Chapter 5

PLANE GRID ANALYSIS

5.1 Introduction

In Chapter 4, plane frame structures were introduced and the necessary theory developed for an understanding as to how they transfer loads in the plane of the frame to the foundation supports. It was shown that for plane frames the basic member forces are bending moments and axial force. In the various examples in Figure 4.6 many familiar building structure types will be recognized. Frames for office buildings, industrial sheds, bridge piers and many other applications. There is another class of two dimensional structure in which the members lie in one plane but whose loads are act transverse to this plane. These are grid structures and they have wide use in bridge decks and all type of floor support systems. Whereas loads are transverse to the plane of the grid, moments are applied to the grid in its plane as shown in Figure 5.1. In this text, the plane of the grid will be the $X - Z$ coordinate plane and the Y axis is at right angles to this plane. Grid structures differ from plane frame structures because the members of the grid are subjected to twisting moment(torque). There is now an added complication, that except for circular cross sections the evaluation of the stresses due to torque and the torsion stiffness are not straight forward. Grid structures like plane frames have rigid joints and are usually (though not always) statically indeterminate. The theory will be developed in the same way as for all the structure types in that if the grid is statically determinate the solution proceeds directly to the calculation of the member force transformation matrix by inverting the equations of static equilibrium. A simple grid, in this case statically determinate, is shown in Figure 5.2. To develop the joint equilibrium equations in the global coordinate axes, member forces are firstly set up for a single member in the local, member coordinate axes and then transformed to global axes components. For the whole structure, the three conditions still apply for the solution of the equilibrium equations with NR rows and NC columns. That is,

$$\begin{aligned}
\text{equations(NR)} &> \text{(member forces + reactions)(NC) (unstable)} \\
\text{equations(NR)} &= \text{(member forces + reactions)(NC) (determinate)} \\
\text{equations(NR)} &< \text{(member forces + reactions)(NC) (indeterminate)}
\end{aligned}$$

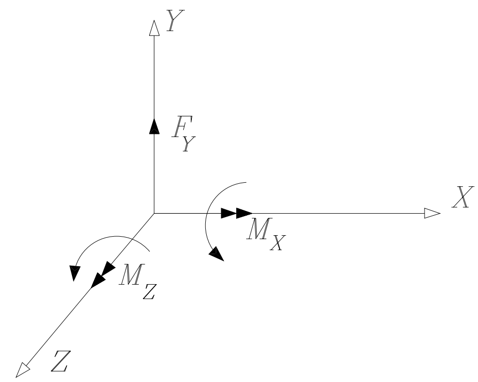

Figure 5.1: Grid forces moments and transverse load.

The software tests for all these three conditions and the determination of the member forces is possible for conditions (2) and (3). It should be mentioned that the conditions are necessary but not sufficient. The exception of unstable indeterminate structures does not usually occur and is beyond the scope of this text. A statically determinate grid (also called a cranked cantilever) is shown in Figure 5.2.

5.2 Member forces– member node forces

The basic member forces for the grid member are (M_T, M_i, M_j). That is, axial torque and bending moments. Their positive sense and local coordinate axes are shown in Figure 5.3 (a), (b). Then the nodal components Figure 5.3(b), in the X', Z' axes of the member are given by the transformation,

$$\left\{\begin{array}{c} M'_{ix} \\ M'_{iz} \\ F_{iy} \\ M'_{jx} \\ M'_{jz} \\ F_{jy} \end{array}\right\} = \begin{bmatrix} -1 & 0 & 0 \\ 0 & -1 & 0 \\ 0 & -\frac{1}{a} & \frac{1}{a} \\ 1 & 0 & 0 \\ 0 & 0 & 1 \\ 0 & \frac{1}{a} & -\frac{1}{a} \end{bmatrix} \left\{\begin{array}{c} M_T \\ M_i \\ M_j \end{array}\right\} \tag{5.1}$$

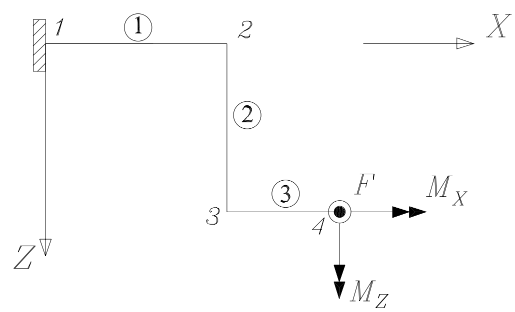

Figure 5.2: Simple grid cantilever structure in $X - Z$ plane.

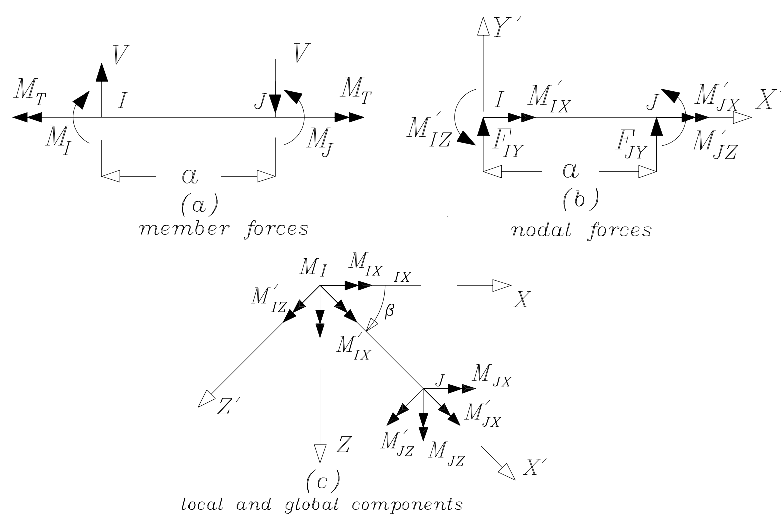

Figure 5.3: Member forces, local and global axes

Notice that the F_y force is at right angles to the $X - Z$ plane and does not require coordinate transformation. These six force components are then transformed to global components, (see Figure 5.3(c)), using the transformation,

$$\left\{ \begin{array}{c} M_x \\ M_z \\ F_y \end{array} \right\} = \begin{bmatrix} \cos\beta & -\sin\beta & 0 \\ \sin\beta & \cos\beta & 0 \\ 0 & 0 & 1 \end{bmatrix} \left\{ \begin{array}{c} M'_x \\ M'_z \\ F_y \end{array} \right\} \quad \text{or} \quad \{F\} = [L]\{F'\} \tag{5.2}$$

If equation (5.1) is written as,

$$\{F'\} = [A']\{S\} \tag{5.3}$$

then using equation (5.2) the global components are given by,

$$\{F\} = [L_D]\{F'\} = [L_D][A']\{S\} = [A]\{S\} \tag{5.4}$$

The matrix $[A]$ is thus defined by,

$$[A] = [L_D][A'] = \begin{bmatrix} L & 0 \\ 0 & L \end{bmatrix} [A'] \tag{5.5}$$

The joint equilibrium equations can be written as before,

applied force + reaction + member *node* force = 0

so that,

member force − reaction = applied force

Using this equation for all node and member forces of the grid, together with the reaction contributions enables the setting up all the joint equilibrium equations for the grid structure. For example in Figure 5.2,

number of member forces = (3×3)	=	9
number of reactions	=	3
number of joint equations	=	12

This cantilever is stable and determinate. Reactions are restricted to global components only and are given in the sequence 1, 2, 3 as shown in the Figure 5.4. For each reaction a (-1) is added to the corresponding (row, column) location in the $[A]$ matrix. When all the equilibrium equations are assembled they are written in symbolic form,

$$[A_{SM}|A_{SR}] \left\{ \begin{array}{c} S_M \\ S_R \end{array} \right\} = \{R\} \tag{5.6}$$

or simply,

$$[A]\{S\} = \{R\} \tag{5.7}$$

The equilibrium matrix is generated in STATICS-2020 by using the command,

GRIDEQ A B C

The matrix A contains the X, Z coordinates of the grid nodes in the natural order of node numbering. The topology matrix B has three columns and is an integer matrix. The

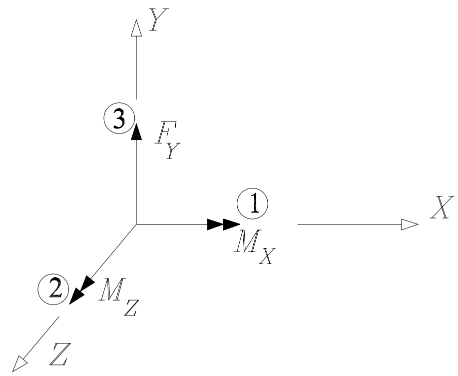

Figure 5.4: Reaction numbering sequence at a joint

first column being the member number, the second and third columns being the member end node numbers (I, J). The matrix C is an integer matrix containing the reaction data. If for example node I is fully fixed, the reaction array will contain three rows with the following information:

I 1
I 2
I 3

Then a (-1) is then placed in the appropriate (row, column) location of the $[A]$ matrix. The matrix $[A]$ in equation (5.7) is the program defined matrix EQ. If $[A]$ is a square non-singular matrix, then within the command GRIDEQ, the matrix EQ is inverted so that it now contains the $[b]$ matrix. In the 7 examples provided in STATICS-2020, see Figure 5.7, the command,

GRIDEX E=? D=?,?,?,?

generates the data matrices A, B and C for the appropriate grid in the Figures 5.7 to 5.11. The dimensions (a, b, c, d) given in Figure 5.7 are supplied in D, in that sequence. For the grid shown in Figure (5.6)-6 the 3rd and 4th values (c, d) are, (c) the number of nodes in the longitudinal members between the cross members, and (d) the number of main girders. These two parameters are restricted to the ranges (c) 1-4 and (d) 2-6. In Figure (5.6)-6 these values are 1 and 4 respectively. These are the values used in DATN.DAT, example G5. This problem may be used to study the effects of cross girder stiffness on main girder bending moments. That is, to study the transverse distribution of applied loads to the main girders.

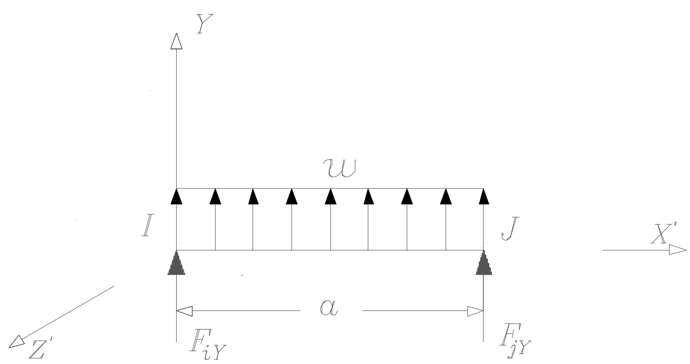

Figure 5.5: Uniformly distributed load in member coordinates

5.3 Node and member loads

Loads are restricted to concentrated loads and moments applied to the nodes and uniformly distributed loads applied over the whole of a member length. For the nodal forces the command GRIDLD may be used. The command is,

GRIDLD B E F C=? D=?

The options are:

C = 0, 1; concentrated loads 0 = none, 1 = present

D = 0, 1; distributed loads 0 = none, 1 = present

The load vector is generated from E and F in the program defined array 'LO '. The matrix E for concentrated loads will have one row per load value with the following information:

[node number] [force or moment direction $(M_x - 1, M_z - 2, F_y - 3)$] [magnitude]

The moments are positive clockwise when looked at from the origin along the axis. Uniformly distributed loads if present in F will be positive in the positive Y direction, see Figure 5.5. For the statically determinate grid the end forces due to F for the loaded members will be given in equation (5.8).

$$\{F'\} = \frac{wa}{2}\begin{Bmatrix} 0 \\ 0 \\ 1 \\ 0 \\ 0 \\ 1 \end{Bmatrix} \tag{5.8}$$

These forces require no transformation to global components and are added to the nodal load vector LD. See the Section 5.3.1 on loads for statically indeterminate structures; for this case fixed end release moments are also required. Once the global load vector has been formed then member forces and reactions are obtained using the command, GRDFRC as follows,

GRDFRC M V S T

In this command, M are the member bending moments, V the end shears, S the structure reactions and T the member torques.

5.3.1 Statically indeterminate grids

In the case for which NR<NC, the analysis can still proceed for the indeterminate grid and the $[b]$ matrix in,

$$\{S\} = [b]\{R\} \tag{5.9}$$

calculated using compatibility conditions of the deformations in an identical manner to that for statically indeterminate trusses and beams. Starting from,

$$[A]\{S\} = \{R\} \tag{5.10}$$

the contragredient principle shows that the corresponding displacement transformation is given,

$$\{v\} = [A]^T\{r\} \tag{5.11}$$

in which $\{v\}$ is the array of the member relative twist of the ends and the end rotations relative to the chord. The relationship between $\{S\}$ and $\{v\}$ is easily established for a prismatic member of circular cross section, member (n), length l_n, area of cross section, A_n and second moment of area, I_n

$$\left\{\begin{matrix} M_T \\ M_i \\ M_j \end{matrix}\right\}_n = \{S_n\} = \frac{EI_n}{l_n}\begin{bmatrix} k_{tn}/I_n & 0 & 0 \\ 0 & 4 & -2 \\ 0 & -2 & 4 \end{bmatrix}_n \left\{\begin{matrix} \Delta\theta_t \\ \phi_i \\ \phi_j \end{matrix}\right\}_n = [k_n]\{v_n\} \tag{5.12}$$

In equation (5.12), k_{tn} is the torsion stiffness parameter for the member ($=\pi D^4/32$ for a solid circular cross section), and $\Delta\theta_t$ is the relative twist of end J to end I taken clockwise positive viewed from I. It is seen that the expression for grid stiffness simply combines torsion and bending stiffnesses. For all member forces (including reactions), these equations are combined as,

$$\{S\} = [k]\{v\} \tag{5.13}$$

The stiffness corresponding to a reaction is set equal to a large number e.g. (10^{20}). Combining equations (5.10), (5.11) and (5.13),

$$[A][k][A]^T\{r\} = [K]\{r\} = \{R\} \tag{5.14}$$

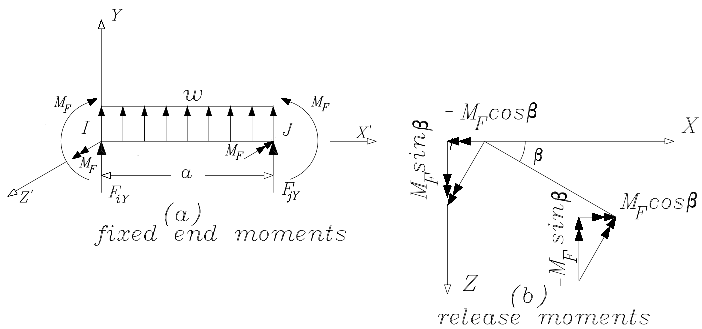

Figure 5.6: Member forces with U.D.L., fixed end moments

The structure stiffness matrix $[K]$ is now nonsingular and symmetric and the node deflections $\{r\}$ are obtained by solving equation (5.14),

$$\{r\} = [K]^{-1}\{R\} \tag{5.15}$$

The force transformation matrix from nodal to member forces is given, in the equation by combining equations (5.13), (5.11) and (5.15),

$$\{S\} = [k][A]^T[K]^{-1}\{R\} = [b]\{R\} \tag{5.16}$$

That is, the force transformation matrix for the indeterminate grid is,

$$[b] = [k][A]^T[K]^{-1} \tag{5.17}$$

Notice that this is an identical matrix transformation to those for indeterminate trusses, beams and frames so that the matrix theory presents a uniform approach to structural analysis. If the grid is statically indeterminate the condition is indicated and the user may continue the analysis as outlined above to obtain the $[b]$ matrix. The steps are thus the same as for indeterminate trusses, beams and frames. Commands are available in STATICS-2020 to make these steps. The user must firstly supply the torsion stiffness factor, second moment of area and Young's modulus of elasticity of each member. That is, a matrix with three columns is to be read in with the following values for each member,

[torsion constant] [second moment of area] [Young's modulus]

That is, there is one row for each member with 3 values. The program assumes that all members are of the same material and only the (1, 3) row value needs to be loaded with the Young's modulus. Of course, the torsion constant and second moment of area must be in units consistent with those used to describe the grid nodal geometry. The command,

LOADR I R=? C=3

is used to load these member properties into the matrix (I). The concentrated node loads and member distributed loads are processed via the command,

GRIDLD B E F C=? D=?

B=member nodal numbers.
If concentrated loads are present in E then C=1, otherwise C=0.
If distributed loads are present in F then D=1, otherwise D=0.
The matrix E has one row per node load with the information:
node number, force component identification (1, 2 or 3), magnitude.
The $(3-Z)$ component being a force in the positive Y direction, 1 and 2 being moments about the $X-Z$ axes. (see Figure 5.1). The matrix F is a row matrix giving uniformly distributed loads for all members. A zero is required for a member without loads.
The command,

GRMSTF B I MS

then calculates the matrix $[k]$ of member and reaction siffnesses. The equilibrium matrix has already been generated and stored in the program defined matrix EQ, so that the global stiffness command that generates $[b]$ and stores it in the program defined array KA, is

GRGSTF EQ MS K

Note that the same routines are used for all equations such as equation (5.17). Finally member forces and reactions are obtained with the command,

GRDFRC M V S T

In which, M are the member bending moments, V the member end shears, S the structure reactions and T the member torsions.

5.3.2 Nodal forces, statically indeterminate grids

Forces applied directly to the nodes are treated in the identical way as for statically determinate grids. However for distributed loads on members, in the kinematically determinate state, $\{r\}=0$, fixed end moments are induced by the zero rotation conditions and these with signs reversed must be applied to the nodes at the ends of the member. Thus from Figure 5.6, with w/unit length as shown in the member coordinate system the nodal release forces induced in the local coordinate system are,

$$\{F'\} = \frac{wa}{2}\left\{\begin{array}{c} 0 \\ a/6 \\ 1 \\ 0 \\ -a/6 \\ 1 \end{array}\right\} \tag{5.18}$$

The transformation of the fixed end release moments to global components is shown in Figure 5.6(b).

When member forces are calculated from the nodal loads the fixed end moments must be included in the member forces.

5.3.3 Grid analysis using examples 1 to 7 in software

The examples available are shown in Figure 5.7 and they represent typical simple grids that are met in practical analysis. Because grids tend to be indeterminate only the commands for this case are given in this section. The grid dimensions are shown in Figure 5.7 and are entered in the sequence D=a,b,c,d.

Command sequence

GRIDEX	E=? D=?,?,?,?	E=1 to 7, D gives grid dimensions see Figure 5.7
PLTGRD	A B C	plot grid, nodes, members, reactions
GRIDEQ	A B C	set up joint equilibrium equations in EQ
LOADR	E R=? C=2	read in node concentrated loads, if present
LOADR	F R=? C=?	read in member distributed forces, if present
GRIDLD	B E F C=? D=?	generate node forces
LOADR	IN R=? C=3	read in area,second moment of area, Young's modulus each member
GRMSTF	B IN MS	generate member stiffness matrix in MS
GRGSTF	EQ MS K	generate global stiffness K, b matrix in KA
GRDFRC	M V S T	calculate member forces (M,V,T), reactions S
PRINT	M	print member moments
PRINT	V	print member end shears
PRINT	T	print member axial forces
PRINT	S	print structure reactions

Note! At any stage, LIST gives matrices in the incore data base, and PRINT (array name) prints the array values. If the program stops because of an attempt to access a matrix not defined, then the RESUME command will restore incore data base when STATIC is restarted

5.4 Grid analysis module

The theory for the analysis of plane grids is given in Sections 5.2 to 5.3. See these sections for the basic theory for member forces, node equilibrium and the setting up of the node equilibrium equations. the basic equation to do this is:

GRIDEQ A B

in which A stores the node coordinates, B the member node connectivity matrix and C the support boundary conditions. Node forces can be generated using the command,

GRIDLD A B E F C=? D=?

In which A and B are as given in the command GRIDEQ, and E and F are matrices containing data for concentrated node forces and uniformly distributed loads on members respectively. See Section 5.3 for the node forces for statically determinate and indeterminate grids that are different because the first solves by statics whereas the second uses the stiffness method (see the above sections for the details). The command GRIDEQ generates the equilibrium equations and for the determinate grid inverts the matrix and stores this in the programme defined array, EQ. This is the $[b]$ matrix. Member forces are obtained from the node loads by the command, (See Section 5.3).

GRDFRC M V S X

For statically indeterminate beams, the same strategy as for the analysis of statically determinate trusses, beams and grids is used to generate $[b]$. (See Section 5.3.1, equation 5.17). Then the commands

GRMSTF B I MS

GRGSTF EQ MS K

are used. This latter command generates the $[b]$ matrix stored in the program defined array, KA. When GRIDLD is used for statically indeterminate grids, the node forces include the release fixed end moments (See Section 5.3.2 Figure 5.6(b). The exercises given in Figure 5.7, 4 to 7, are for statically indeterminate grids. The set of commands to analyse any of these exercises is given in Section 5.3.1. the exercises for grid analysis are given in assignments Section 5.4.1, The command

PLTGRD A B C A=HA,VA,ZOOM N=1

will plot a perspective view of the grid in the XZ plane viewed from the angle (HA,VA) and scaled by the factor ZOOM, coordinates are in A, member node numbers in B and reaction data in C.

PLTGRD A B M A=HA,VA,ZOOM N=2

will plot a perspective view of the grid with its bending moment diagram (M) in the XY plane viewed from the angles (HA,VA) and scaled by the factor ZOOM, coordinates are in A, member node numbers in B. Finally the command

PLTGRD A B R A=HA,VA,ZOOM N=3

will plot a perspective view of the grid deflected shape (R) in the XY plane from the horizontal and vertical angles (HA,VA) and scaled by the factor ZOOM, coordinates are in A, member node numbers in B. In most cases the viewing angles HA=40^o and V A=30.0^o should be satisfactory. The zoom factor may be chosen to give a plot of a convenient size.

5.4.1 GRID ANALYSIS TUTORIAL EXERCISES

In all these exercises use a Young's modulus of elasticity of 200×10^3Mpa if using metric units and 30×10^6lb/sq inch if using Imperial units.

5.4.2 Exercises in grid analysis

(G1) The grid in Figure 5.7 has dimensions

1. a=c=3m, b=3m.
2. a=3m c=1m, b=3m

If the members are tubular so that the relative inertias are I and $2I$, draw the bending moments for the following load cases;

1. Load in the Y direction of 1kN on node 2.
2. Load in the Y direction of 10kN on node 3.
3. Uniformly distributed load of 3kN/M on member (2).

Draw a free body diagram of node 2 in each case and proe that the forces acting on the joint are in equilibrium.

(G2) The example in Figure 5.7(b)(4) represents a typical electric pole. If it is constructed of tubular members and is symmetric then its response to loads may be considered to be
(a) loads in the $X-Z$ plane, moment in the Z direction, a planar structure response, or
(b) load in the Z direction, moments in the $X-Y$ plane, a grid response.
Use the following dimensions in the figure: a= 7m, b=c=1.5m, d=1.5m. A load of 1kN acts on node 6 in the positive Z direction.

1. Calculate the bending moments and torques in the members of the structure.
2. Draw a free body diagram for node 5 forces , using M,V,T print outs and prove that the node is in equilibrium.
3. If the tubular members are of steel, diameter 150mm, wall thickness 3mm, Young's modulus, 200×10^3MPa calculate the nodal deflections.

(G3) The grid in Figure 5.7(2) is fixed at nodes 1 to 4 both for Z displacement and Z moment. Its dimensions are as follows: a=m, b=6m. For the members 4 to 7, $I = 100, I_P = 200$, and members 8 to 10 $I = 50, I_P = 100$. Determine the bending moments in members 40 to 7 for loads (1) 10kN at 5 (2) 10kN at node 6.

(G4) The indeterminate grid in Figure 5.9(5) can be analysed using SUBMIT G4 on DATN.DAT. The following dimensions and properties are used, a=2, b=8, $I_{1-4} = 10.0, I_{5-7} = 2.5, I_{P(1-4)}, I_{P(5-7)} = 5.0$ and Young's modulus of elasticity 200.0. The purpose of this exercise is to examine how the load sharing between members 1 to 4 is influenced by the stiffness of members 5-7. Apply the following loads,
(1) Node 5, 100kN.
(2) Node 6, 100kN.

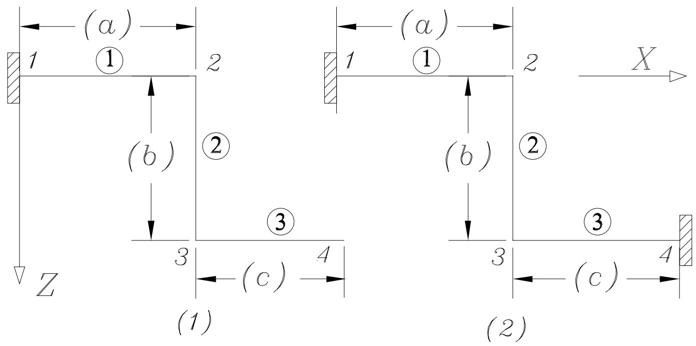

Figure 5.7: Grid examples 1 and 2

Print S and determine how much of the load is carried by 1-4. Also plot deflections R0 at 5-8. Repeat the analysis doubling the stiffnesses of members 5-7.

(G5) The indeterminate grid in Figure 5.10-(6) can be analysed using the SUBMIT G5 command. It represents a simply supported bridge, girders (4-11, 5-12, 6-13 and 7-14), with a cross girder (8-10) that distributes the transverse loading between the girders. The data generation for G5 takes the form

GRIDEX E=5 A=a,b,im,ig

See Fig 5.10-(6) for the definitions of the girder spacing a and the span b. The parameter im gives the number of nodes in each half of the span and ig gives the number of longitudinal girders. In figure 5.10-(6) im=1 and ig=4. The ranges of possible values are im = 1 to 4 and ig = 2 to 6. See DATN.DAT to determine relative inertias of the longitudinal beams to the transverse cross girder value. Analyse the system for,

(1) 100kN at node 8

(2) 100kN at node 7

(a) Compare the distributions of load to the four girders.

(b) Plot the vertical deflections of nodes 5 to 8. Increase the stiffness properties of members 6 to 8 by a factor of 10 and repeat the analysis. Repeat (a) and (b) for the new results. Compare the results of the two analyses.

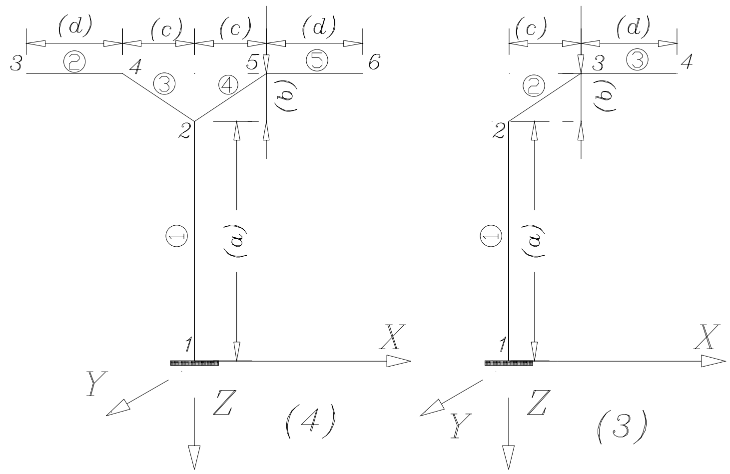

Figure 5.8: Grid example 3 and 4

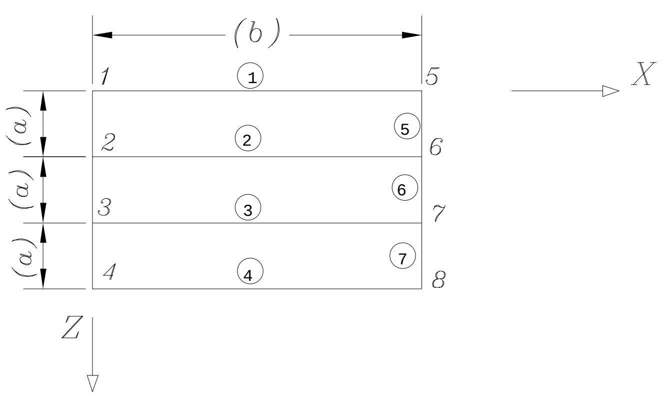

Figure 5.9: Grid example 5

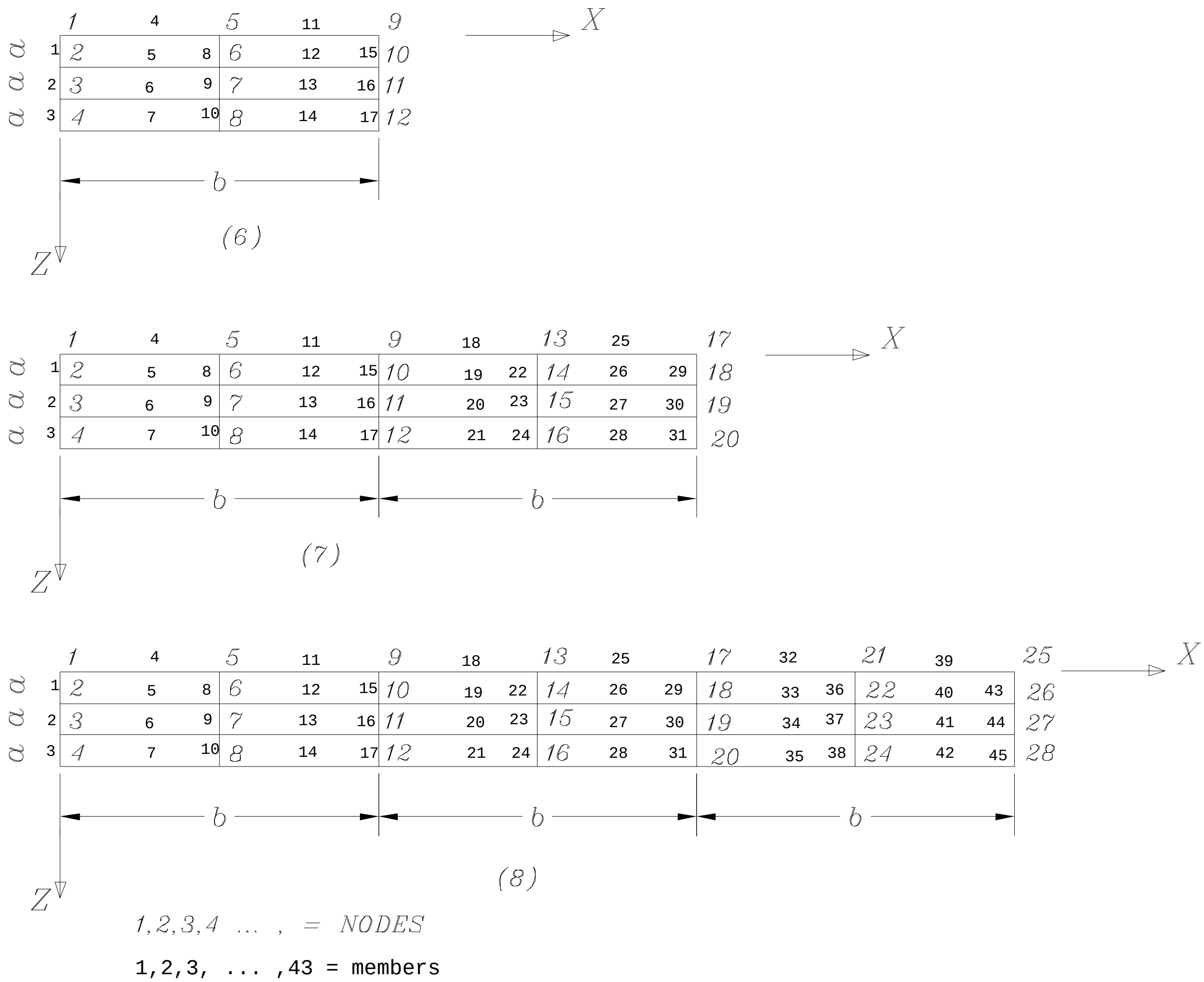

Figure 5.10: Grid examples 6 to 8

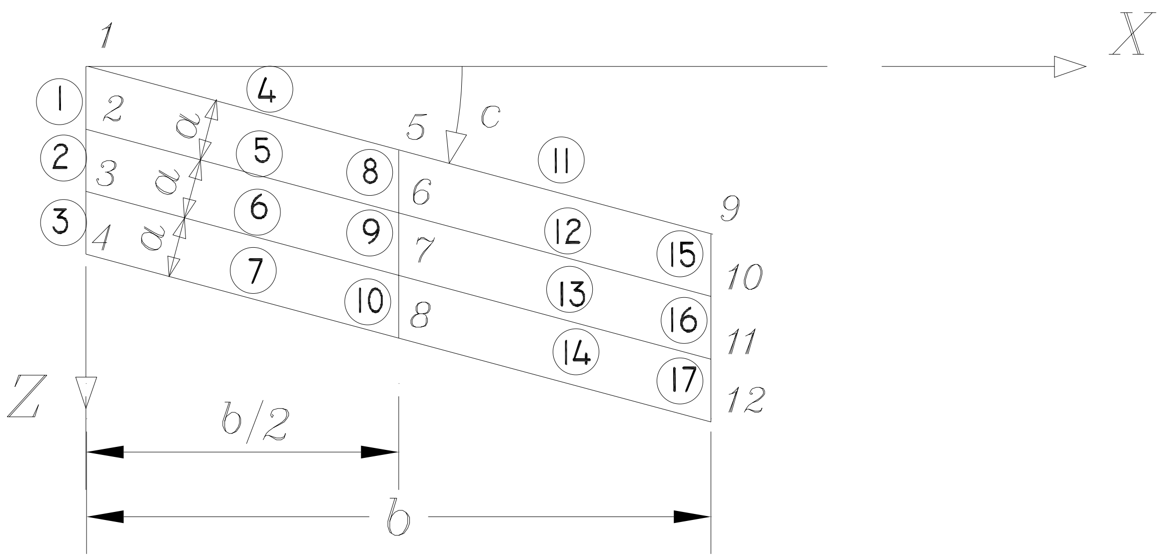

Figure 5.11: Skew grid example 9