Practical analysis procedures of steel portal frames having different connections rigidities using modified stiffness matrix and end-fixity factor concept

IOP Conf. Series: Materials Science and Engineering **518** (2019) 022037 doi:10.1088/1757-899X/518/2/022037

Practical analysis procedures of steel portal frames having different connections rigidities using modified stiffness matrix and end-fixity factor concept

Ali Msabawy[1,2,*] and Fouad Mohammad[1]
[1]School of Architecture, Design and the Built Environment, Nottingham Trent University, Nottingham, NG1 5JS, United Kingdom
[2]Engineering Works Division, Directorate of Civil Status, Passports and Residence, Ministry of Interior, Babylon, Iraq
Email: alimsabawy@gmail.com.

Abstract. The real behaviour of connections in the steel buildings is often underestimated by designers at the structural analysis and design stages, despite their influences on the structural performance, deflection limits and a possibility of achieving a reduction in the material weights, which can significantly reduce the overall cost and amount of energy embodied. This paper, therefore, described systematic and simplified procedures to conduct a first-order elastic structural analysis of the semi-rigid steel portal frames in order to implement a design optimization using a Generalized Reduced Gradient (GRG) algorithm within Solver Add-in tool in Microsoft Excel. The written program used the robustness and efficiency of the Finite Element (FE) method with the versatility of a spreadsheet format in Excel. To simulate the semi-rigid response of the connections, the mathematical representation through End-Fixity Factor and the Modified Stiffness Matrix were used to incorporate such behaviour into structural analysis packages. To validate the written program, a computer-based analysis was conducted using Prokon® software and comparing analysis results with those obtained from the Excel spreadsheet. It demonstrates that Excel's results were perfectly accurate. Consequently, the procedure of establishing spreadsheets as a finite element analysis software for a certain form of frames demonstrates its validity.

1. Introduction

It is estimated that around 50% of all constructional steel used in the United Kingdom is in erecting single-story buildings [1]. Steel Portal frames are the most preferable constructional form in pitched-roof buildings within this major market sector [2]. This form employed for constructing industrial, distribution, retail and leisure facilities, owing to their cost efficiency, versatility, sustainable contributions as well as it is the most efficient form that can accommodate changes within the structure.

In a steel framing system, the structural elements and connections are modelled considering some idealizations [3]. Ideally, the eave, apex and base connections in a portal frame are categorized into two idealized types: the fully flexible, ideally pinned, connections and fully-rigid connections. According to BS EN 1993-1-8 [4], connections can be classified as nominally pinned when are capable of transmitting axial and shear forces without creating a considerable bending moment as well as they are able to accept the resulting rotations under the design loads. While connections can be classified as fully-rigid in case they are not free to rotate and can transmit all three types of forces: axial, shear and bending moments.

In practice, the connections of steel structures often behave in a manner that falls somewhere in between these categories, owing to friction and material behaviour, in which connections can both transmit moment and experience some rotation that can contribute substantially to overall frame displacements [5]. The term semi-rigid connections, or joints, are commonly adopted to denote such form of steel connections. A connection may be classified as semi-rigid by exhibiting a behaviour intermediate between that of standard pinned and rigid. It is common for the structural engineer to idealize the frame and behaviour of connections as to simplify the analysis and design processes. However, the predicted response of the idealized structure may be quite unrealistic compared to the response for the actual structure [6].

For instance, the fully rigid connection assumption may lead to underestimate of structure drift and overestimate of structure strength, while the ideally pinned connection assumption may result in an overdesign of the rafters and an underdesign of the columns [7]. In addition, ideally pinned connections must have adequate flexibility to accommodate rotations without developing significant moments which can result in premature failure of the structure or parts of it. Hence, by treating the connections as semi-rigid, two major advantages can be obtained. Firstly, a more reliable prediction of structural behaviour. Secondly, the possibility of achieving greater economy by making use of the stiffness and strength of connections that would otherwise be considered as pinned, as well as by avoiding the stiffening often required in rigid connections [5]. For illustration, figure 1 presents the results of a linear analysis obtained for simple and pitched-roof steel portal frames, subjected to a uniform distributed load, with rigid and semi-rigid beam-column connections. All the above emerges the necessity of examining the structural performance for semi-rigid frames, in which their connections can influence the stability, weight and the overall behaviour of the frame.

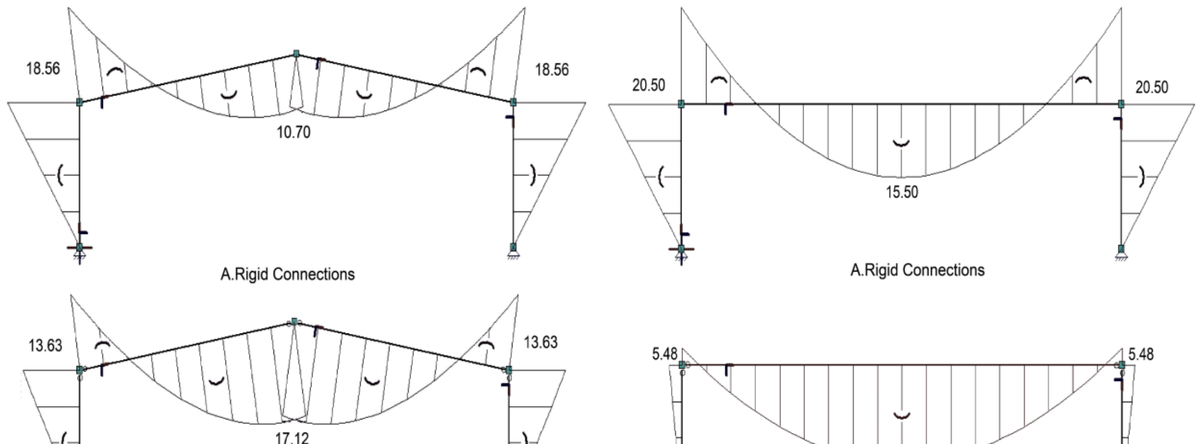

Figure 1. Bending moment diagrams of portal frames: A) Rigid connections; B) Semi-rigid connections

2. Modelling and Analysis Semi-Rigid Joints

Since that the real response of steel structures is influenced by the mechanical characteristics of their connections, namely strength, stiffness and rotational capacity. The stiffness of the connections is a key element in well-designed steel structure, the fact that the stiffness of connections affects the deflections of the entire structure, particularly in non-braced frames [8]. The most realistic knowledge of the joint stiffness behaviour and its properties can basically be observed through the moment-rotation curves that are obtained by the experimental investigation [9]. However, the experimental techniques are usually expensive and time-consuming to be implemented for daily design practice, and they are normally employed for academic purposes only.

For the theoretical modelling of semi-rigid connection within portal frames, there are two common approaches to adopt joint stiffness into a structural analysis. The first technique is to introduce "additional connection elements" to simulate the behaviour of such connections directly in the package programs such as ANSYS and SAP2000. This approach is a time-consuming and required a complex programming to be incorporated with optimization or parametric studies. The other difficulty is to achieve a physical sense of the connection member stiffness since it is detached from the attached end connections. In the literature, the effects of semi-rigidity for connections were neglected in many cases, owing to the lack of a systematic method of conducting the structural analysis for such frames. Therefore, the second technique is to model the flexibility of connections, it employs the stiffness matrix method (displacement method) and its modifications to conduct the structural analysis. Many researchers and structural engineers successfully adopted the stiffness method to analyze the two-dimensional (2D) frames due to its efficiency and ease of generating the required matrixes using computer operations, for instance, [3], [5,6] and [10].

This paper outlines the methodology adopted based on the second approach to implement a structural analysis for portal frames using the traditional stiffness method and its modifications under certain loads with different member-end restraint conditions. The conventional form of stiffness method is as follows:

$$\{\mathrm{Q}\} = [K]\{u\} \quad \text{..........................} \quad (1)$$

Where: {Q} is the 6×1-member end-force vector in the local coordinate system; [K] is the stiffness matrix of a member in the local coordinate system; {u} is the 6×1-member displacement vector in the local coordinate system.

In 2D plane, the frame element with elastic restraint has three degrees of freedom at each end, namely, horizontal displacement, vertical displacements and rotations. The axial force, shear force and bending moment represented the corresponding forces to these displacements respectively. To establish a relationship between forces and displacements, the stiffness matrices for the element are derived by assuming having fully rigid connections. Therefore, figure 2 is shown a semi-rigid member that described by Xu [10] consisted of a finite-length beam-column member with a zero-length rotational spring at both ends of the member. The connection flexibilities are modelled by rotational springs of stiffness R_1 and R_2 at both ends of the member. The relative stiffness of the beam–column member and the rotational end-spring connection is measured by an end-fixity factor that developed by Monforton and Wu [11]. The mathematical expression of the end-fixity factor r_j is as follows:

$$r_j = \frac{1}{1 + 3EI/R_j L}\,(j{=}1{,}2) \quad \text{..............} \quad (2)$$

Where: R_j is end-connection spring stiffness; E is the modulus of elasticity of the prismatic member, m; I is the second moment of area of the prismatic member, m; L is the length of the prismatic member, m.

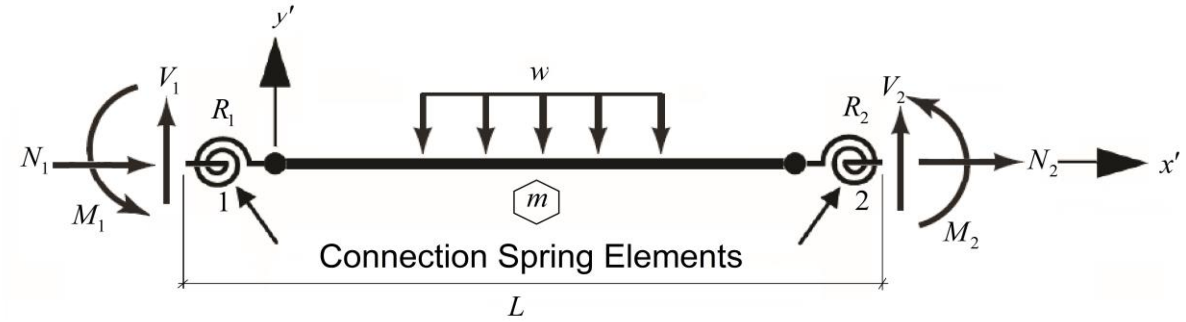

Figure 2. Model of undeformed semi-rigid member.

The values of the end-fixity factor are ranging from zero up to 1. The connection rotational stiffness of a pinned type is idealized as zero thus the end-fixity is zero, $r_j = 0$. The connection rotational stiffness of a rigid type is taken to be infinite and the end-fixity factor has a value of unity, $r_j = 1$. Hence, the semi-rigid connections are modelled with end-fixity factors between zero and one, $0 < r_j < 1$. This mathematical model made the structural analysis of semi-rigid structures a systematic and straightforward process due to its connectivity to the stiffness matrix method. In addition, it could easily be incorporated into the analytical model written in Excel spreadsheets that can handle large matrices. In which various member-end restraint conditions are modelled by setting a suitable combination such as rigid-pinned, rigid–semi-rigid or pinned–semi-rigid, with their appropriate values of the end-fixity factors at both ends of the member. The classical stiffness matrices of a rigid member, equation 1, modified by so-called correction matrix, equation 3, that embeds the end-fixity factor within it [10-11]. Consequently, the modified stiffness matrix, equation 4, was produced for member *m* having semi-rigid end-connections. The implementation of end-fixity factor approach into structure analysis is straightforward process due to its connectivity to the stiffness matrix method.

To simplify the calculation, this paper adopted the linear representation of the moment-rotation curve of such connections. This assumption was to avoid iterative the process that is too complex to be modelled into the written program. This is the simplest connection model to be used in the linear, vibration and bifurcation analysis where the deflections are relatively small [12]. The adopted methodology also assumed that all members are straight and prismatic as well as connection dimensions are assumed to be negligible compared to the lengths of the rafters and columns.

$$[C]=\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & \frac{4r_2-2r_1+r_1r_2}{4-r_1r_2} & -\frac{2Lr_1(1-r_2)}{4-r_1r_2} & 0 & 0 & 0 \\ 0 & \frac{6(r_1-r_2)}{L(4-r_1r_2)} & \frac{3r_1(2-r_2)}{4-r_1r_2} & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & \frac{4r_1-2r_2+r_1r_2}{4-r_1r_2} & \frac{2Lr_2(1-r_1)}{4-r_1r_2} \\ 0 & 0 & 0 & 0 & \frac{6(r_1-r_2)}{L(4-r_1r_2)} & \frac{3r_2(2-r_1)}{4-r_1r_2} \end{bmatrix} \quad \text{.....} \ (3)$$

$$[K]^{SR}=[K][C] \quad \text{...} \ (4)$$

2.1. Formulations for End-Reactions of a Semirigid Member

This paper adopted the modified formulations of fixed-end reactions for various loading types making the analysis of a semi-rigid member by stiffness matrix method more capable of using end-fixity factor concept. These modified formulations are applied to simulate the behavior of semi-rigid connections within steel portal frames. This paper refers to [10] and [13] for further details about the end reactions of a loaded member, that applied to different restraint conditions at the ends depending on the values of the end-fixity factor.

3. Finite Element Analysis Procedures

Finite element method (FEM) is a numerical approach for solving engineering and mathematical problems, it is a process of modelling complex problems by subdividing into an equivalent system of smaller and simpler parts [14]. The most common engineering problems are solvable by utilizing the FEM which is an essential tool in structural analysis, fluid flow, mass transport. However, the common idea in mind of structural engineers is that Finite Element Analysis (FEA) is complex and normally requires a high-priced commercial software.

Therefore, and for practical needs, the spreadsheets that available in Microsoft Excel were employed in this paper to establish FEA program. Microsoft Excel spreadsheets are free and most of the structural engineers are already familiar with Excel. Besides to the affordability and versatility of Excel spreadsheets, it presents the input information, the output results, the intermediate calculations and the secondary and supporting formulas on a single spreadsheet. This contrasts with commercial software which normally does not present intermediate steps, nor the implied formulas. Ease of modifying the spreadsheets is another feature to accommodate users who want to customize the spreadsheets for particular requirements. It can readily be connected with different spreadsheets, sequenced with others, adjustable for use with other configurations of 2D frames.
Hence, this paper developed "Finite Element Analysis Software " in a simplified spreadsheet form to conduct a first-order elastic analysis of steel portal frames with different connections rigidities, by combining the robustness and efficiency of FE method with the versatility of a spreadsheet format in Microsoft Excel. The written program was formulated by the authors in Microsoft Excel Spreadsheets, it is capable of analyzing cold-formed or hot-rolled steel portal frames having different end-connection, pinned, rigid or semi-rigid joints, between their attached members. Furthermore, this spreadsheet can implement the structural analysis of portal frames with different cases of applied loads in the global or local axes systems.

3.1. Analytical Model and Cartesian Coordinate Systems

Based on basic concepts in FEA that presented in Section 3, 2D frame was divided into members and joints for conducting a structural analysis. The first and possibly the most significant step in such an analysis is the process of preparing a so-called Analytical Model. It is an idealized representation of a real frame and its purpose is to facilitate the analysis of complex structures through presenting details about joints, members, etc. Six key steps of establishing an analytical model can be described. Define systems of coordinates firstly and for the sake of convention, this paper followed Hibbeler's approach [15] for specifying coordinate systems of both the structural members individually and the entire frame.
Hence, two systems of the coordinates were established. The first was the Local Coordinates System (LGS) that identifies the direction of the internal forces for each member within the frame and their corresponding displacement as shown in figure 3. The system was denoted by the subscripts x', y' and z'**.** The beginning i of the member m represents the origin of x'-axis that coincides with the centroidal axis (geometric centre axis) of the member in the undeformed case. The local x'-axis is positive when it be oriented from the i end toward the j end. The y'-axis is always perpendicular to x'-axis.

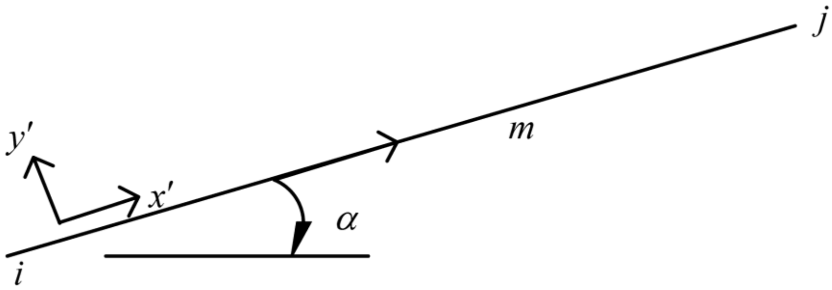

Figure 3. The local coordinates system of a member.

The second system was the Global Coordinates System (GCS) for plane frames and was denoted by the symbols *x*, *y* and *z*. Generally, it is proper to locate the origin of this system at the lower left joint of the proposed structure as demonstrated in figure 4. The *x*-axis oriented in the horizontal direction where the right side is considered the positive direction while the *y*-axis oriented in the vertical direction where upward direction is considered the positive direction. All the external loads, support reactions and joint displacements of the frame were specified using this system. In terms of the third axis (z or z'), it is perpendicular into the plane x-y or x'-y'. The rotation about the mentioned axes is considered positive if it acts counterclockwise when seen in the positive direction of the axis.

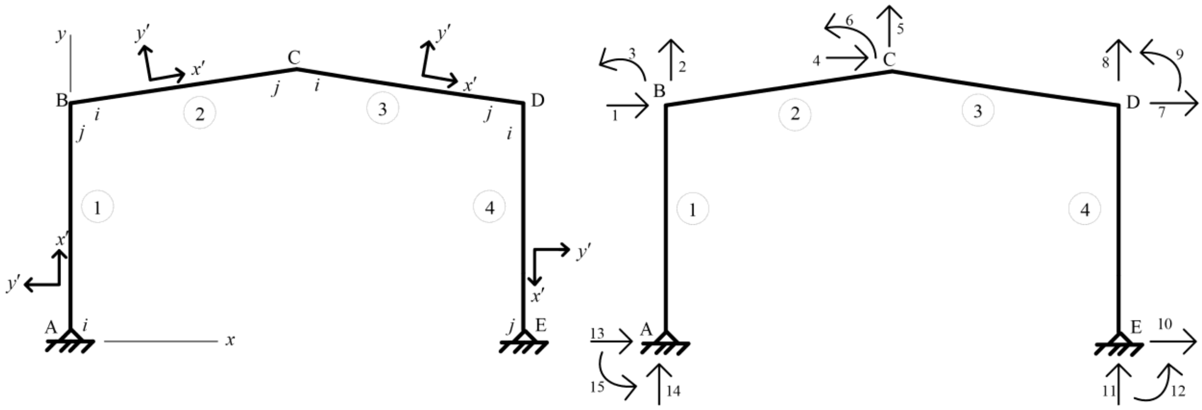

Figure 4. The analytical model of a portal frame showing global, local coordinate systems and code numbers at the nodes

Secondly, the frame must be sorted and then divided it into members and joints in accordance with FE concept, all the members must be straight and prismatic. The joints were identified alphabetically as A, B, C, etc. The structural members were divided numerically as 1, 2, 3, etc. The members' numbers were enclosed within circles to distinguish them from others. Thirdly, identify the beginning and the end of each member, in which the codes that used were *i* and *j* for the beginning and the end of a member respectively, in both the local and global systems. The fourth step was to establish a table of locations of all joints and connectivity points of the members. This table is beneficial to draw a portal frame using Excel. The table defined the locations of each joint as coordinates of *x* and *y*. For example, the coordinates of the joints A and B were (0,0) and (0, *y*) respectively where the origin of the global coordinate system was located at A for both *x* and *y* coordinates. In relation to the members, their connectivity points were defined according to the joints at their beginning and end. For instance, the member 1 and member 2 connectivity points were (A, B) and (B, C) respectively.The next step was to define the degrees of freedom of the entire structure (DOFS) which are basically the unknown displacements of its all joints. In 2D rigid frames, unsupported (unrestrained) joints can move in any direction in the *x*-*y* plane in which that will produce two linear displacements, horizontal translation and vertical translation along the *x*-axis and *y*-axis respectively as well as it can rotate about the *z*-axis. These displacements were necessary to identify the deformed position of the frame element, it called the unknown displacements. Regarding joints attached to support, it can be identified according to their type. A joint attached to a fixed support, for example, can neither translate nor rotate; consequently, it does not have any unrestrained DOF. These degrees of freedom are named "known degrees of freedom" considering values of their displacements are equal to zero. Thus, the number of degrees of freedom of the entire structure (NDOFS) can be calculated using the equation as follows:

$$\mathrm{NDOFS} = \left(3*\mathrm{NJ}\right) - \mathrm{NR} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (5)$$

Where: NR is number of joint displacements restrained by supports (equal to number of support reactions or number of DOFs have displacement equals zero); NJ is number of joints in the structure.
Lastly, figure 4 is shown the structure coordinates (coding components) that were specified numerically on the analytical model at each node (joint or support) of the frame. To implement this step, designating numbers to the arrows that were drawn beside the joints in the positive directions of the joint displacements (unconstrained and/or restrained coordinates). The code numbers at the nodes of the frame were designated with numbers assigned first to the unconstrained (unknown) degrees of freedom followed by the remaining joints that have the highest unknown of displacements.

3.2. Displacement and Force Transformation Matrices
Unlike beams, the 2D frames normally included elements that can be moved or rotated in various directions within the plane of a structure. Therefore, transformation matrices that were given in [15-16] were utilized in this paper to be able to transform the internal member loads Q_f and displacements from local to global coordinates systems. The first transformation matrix was called the displacement transformation matrix [*T*] that can be expressed as follows:

$$[T] = \begin{bmatrix} \lambda_x & \lambda_y & 0 & 0 & 0 & 0 \\ -\lambda_y & \lambda_x & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \lambda_x & \lambda_y & 0 \\ 0 & 0 & 0 & -\lambda_y & \lambda_x & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (6)$$

Where:

$$\lambda_x = \frac{x_j - x_i}{L} = \frac{xj - xi}{\sqrt{(xj-xi)^2 + (yj-yi)^2}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (7)$$

$$\lambda_y = \frac{yj - yi}{L} = \frac{yj - yi}{\sqrt{(xj-xi)^2 + (yj-yi)^2}} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (8)$$

Where: *x* and *y* are global coordinates for the beginning point *i* and the end point *j* of the member *m* in figure 3.

Therefore, the transformation relationship of an element can be expressed as follows:

$$\{u\} = [T]\{v\} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (9)$$

Where: {*v*} is the member end displacements vector in the global coordinate system.

While the second matrix was the force transformation matrix $[T]^T$ that is the transpose of the transformation matrix [*T*] that can be expressed as follows:

$$[T]^T = \begin{bmatrix} \lambda_x & -\lambda_y & 0 & 0 & 0 & 0 \\ \lambda_y & \lambda_x & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & \lambda_x & -\lambda_y & 0 \\ 0 & 0 & 0 & \lambda_y & \lambda_x & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (10)$$

Therefore, the transformation relationship of a frame element as follows:

$$\{F_f\} = \{T\}^T \{Q_f\} \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (11)$$

Where: $\{F_f\}$ is member global fixed-end force vector; $\{Q_f\}$ is member local fixed-end force vector. These matrices were adopted to transform the stiffness matrix of an element in the equation 4 from local to global coordinate systems. Hence, the final expression of the modified stiffness matrix of elements with semi-rigid joints in the global system $[K]_g^{SR}$ were as follows:

$$[K]_g^{SR} = [T]^T [K]^{SR} [T] \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (12)$$

It is worth noting that it is essential for designers to follow the same order that presented in equation 12 when multiplying the matrices because that any different order will result in incorrect solutions.

3.3. Member Loads and Member Local Fixed-End Force

Figure 5 shows that the structural elements of planar frames might be subjected to loads oriented in various directions within plane of the structure. Hence and before moving to the computation of the fixed-end forces Q_f, the loads W acting on the structural members in inclined directions must be resolved into their rectangular components in the directions of the local axes of member *m* as follows:

$$Wx' = W \sin\alpha \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (13)$$

$$Wy' = W \cos\alpha \quad \ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots\ldots \quad (14)$$

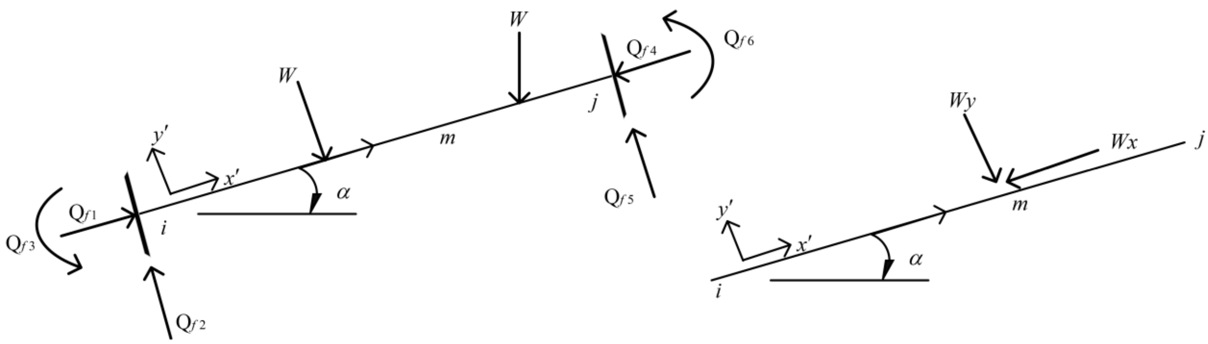

Figure 5: The relationship between member loads and member local fixed-end forces

This procedure was essential to analyze pitched-roof portal frames that normally subjected to different types of loads and due to their inclined members. In addition, the fixed-end forces equations that presented in [13] are only to process the loads that are perpendicular to the longitudinal axis of the structural elements. The directions for the member local fixed-end forces Q_f, axial and shear forces, were counted as positive when moving in the positive directions of the member's local system. Simultaneously, the local fixed-end moments were counted as positive when rotating counterclockwise. Conversely, the member loads, Wx' or Wy', are often defined to be positive in the directions opposite to Q_f directions. In other words, the member axial and perpendicular loads are

IOP Conf. Series: Materials Science and Engineering **518** (2019) 022037 doi:10.1088/1757-899X/518/2/022037

counted as positive if it was in the negative directions of the member's local x'-axis and y'-axis, respectively, and the external couples considered as positive when rotating clockwise.

4. Structural Analysis Using Microsoft Excel Spreadsheet

The key steps that required for conducting a first-order elastic structural analysis of portal frames by utilizing Microsoft Excel Platform to establish Finite Element Program were described in this section.
Step 1 was to define and draw the geometric shape of the proposed portal frame. Define node, joint or support, locations as a point in the *xy*-plane as explained in Section 3.1. Defining locations of these nodes was to facilitate the diagrammatic representation of the proposed portal frame by employing Scatter charts that available in Microsoft Excel.
Step 2 was to define the names of structural members and calculate their lengths. By employing Pythagoras' theorem and subtract the numerical values of the coordinates, *x* and *y*, of a member and the attached member at points of the start and end of these members. This procedure was to modify the arrangement of portal frames automatically. The cells in figure 6 show the formulas for computing the length of each member. In the formula SQRT ((D10-D9) ^2 + (E10-E9) ^2), for example, the SQRT refers to the square root function in Excel while the D10 and D9 are the cells that return the *x*-coordinates of the number 2 and 1 respectively. While the E10 and E9 are the cells that return the *y*-coordinates of the member 2 and 1 respectively. The Excel will calculate members' lengths automatically for any new arrangement just by changing the value of the cells.

Member Name	Member Length(m)	Joint No.	Joint No.
Member 1	=SQRT((D10-D9)^2 + (E10-E9)^2)	A	B
Member 2	=SQRT(SUM((E11-E10)^2, D11^2	B	C
Member 3	=SQRT(SUM((E11-E10)^2, D11^2	C	D
Member 4	=E12	D	E
ϴ(degree) (Roof pitch)	=DEGREES(ACOS((D11)/C16))	=DEGREES(ASIN((E11-E10)/C16))	

Figure 6: Examples of formulas for computing the length of members in Excel spreadsheet

Step 3 was to establish a table for defining the parameters of each member of the portal frame. Setting up such table was to facilitate the structural analysis process throughout its stages. Table of the parameters must be inputted manually by the structural engineer. It was entered either as a numerical value such as the loads on joints, cross-sectional area, modulus of elasticity and the second moment of the area or formulated such as λ_x, λ_y and α. λ_x and λ_y outcomes were compared with the built-in trigonometric formulae in Excel COS(RADIANS(number)) and SIN(RADIANS(number)) where the "number" was the angle value that calculated using inverse cosine or inverse sine of the angle. To obtain angles in degree system, these relationships must be preceded by function "Degrees" to convert the result from radians system that used in Excel as DEGREES(ACOS((D10-D9)/C15)).
Step 4 was to formulate stiffness matrices [*K*], of each member of the frame in LCS. [*K*] of the prismatic member, *m* is a relationship between four parameters, *E*, *A*, *I* and *L*. Each one of these parameters was entered in a table of the parameters as in step 3. It is important to notice that [*K*] must be 6 rows x 6 columns as well as it must be symmetric about the main diagonal where the upper left is symmetric with the lower right. Besides, all the elements of the diagonal of stiffness matrix must be positive to generate so-called the positive definite. Otherwise, it is a sign that the structural engineer might be made mathematical errors somewhere in the formulations of such matrices.
Step 5 was to formulate correction matrix [*C*] of each member. [*C*] was formed as a relationship between end-fixity factors at both ends of the member and its length. The structural engineers will be able to increase or decrease values for end-fixity factors and length of the member just changing the cells related to these parameters in step 3, and then the correction matrix will be recomputed accordingly.

Step 6 was to compute the modified stiffness matrix $[K]^{SR}$ for each member. $[K]^{SR}$ was calculated by multiplying the direct stiffness matrix $[K]$ and correction matrix $[C]$ of a member in their local system. Excel's function ''MMULT(array1, array2) was utilized to implement this multiplication. For the sake of validity of the formulations within this stage, the modified stiffness matrix of a member was compared with the generalized form of the modified stiffness matrix that presented in [5].
Step 7 was to generate the transformation matrices $[T]$ and $[T]^T$ for each member by utilizing equations 6 and 10 respectively. $[T]$ was implemented just by selecting the cells that contain λ_x and λ_y within the table of the parameters for the concerned member. $[T]^T$ was formulated either with the same steps for $[T]$ or by using the commands MINVERSE(array) or TRANSPOSE(array) where the "array" here represents the matrix. Step 8 was to compute the global stiffness matrix $[K]_g^{SR}$ for each semi-rigid member using equation 12. One of limitations in Excel is multiplying a triple product of matrices as $[K]_g^{SR}$. Consequently, this multiplication must be done in two stages. Firstly, $[K]^{SR}$ was multiplied by $[T]$ to produce an intermediate matrix. Secondly, Then, $[T]^T$ was multiplied by the intermediate matrix. For the sake of validity of the formulations within this stage, $[K]_g^{SR}$ was compared with the generalized form of the global stiffness matrix that presented in [15]. By assigning the end-fixity factors for both ends of the member in spreadsheet equals to 1 because the mentioned matrix is specified for rigid member. Step 9 was to compute the member local fixed-end force vector $\{Q_f\}$ and the member global fixed-end force vector $\{F_f\}$. $\{Q_f\}$ was calculated as explained in Section 2.1. $\{F_f\}$ was calculated using equation 11. It is essential to write the coding for each component of the vector as set up in the analytical model because it will be required to be assembled later., see Figure 7.

Member Local Fixed-End Force Vector		
$\{Q_f\}$	=(D159*D152)/2	
	=(((H155+H158)/D152))+((D160*D152)/2)	
	=((D160*(D152^2))/12)*(((3*D155*((2-D156))/(4-(D155*D156)))))	
	=(D159*D152)/2	
	=-(((H155+H158)/D152))+((D160*D152)/2)	
	=-((D160*(D152^2))/12)*(((3*D156*((2-D155))/(4-(D155*D156)))))	
The Member Global Fixed End Force Vector		Coding in GCS
$\{F_f\}$	=(H153*D168)-(H154*D169)	1
	=(H153*D169)+(H154*D168)	2
	=H155	3
	=(H156*D168)-(H157*D169)	4
	=(H156*D169)+(H157*D168)	5
	=H158	6

Figure 7: Example of $\{Q_f\}$ and $\{F_f\}$ in Excel spreadsheet.

Step 10 was to compute and define the joint load vector {P} and the structure fixed-joint force vector $\{P_f\}$. These vectors are dealing with forces acting on the structural members and joints in GCS. {P} represents all the loads that applied to the joints directly, the number of its components equals NDOFS. $\{P_f\}$ calculated directly by algebraically adding $\{F_f\}$ of each of degrees of freedom having a similar coding within the frame in the same location and direction. The number of its components equals the total number of all degrees of freedom of the frame.
Step 11 was to assembly of the structure stiffness matrix $[S]$. It was formulated directly by algebraically adding the pertinent components of $[K]_g^{SR}$ for each member within the frame according to their coding numbers. Alternatively, it can be formulating this matrix in spreadsheet by employing three sophisticated functions in Excel which are "IFERROR(value, value_if_error)", VLOOKUP (value, table, col_index, [range_lookup]) and MATCH(lookup_value, lookup_array,[match_type]). The square form is a distinguishing characteristic for this matrix where the number of its rows and the number of its columns equal to the total number of all degrees of freedom, known and unknown.

Moreover, these matrices for linear elastic structures are similar to local stiffness matrices where it must be symmetric about the main diagonal. Figure 8 is shown a part of [S] in Excel spreadsheet.
At this point, the analysis results were determined. Step 12 considered the first in obtaining analysis results. Compute the displacement vector that represents the unknown displacements in GCS of the frame. The values of these displacements were computed using the equations below:

$$\{\mathrm{P}\}-\{\mathrm{P}_f\}=[S]\{D\} \quad \text{...} \quad (15)$$

It also can be expressed as:

$$\begin{bmatrix} \mathrm{Q}_k \\ \mathrm{Q}_\mathrm{u} \end{bmatrix} = \begin{bmatrix} K_{11} & K_{12} \\ K_{21} & K_{22} \end{bmatrix} \begin{bmatrix} \mathrm{D}_\mathrm{u} \\ \mathrm{D}_\mathrm{k} \end{bmatrix} \quad \text{...} \quad (16)$$

Partition this matrix and substituting {D_k} to zero will result in:

$$\{\mathrm{Q}_k\}=[K_{11}]\{\mathrm{D}_\mathrm{u}\} \quad \text{...} \quad (17)$$

$$\{\mathrm{Q}_\mathrm{u}\}=[K_{21}]\{\mathrm{D}_\mathrm{u}\} \quad \text{...} \quad (18)$$

Where: {D_u} is the unknown displacements vector that corresponds to the force vector in GCS {Q_k}; {D_k} is the known displacements vector that corresponds to the support reaction vector in GCS {Q_u}; [K_ij] is the four components of [S], as seen in figure 8.

			K_{11}									K_{12}					
			1	2	3	4	5	6	7	8	9	10	11	12	13	14	15
$\mathrm{Q_k}$	=I433	1	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I434	2	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I435	3	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I436	4	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I437	5	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I438	6	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I439	7	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I440	8	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	=I441	9	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
$\mathrm{Q_u}$	R10	10	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	R11	11	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	R12	12	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	R13	13	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	R14	14	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
	R15	15	=(IFERROF	=(IFERROI	=(IFERR	=(IFERRC	=(IFERR(	=(IFERRC	=(IFERRC	=(IFER	=(IFERF	=(IFERR	=(IFERR	=(IFERF	=(IFERROF	=(IFERROI	=(IFERRC
			K_{21}									K_{22}					

Figure 8: Example of partial structure stiffness matrix in Excel spreadsheet.

Thus, {D_u} was determined from the following equation;

$$\{\mathrm{D}_\mathrm{u}\}=[K_{11}]^{-1}\{\mathrm{Q}_k\} \quad \text{.......................................} \quad (19)$$

Where: [K_{11}]$^{-1}$ is the inverse of the part of [S] that is pertinent to the force vector {Q_k}.
Step 13 was to compute {Q_u} in GCS, that was calculated directly using equation 18 and then to calculate the member end-forces vector {Q} in LGS for each member. By using the equations below:

$$\{\mathrm{Q}\}=[K]^{SR}\{u\}+\{\mathrm{Q}_f\} \quad \text{...} \quad (20)$$

$$\{u\}=\{T\}\{v\} \quad \text{...} \quad (21)$$

$$\{\mathrm{Q}\}=[K]^{SR}[T]\{v\}+\{\mathrm{Q}_f\} \quad \text{.....................................} \quad (22)$$

IOP Conf. Series: Materials Science and Engineering **518** (2019) 022037 doi:10.1088/1757-899X/518/2/022037

Equation 22 consists of two components in which the first component was a triple product. As explained previously, one of the limitations in Excel is multiplying a triple product of matrices. Thus, solving such an equation was done in four stages. Firstly, extracting the member end displacements from $\{D_u\}$. Secondly, $[T]$ was multiplied by the vector $\{v\}$. Thirdly, the resultant vector that is member displacement vector $\{u\}$ multiplied by the modified stiffness matrix $[K]^{SR}$. Finally, the resultant vector from the previous action was algebraically added to $\{Q_f\}$ and the result will be the member end-force vector {Q} in LCS. In this stage, the formulation of establishing an FE program has ended and the designer is capable of modifying any parameter within the spreadsheet. FE program in this stage has the ability to repeat all steps automatically. Equilibrium check must be satisfied for each element in the frame to ensure that the calculations of the structural analysis were conducted correctly. By applying the equilibrium conditions to the free body of each member after calculating$\{F_f\}$, each joint after calculating {P} and then for whole structure after calculating its support reactions $\{Q_u\}$. Figure 9 includes a flow-chart summarizing the procedure of establishing the FE program in this paper that was developed by the authors.

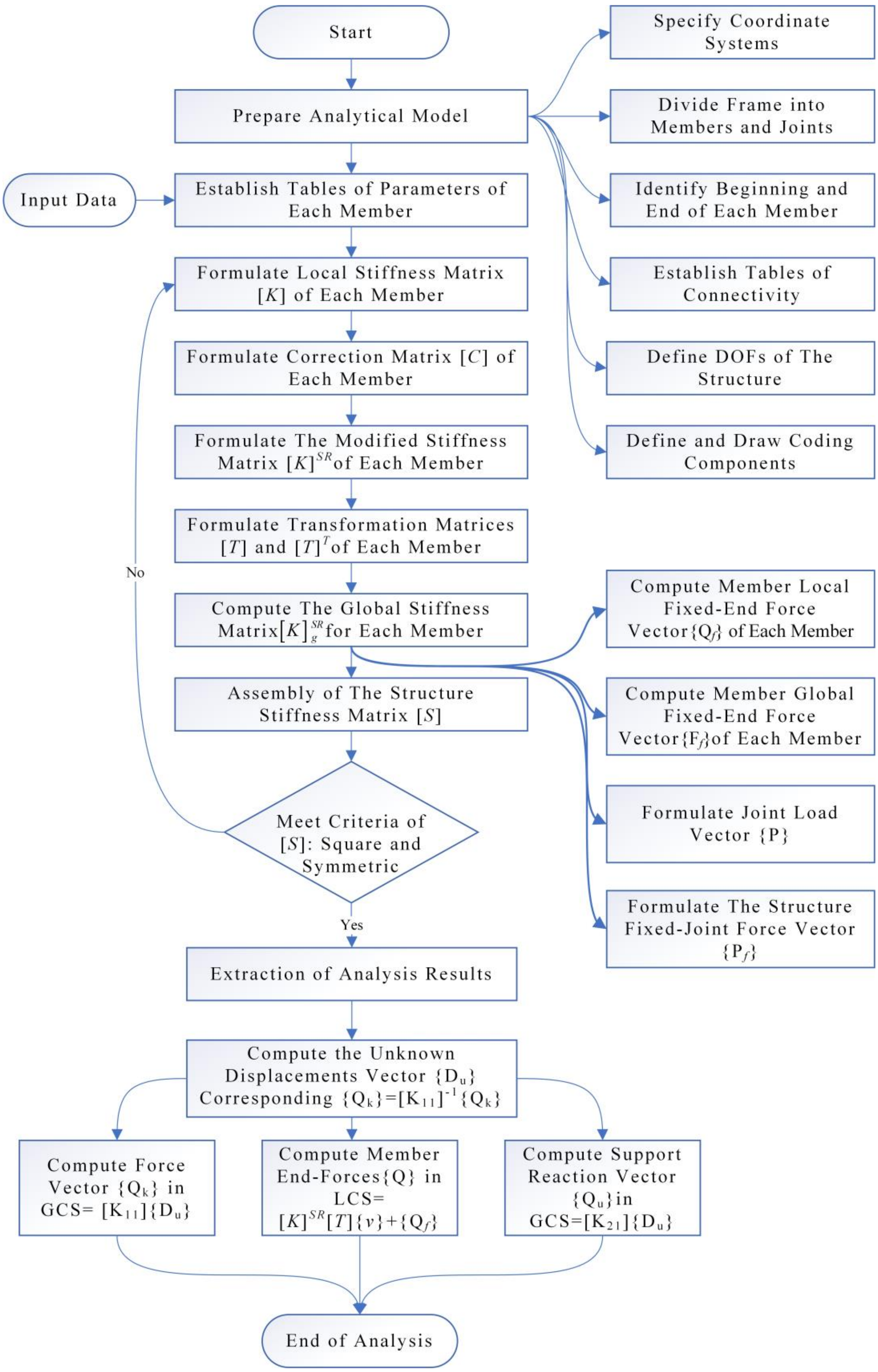

Figure 9: Computational Flow-chart of establishing the FE program in this paper.

IOP Conf. Series: Materials Science and Engineering **518** (2019) 022037 doi:10.1088/1757-899X/518/2/022037

5. Validity of Structural Analysis Procedure

A single-span symmetrical pitched roof steel portal frame with rigid joints was modelled and analyzed using the analysis procedures in Section 4 to prove the validity of suggested procedure. The frame was presented and analyzed in Kassimali's book [16, P.332].

This paper also employed Prokon® software to conduct computer-based analysis against the hand calculations that presented in the mentioned book. The same values of the unfactored forces and material properties that include cross-sectional area, modulus of elasticity and second moment of area for columns and rafters were used. This paper used the analytical model that established in Section 3.1. The obtained joint displacements and support reactions using Excel spreadsheet, Kassimali example [16] and Prokon® Software are tabulated as follows:

Table 1: The joint displacements using three different method

Joint Displacements			
Coding of unknown displacements	FE Program	Kassimali [16]	Prokon®
d1 (in)	3.447	3.447	3.446
d2 (in)	-0.009	-0.009	-0.009
d3 (rad)	-0.019	-0.019	-0.019
d4 (in)	3.952	3.952	3.952
d5 (in)	-1.315	-1.315	-1.315
d6 (rad)	0.007	0.007	0.007
d7 (in)	4.424	4.424	4.424
d8 (in)	-0.021	-0.021	-0.021
d9 (rad)	-0.009	-0.009	-0.009
d12 (in)	-0.023	-0.023	-0.023

Table 2: Support reactions using three different method

Support Reactions			
Coding of support reaction	FE Program	Kassimali [16]	Prokon®
R10 (kip)	-33.502	-33.501	-33.502
R11 (kip)	76.195	76.195	76.196
R12 (k-in)	0.000	0.000	0.000
R13 (kip)	-67.355	-67.356	-67.356
R14 (kip)	33.015	33.014	33.016
R15 (k-in)	13788.655	13789	13788.65

By comparing the analysis results that obtained through the three methods, it showed that Excel's results were perfectly accurate. Consequently, the proposed procedure of establishing spreadsheets as a Finite Element Analysis Software for a certain form of frames demonstrates its validity and efficiency.

6. Conclusions

A first-order elastic structural analysis was performed in this paper. The effects of semi-rigidity for steel connections were taken into consideration, by incorporating the well-known direct stiffness matrix with the mathematical method employing the end-fixity factor. A computer program based on a finite element method was developed and formulated by the authors utilizing Microsoft Excel spreadsheet. The developed FEA spreadsheet was capable of analysing frames having different end-connection, such as pinned, rigid joints or semi-rigid joints, between their attached members. The FE program was verified through two different methods and the results showed that the Excel spreadsheets as a Finite Element Software are perfectly accurate and capable of conducting the structural analysis of two-dimensional (2D) frames with different end-rigidity factors.

7. References

[1] Koschmidder D M and Brown D G 2012 *Elastic design of single-span steel portal frame buildings to Eurocode 3 (P397)* (Steel Construction Institute)
[2] Salter P, Malik A S and King C M 2004 *Design of Single-Span Steel Portal Frames to BS 5950-1 : 2000* (Berkshire: Steel Construction Institute)
[3] Hayalioglu M S and Degertekin S O 2005 *Minimum Cost Design of Steel Frames with Semi-Rigid Connections and Column Bases via Genetic Optimization Computers & Structures* **83** 1849–63
[4] European Standard 2005 Eurocode 3*: Design of steel structures - Part 1-8: Design of joints* (Brussels)
[5] Simões L M C 1996 *Optimization of Frames with Semi-Rigid Connections Computers & Structures* **60** 531–9
[6] Xu L and Grierson D E 1993 *Computer‐Automated Design of Semirigid Steel Frameworks* Journal of Structural Engineering **119** 1740–60
[7] Lui E M and Chen W F 1986 *Analysis and Behaviour of Flexibly-Jointed Frames Engineering Structures* **8** 107–18
[8] Ivanyi M and Baniotopoulos C C 2000 *Semi-Rigid Joints in Structural Steelwork* vol 134 (Vienna: Springer Vienna)
[9] Faella C, Piluso V and Rizzano G 2000 *Structural steel semirigid connections: theory, design, and software* (London: CRC Press)
[10] Xu L 2005 *Semirigid Frame Structures Handbook of Structural Engineering* ed W-F Chen and E M Lui (CRC Press)
[11] Monforton G R and Wu T S 1963 *Matrix Analysis of Semi-Rigid Connected Frames*, Journal of the Structural Division **89** 13–24
[12] Chan S L and Chui P P T 2000 *Non-linear static analysis allowing for plastic hinges and semi-rigid* joints *Non-Linear Static and Cyclic Analysis of Steel Frames with Semi-Rigid Connections* (Oxford) pp 123–94
[13] Msabawy A 2017 *Minimum Weight of Cold-Formed Steel Portal Frames with Semi-Rigid Joints Using GRG Algorithm* (Nottingham Trent University)
[14] Logan D L 2017 *A First Course in the Finite Element Method* (Cengage Learning US)
[15] Hibbeler R C 2018 *Structural Analysis* (Pearson)
[16] Kassimali A 2011 *Matrix Analysis of Structures* (Stamford, CT: Cengage Learning Custom Publishing)