Data formats for analysis programs

Iain A MacLeod

1. GENERAL

This section describes the basic format of data for finite element analysis programs.
With modern packages the input is generated via a graphical interface (pre-processor). The descriptions here are not based on a specific software package. Conventions for input are varied and software documentation needs to be consulted in particular cases.

2. POINTS, LINES AND SURFACES

Some systems require a definition of geometry before information about the analysis model is defined. The geometry is defined by - *points* which represent positions, *lines* which connect points and *surfaces* which are defined by lines.

3. POINTS AND NODES

Point A 'point' is normally defined by co-ordinate values relative to a set of cartesian co-ordinate axes - the global axes. Points are typically at intersections between structural members, corners, ends of lines, between endpoints, etc.
Node The behaviour of the elements and of the system is defined in relation to the force actions and corresponding displacements at *nodes*.
For a cartesian co-ordinate system, the positions of the points/nodes are defined in a table of the type shown in Table 23.1

Some analysis programs work with nodes only and do not use the concept of points.

Table 1 Co-ordinates Table

Point/Node Number	X	Y	Z
1			
2			
etc.			

For two dimensional systems the third co-ordinate value is zero. The origin of the co-ordinate system is arbitrary but is normally at a low corner of the structure. The co-ordinate values should be quoted to the accuracy to which they can be defined (usually to the nearest millimetre).

4. MEMBERS, ELEMENTS, MESHES

Member The term 'member' denotes a component part of the structural system ,e.g. a beam or a column.

Element is a mathematical representation of part of the structure which forms part of an analysis model. The behaviour of a member may be defined by one or more elements.
An element is defined by:

- A set of node numbers.
- Geometric properties. These may comprise: cross-sectional properties, thickness, etc.
- Material properties such as values for Young's modulus, Poisson's ratio, density, etc.
- A definition of local axes for 3D elements.

Mesh A 'mesh' is a set of elements defined by:

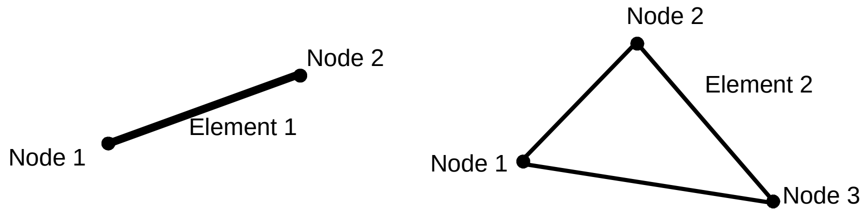
- The type of element e.g. beam elements, quadrilateral shell elements, etc.
- The element 'discretisation' which is the shape and spacing of the mesh of elements

5. UNITS

Some systems require input in specified units and also control the output units. Other systems use a single set of units for both input and output. In the latter situation the user is free to name the units but there is no units conversion.

6. NUMBERING

Points, nodes and elements are given numbers normally assigned by the system. Each element has a number of nodes associated with it. For example a line element (i.e. an element the properties of which are defined along a line e.g. beam elements and bar elements) will have at least two nodes - one at each end. A triangular element will have at least three nodes. - see Figure 1



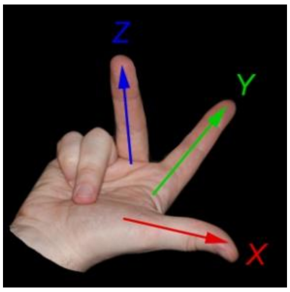
(a) 2 Noded beam element (b) 3 Noded triangular element

Figure 1 Node and Element Numbers

7. AXES

Global Axes

A set of *global* cartesian co-ordinates is normally defined for the system. X is normally a horizontal axis and either Y or Z are vertical. The position of the origin is arbitrary.



Orientation of axes The Cartesian axes 'X', 'Y' and 'Z' are normally based on a 'right hand rule' defined as follows:
Make the thumb, forefinger and middle finger of the right hand point at right angles to each other. If the thumb points in the X direction and the forefinger in the Y direction then the middle finger points in the Z direction.

Sign of rotation/moments Positive rotation/moment about an axis is normally based on the 'right hand screw rule' defined as follows:
Point the thumb of the right hand in the positive direction of the axis. The fingers of this hand then curl around the axis pointing towards the positive rotational direction. This can also be stated as 'positive clockwise looking down the positive direction of the axis'

Local Axes

Loading, properties, output actions and displacements are often defined in relation to local axes i.e. axes which are defined by element geometry rather than by global geometry.

Relationships between global and local axes

Line elements defined in a plane Figure 2 shows a line element in a global X,Y plane. The local x axis is along the length of the element for straight elements (and tangential to the line for curved elements) positive from Node 1 to Node 2. The local y axis is in the global XY plane at right angles to the local x axis (positive towards the positive direction of the Global Y axis). The local z axis will be out of the plane (towards the observer when using the right hand rule).

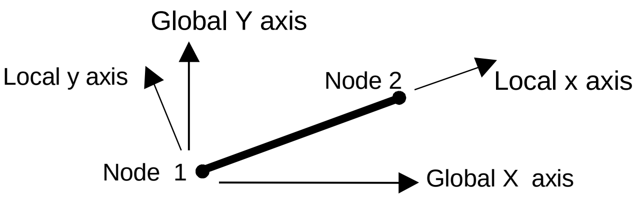


Figure 2 Local and Global axes in a plane frame element

Line elements defined in three dimensions In this case the two end nodes alone do not define the local y and z directions. Methods of defining these directions include:

- A third node - Node 3 - is defined such that the Node 1, Node 2 and Node 3 define the local xy (or xz) plane. The positive direction of the y (or z) local axis is from End1 towards the side on which Point 3 lies - Figure 3
- By defining a rotation angle (normally symbolised as 'β') for the local y (or z) axis from a default position.
- By defining a matrix of direction cosines for the local axes in relation to the global axes

If the software being used normally selects a default direction to relate the local axes to the global axes then the user may be able to alter this using one these techniques.

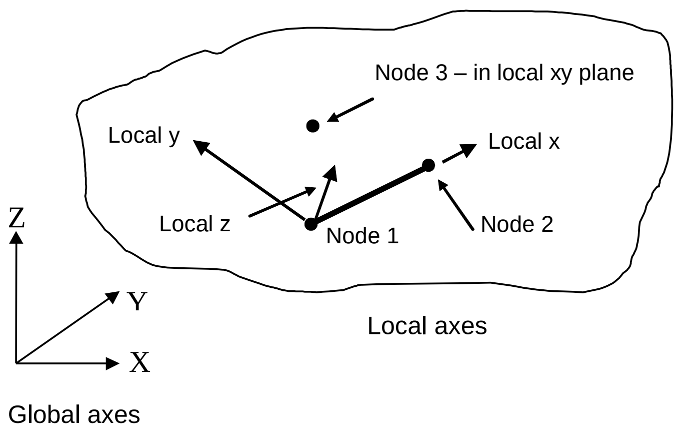


Figure 3 Definition of local axes by a third point

Local axes for surface and volume elements Elements of these types have a number of points associated with them. Normally the first two points in the list of points for the element define a local x axis and the xz plane is defined as for line elements

8. GEOMETRICAL PROPERTIES

'Geometrical Properties' define those geometry features which cannot be derived from the co-ordinates of the element nodes.

Line elements in a plane For a 2D beam element (i.e. element with axial, bending , shear and torsional deformation) the following cross-sectional properties are needed:

- A_x - the cross sectional area
- I_z The second moment of area of the cross section about the z centroidal axes

Also where shear deformation is taken into account

- A_y The shear area for shear in the y direction. This is only important for deep beams (span/depth < 5). The shear are is calculated using: A_y =

$A \times C_s$ where A is the cross-sectional area of the section and C_s is the area shear factor which depends on the shape of the section.
To neglect shear deformation define a large value for the shear area (e.g. $A_x \times 100$).

Line elements in space For a 3D beam element the following cross-sectional properties are required in addition to the 2D values:

- J The torsional constant for the section
- I_y The second moment of area of the cross section about the y centroidal axes
- I_{xy} The cross product moment of area (for elements which take account of unsymmetrical bending).
- A_z (if shear deformation is included)

Eccentricity Some elements allow the nodes to be connected to subsidiary nodes by rigid links. This is useful for modelling composite action of beams and slabs for example - See Section . It is common to define such eccentricity in the local z direction of the element, positive in the positive direction of the local z axis.
Surface elements The thickness (or thicknesses for an element which does not have uniform thickness) of the element is needed as a minimum

9. MATERIAL PROPERTIES

The main material properties required are:

- Young's Modulus
- Poisson's ratio
- Mass density
- Coefficient of thermal expansion

10. RESTRAINTS/ SUPPORTS

The terms *restraint* and *support* tend to both mean a limit placed on the displacement at a node (Section 6.1). Restraints can be in the form of:

- A 'fixity' where the displacement corresponding to a degree of freedom is set to zero
- A fixed movement
- A spring restraint.

Structural analysis programs assume that nodes are unrestrained except those which are defined as support nodes. A *support node* is one which has at least one degree of freedom restrained.
Table 2 is a typical table to define fixities.

Table 2 Supports Table

Node	**DX**	**DY**	**DZ**	θ_x	θ_y	θ_z
1	R	R	R	R	R	R
2	R	R	R	F	F	F
3	F	R	F	F	F	F

Key : DX, DY, DZ - Displacements in the X,Y,Z directions
θ_x θ_y θ_z - Rotations about the X,Y,Z axes
F - Free, R - Restrained

Table 2 is for space frames.
For plane frames DX, DY and θ_z (or X, Z and θ_y) are defined.
For grillages DZ and θ_x and θ_y (or other corresponding combination) are defined.

In Table 2:

- Node 1 is fully fixed (no movement at any of the freedoms)
- Node 2 is pinned (no restraint to rotational movement)
- Node 2 has a single restraint in the (veertical) Y direction corresponding to a roller joint.

11. LOADING

Loading is defined as either on nodes or on elements. Element loading can be defined in local or global co-ordinates. Combinations of load cases can also be specified.

The sign of a direct force/load is positive in the direction of the axis to which the force is parallel. The sign of a moment is normally based on the right hand screw rule (Section 7).

12. OUTPUT

Displacements These are typically quoted at the nodes in global co-ordinates
Support Reactions Also quoted in global co-ordinates
Element End Forces Normally quoted in local co-ordinates
Element Forces (i.e. stress resultants) Axial force, bending moments, shears, torques.
Stresses Direct stress and shear stresses