width (b_f); for the web, it is taken as 0.04 from beam depth (d_g-t_f) measured from the centers of the flanges, as shown in Figure 11, around the openings, the shells must be around the opening to get accurate results.



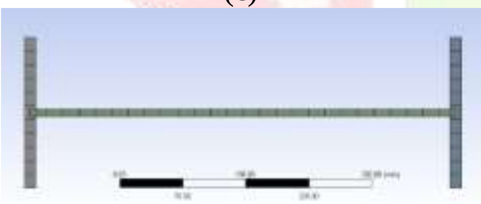
(a)



(b)



(c)



(d)

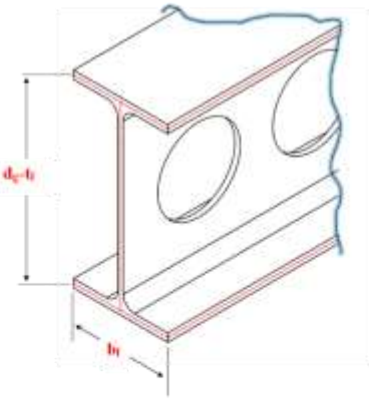


Figure 10: Dimension of shell element of I-sec.

Figure 11: Finite-element meshing for structural analysis.
a) Around the opening. b) 3D-view. c)Flange meshing. d)Plan of the I-profile.

VIII. BOUNDARY CONDITIONS.

The boundary conditions of the column are represented as true fixed support that is prevented from torsion and not free to wrap. The boundary conditions are simulated in the finite element model using a remote point with deformable behavior in which the geometry is free to deform. Remote displacement is assigned to the remote point with fixed configurations. At the start of the column, the sixth degree of freedom is equal to zero. At the end of the column, displacement in the x-direction and rotation about the axis are prevented. The overall column is prevented from deforming in the x-direction. The loading condition is a unit-concentrated load at the remote point at the end of the column.

IX. MATERIAL PROPERTIES.

For the linear buckling analysis run a unit moment is used. For all specimens, the material is linear elastic. The used material is a steel structure with a modulus of elasticity (E=200 GPa), yield stress (Fy=250 MPa) and Poisson's ratio (υ=0.3).

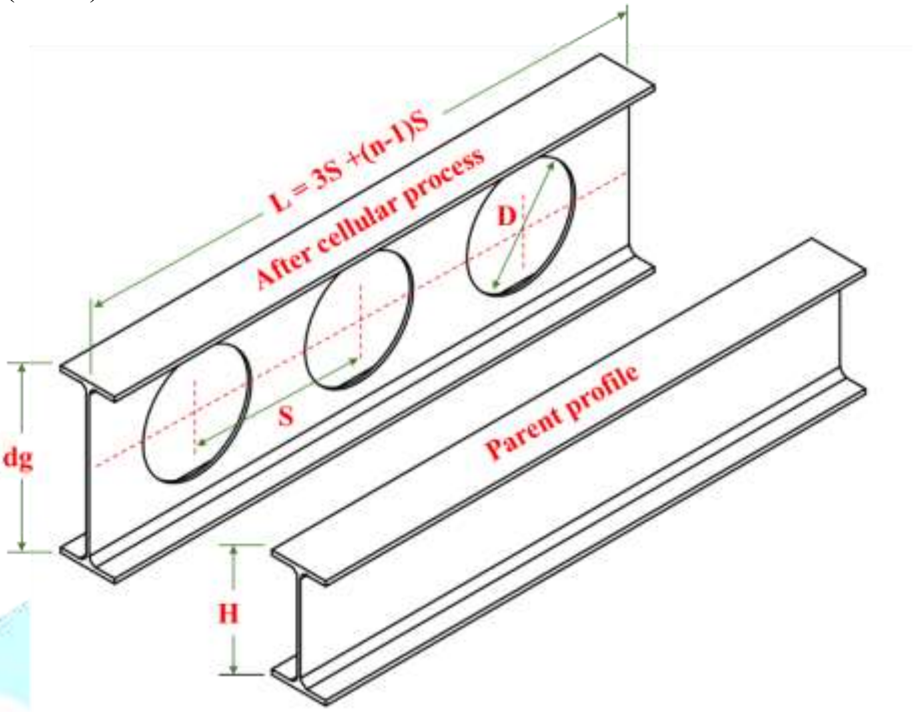


Figure 13: Studied parameters.

Where;

L	= Beam length after cellular process,	mm
D	= Diameter of the opening,	mm
S	= Spacing between openings,	mm
d_g	= Beam depth after cellular process,	mm
H	= Beam depth before cellular process,	mm
n	=Number of openings,	

Parent profile dimensions.

X. EIGEN BUCKLING ANALYSIS.

Eigenvalue Buckling analysis predicts the theoretical buckling strength of an ideal elastic structure. This method corresponds to the textbook approach of elastic buckling analysis. Eigenvalue buckling analysis of a column will match equation 7 and we can use the equation 8 directly to get the I_{FEM}.

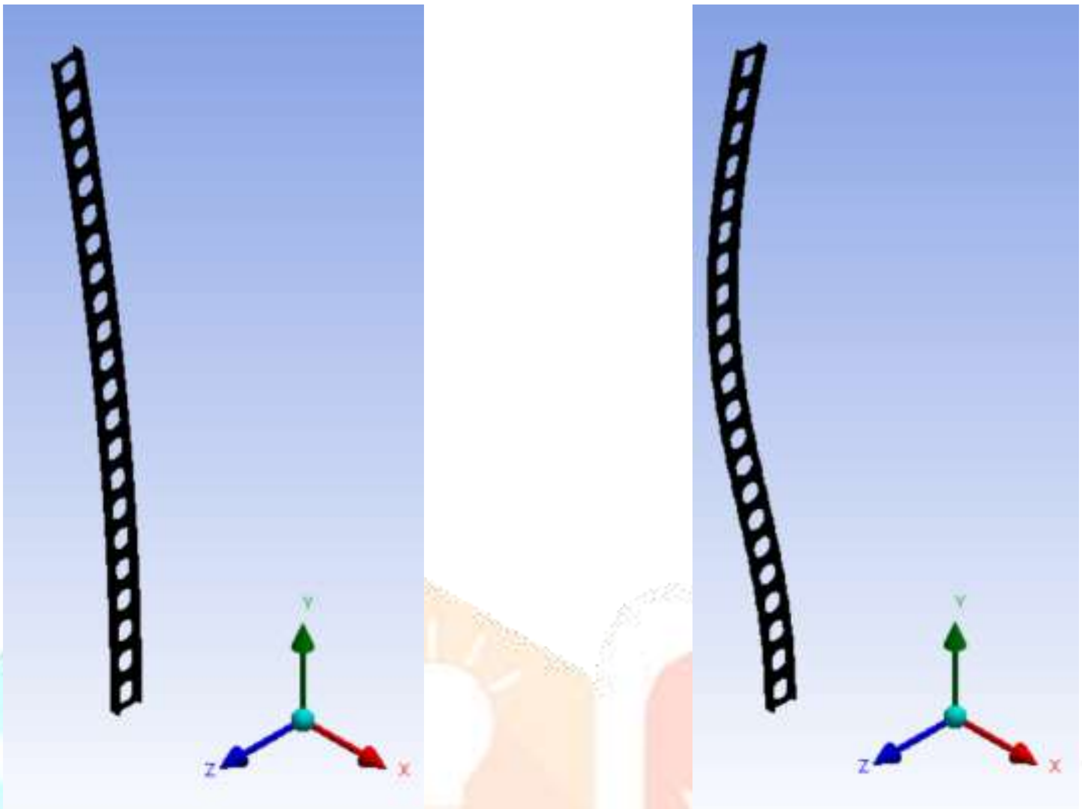
XI. MODEL PROCEDURE.

- Define the steel I-beam component's geometry.
- Define the steel`s elastic material properties.
- Define the steel I-beam boundary conditions.
- Assign a unit load according to the required loading condition.
  Buckling analysis to get the critical buckling failure mode shape

XII. Calculate the Ix using an approximate equation.

After we get the I_{FEM} we will use an approximate equation with sample calculations, that can the designer use easily.

Figures 17 and 18 show the assumption for the two methods.



(a) (b)

Figure 14: The elastic buckling failure modes of the fixed-free column.

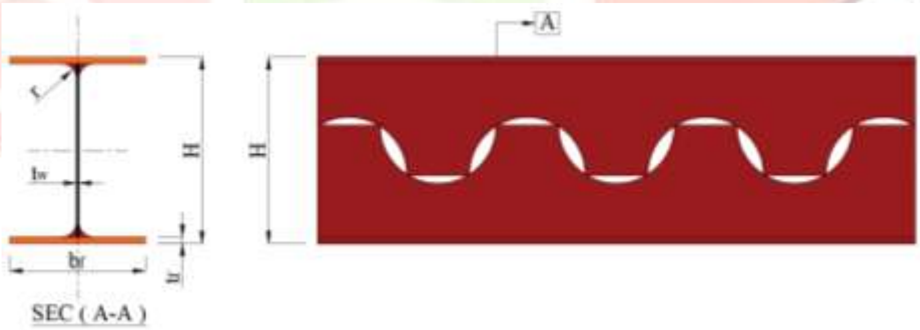


Figure 15: Parent profile before the cellular process.

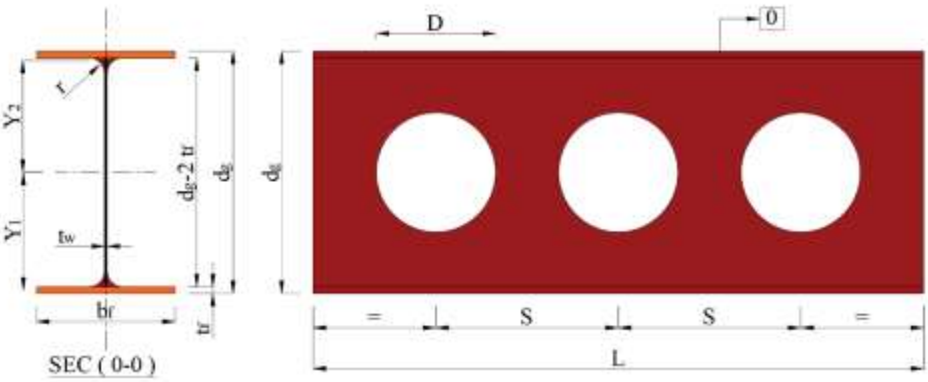


Figure 16: Cellular element dimensions & notations.

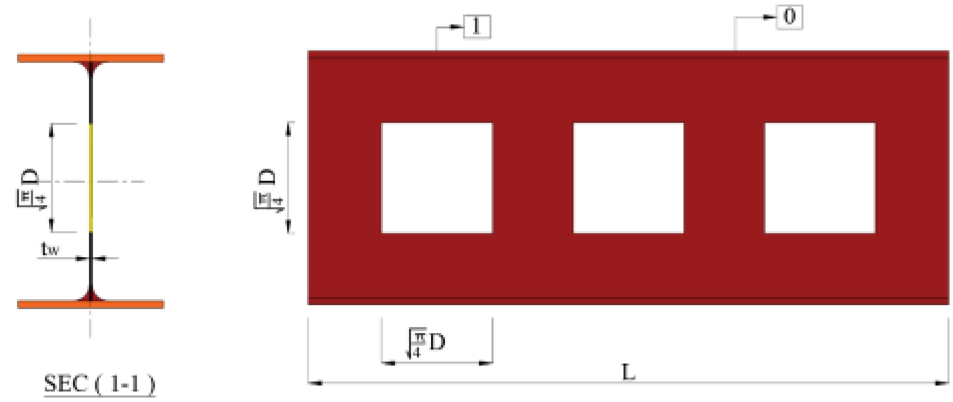


Figure 17: "Method 1" for cellular element.

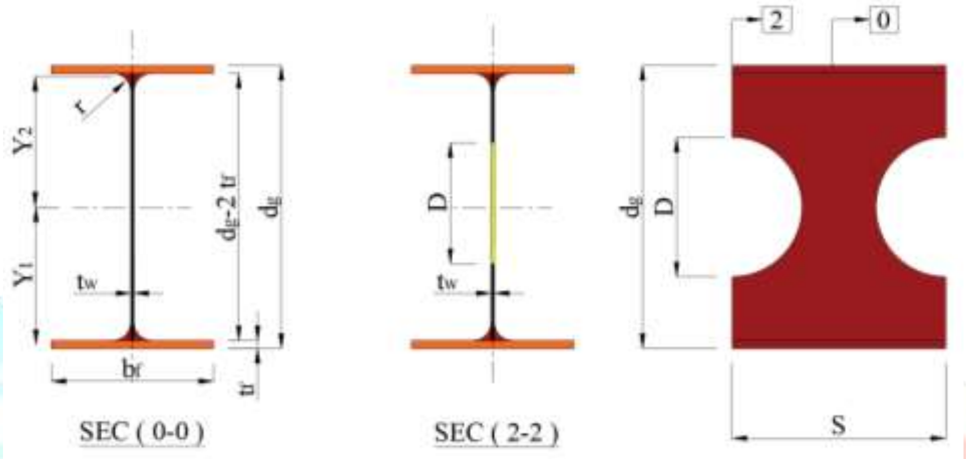


Figure 18: "Method 2" for cellular element.

$$I_g=\frac{(H-2t_f)^3\ t_w}{12}+2\left(\frac{t_f^3\ b_f}{12}\right)+2\left(b_f\ t_f\,(Y_1)^2\right)+4\left(\left(1-\frac{\pi}{4}\right)r^2\ (Y_2)^2\right)+4\left(\frac{9\pi^2-84\pi+176}{144(4-\pi)}\right)r^4 \quad 9$$

$$Y_1=0.5\left[d_g-t_f\right] \quad 10$$

$$Y_2=0.5\left[d_g-2\ t_f\right]-Y_3 \quad 11$$

$$Y_3=\left[\frac{10-\pi\ 3}{12-\pi\ 3}\right]r \quad 12$$

n = no of openings

$$I_{SEC(0\text{-}0)}=\ I_g$$

$$I_{SEC(1\text{-}1)}=\ I_1\ =\ -\ \frac{\left[\sqrt{\frac{\pi}{4}}\ D\right]^3\ t_w}{12}$$

$$I_{SEC(2\text{-}2)}=\ I_2=-\ \frac{[\,D\,]^3\ t_w}{12}$$

$$I_{method.1}=\frac{I_g\ \ L+I_1\ \ n\ \ \sqrt{\frac{\pi}{4}}D}{L} \quad 13$$

$$I_{method.2}=\frac{I_g\ (S-D)+I_2\ D+I_g\ D}{S} \quad 14$$

To decide which method is more accurate for I_{FEM}, we construct a 100-FE model divided into four groups. Each group has the same parameters.

Each group has a fixed ratio of d_g/H, S/D, d_g/D, and the parent profile.

To construct a curve, we add one more opening to the web than the previous sample. Hence, the length of the element gets bigger because the number of openings directly affects it, as shown in Table 3.1, the sample group parameters.

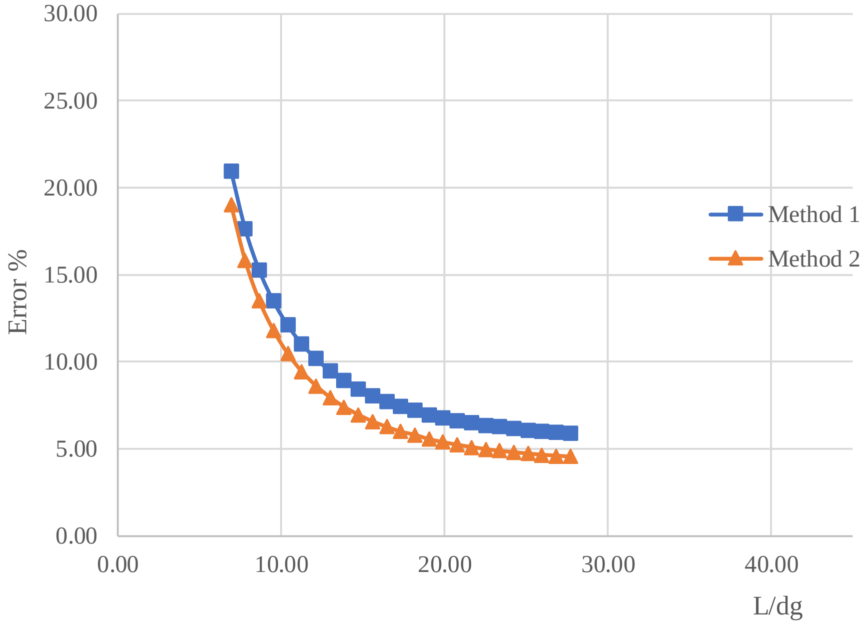
For each group, there is a graph; the X-axis is for L/d_g, and the Y-axis is for the error %.

Where:

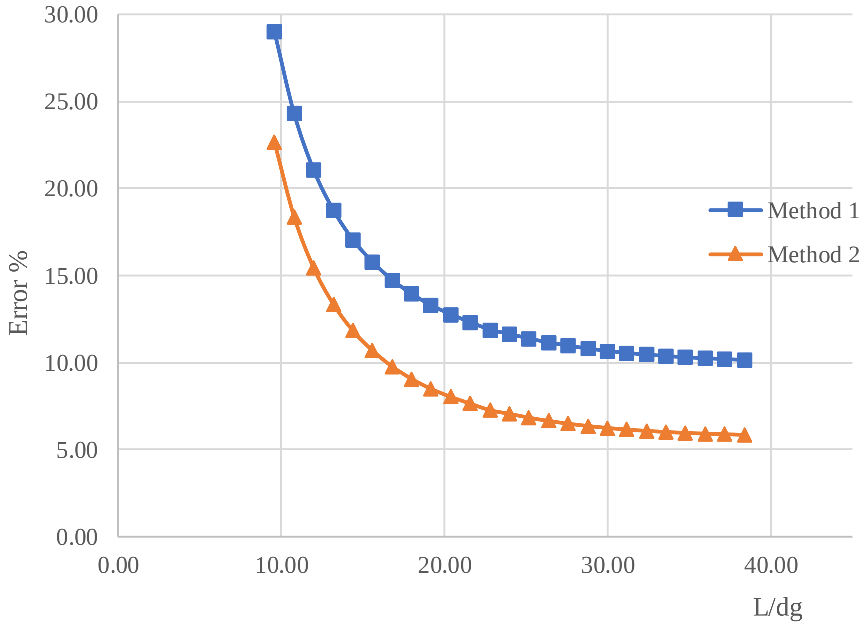
$$\text{error \%} = \frac{I_{Method} - I_{FEM}}{I_{FEM}} \times 100$$

Groups	Parent profile	d_g/H	S/D	d_g/D
1	IPE 200	1.3	1.5	1.733
2	IPE 200	1.5	1.5	1.25
3	IPE 300	1.3	1.5	1.741
4	IPE 300	1.5	1.5	1.25

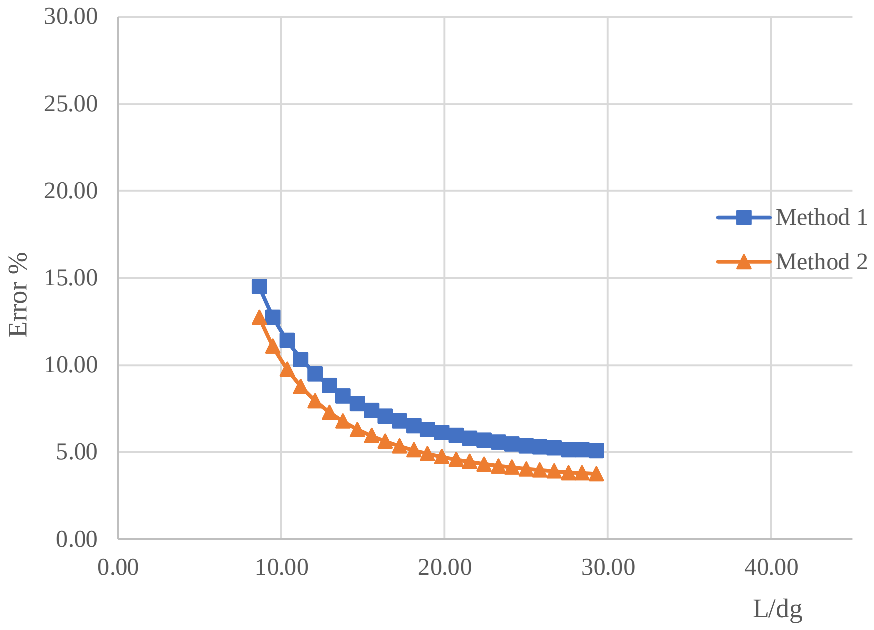
Table 1: Table shows the groups parameters



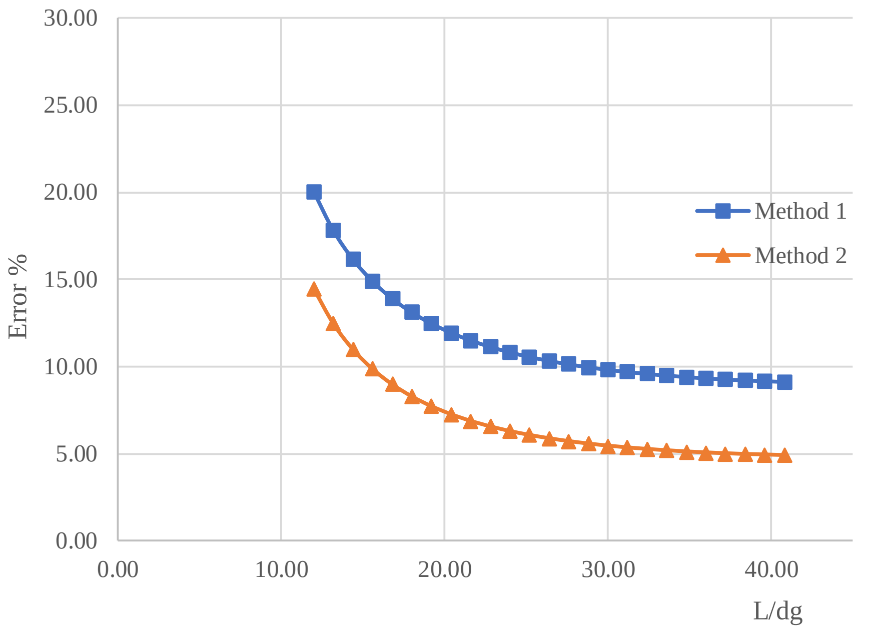
(1)



(2)



(3)



(4)

Figure 19: Relation between the length of the element and Error to the FE model. 1) For group 1. 2) For group 2. 3) For group 3. 4) For group 4.

Figure 19 shows that Method 2 (Eq. 14) is more accurate and gives results more related to the FE model.

So, we can use Eq. 14 to get the elastic major second moment of area for any cellular element. But we need to take the effect of the shear deformation in that equation. So, we will study more groups and the output of this results a graph; the X-axis is for L/d_g, and the Y-axis is for α.

Where:

$$\alpha = \frac{I_{FEM}}{I_{Method\ 2}}$$

To get the true realistic elastic major second moment of area for any cellular element, only the designer can calculate it from Eq. 1.15 by getting the $I_{Method\ 2}$ from Eq 1.14 and multiplying it with α; hence, the shear deformation is taken.

$$I_{TRUE} = \alpha \ I_{Method\ 2} \quad (15)$$

To get the α factor, too many parameters will be studied, as illustrated in Figure 13, Where as follows:

(1) The parent profile dimensions.
(2) The total length of the member.
(3) $R = \frac{d_g}{H}$
(4) $Q = \frac{d_g}{D}$
(5) $Z = \frac{S}{D}$

<u>For parameter No.01</u>, we will extensively study the IPE profile.
[IPE 100, 200, 300, 400, 500 and 600].
<u>For parameter No.02</u>, if the length is too large, the shear deformation has a minor effect, and vice versa, so we will examine the changes using the number of openings.
<u>For parameter No.03</u>, the ratio between the total depth after the cellular process and the original total depth of the parent profile, where the recommended range is 1.3~1.5 [4].
<u>For parameter No.04</u>, the ratio between the total depth after the cellular process and the diameter of the opening, where the recommended range is 1.25~1.75[4].
<u>For parameter No.05</u>, the ratio between the spacing between openings and the diameter of the opening, where the recommended range is 1.08~1.5[77]. The more the spacing between the openings is large, the more the effect of the shear deformation decreases so that we will fix that parameter into 1.5.

Figure 20 shows the effect of Parameters No.3 & 4, and we can conclude that the effect of R has a minor effect, and we can make an extensive study on the Q.

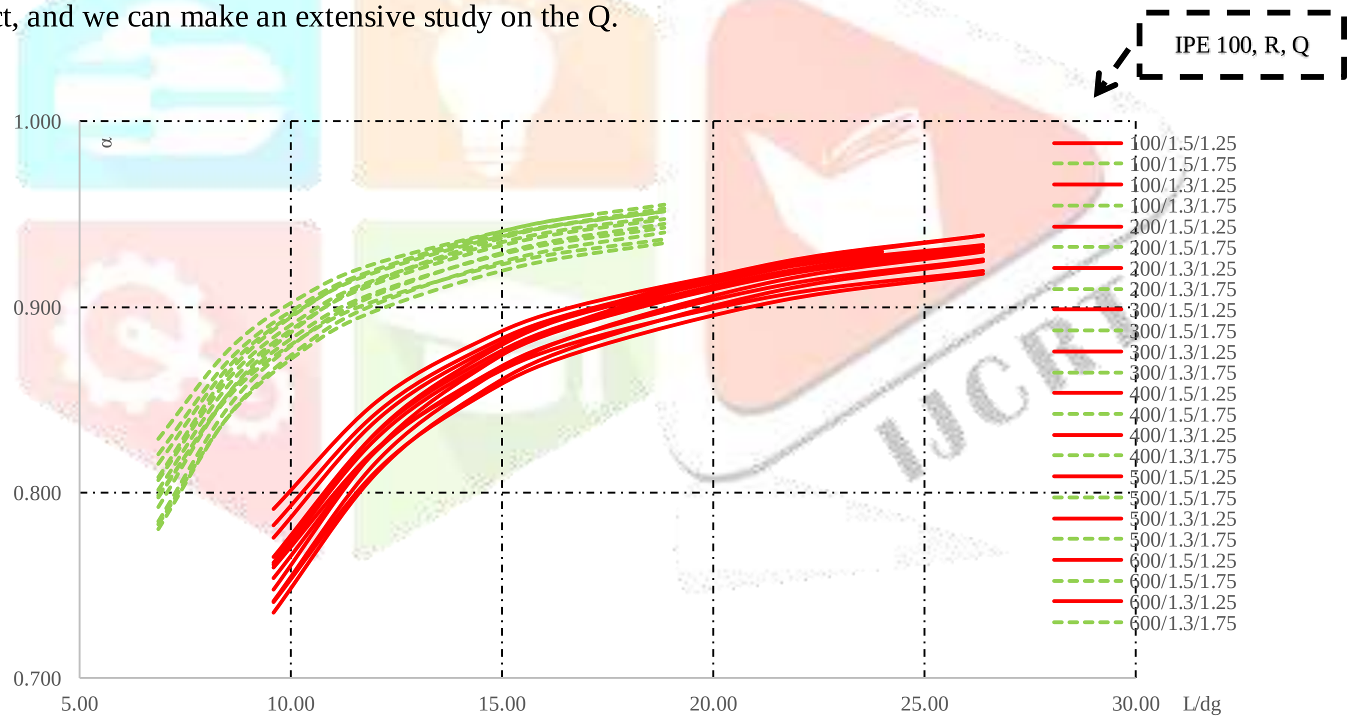


Figure 20: Relation between a factor to the L/dg.

XIII. ACKNOWLEDGMENTS

I'm grateful to God for inspiring me to complete this assignment and to the professors for their invaluable support. Special thanks to Prof. Sherif Kamal Hassan for his guidance and encouragement, and to Dr. Ali Hammad for his insightful advice. Lastly, heartfelt appreciation to my family for their encouragement.

XIV. References

[1] ANSI-AISC 360, “Specification for Structural Steel Buildings,” American National Standard, 2016.

[2] M. Barakat and W.-F. Chen, “Practical Analysis of Semi-Rigid Frames,” AISC Engineering Journal, vol. 27, 1990.

[3] O.A. Bauchau and J.I. Craig, “Structural Analysis With Applications to Aerospace Structures,” SOLID MECHANICS AND ITS APPLICATIONS, vol. 163, 2009.

[4] Sameer S. Fares, J. Coulson, and David W. Dinehart, “Castellated and Cellular Beam Design. Steel Design Guide 31,” American Institute of Steel Construction, 2016.