CUSTOMER SUPPORT NOTE

Influence lines and surfaces

This support note is issued as a guideline only.

Table of Contents

1. The influence line and surface capability

Influence lines are enabled for all effects, including displacements, forces, moments and reactions. They may be specified in element, local or global axis conventions according to requirements and may be used in conjunction with bar, beam, plate and shell elements, including combinations of elements where the degrees of freedom in connected elements match.

This document is divided into a number of sections. The overall pattern is as follows

1. What is an influence line/surface?

2. How is this facility used in Modeller?

   • The overall method is introduced

3. What are the underlying principles?

   • Specific examples are given

2. Introduction

In linear finite element analyses, the response of *any* point in a structure to loading for which the point of application is *fixed* is typically of interest. However, in some applications, the response at a single point in a structure to loading for which the point of application is *changing* is required. Such is the case in bridge design, in which it is useful to know how a change in position of the applied loading (a moving vehicle) would influence various response effects such as support reactions, shear forces and bending moments at a given point.

To measure this variation, the following relationship is used

$$Effect = f * Load$$

where the *effect* might be a bending moment or a shear force. The *load* may consist of any applied loading type and the non-dimensional coefficient (*f*) is normally termed the *influence* coefficient. The evaluation of the influence coefficient at multiple positions in a structure will generate an *influence line* (two dimensions) or an *influence surface* (three dimensions). Because of the above relationship we can state that

> *An influence line/surface shows how the <u>effect</u> (or influence coefficient) <u>varies at a specified point</u> in the structure as a single load moves across the structure*

The knowledge of the shape of the influence line or surface guides the positioning of the loading in the structure to produce the maximum effects. The design for loading under such conditions will then ensure adequate structural strength.

Note that the influence line for any *effect* is independent of the magnitude of the *force* which is, therefore, customarily taken as unity.

It is important to distinguish between bending moment/shear force diagrams and influence lines for bending moment/shear force. A bending moment diagram shows graphically the value of the bending moment at all sections of a structure

under a load whose position is fixed. An influence line for bending moment shows graphically the bending moment at a single position for all positions of a movable load.

For brevity in this document, general references to influence *lines* will implicitly understand the inclusion of influence *surfaces*.

For further reading on the general application and development of influence lines, the following text may be of help: "*Structural Analysis" R.C Coates, M.G Coutie & F.K Kong*.

3. Typical usage

> *"Given a loading case (e.g. vehicle loading) and a point of interest for output (e.g. design point) on a structure (e.g. bridge), find the position of the load which gives the worst case effects (e.g. maximum bending moment)"*

Modeller enables the specification of various points of interest to be defined at the pre-processing stage. An *influence line dataset* is defined to prescribe what effect, at which node and for which degree of freedom the effect is required. Any number of influence line datasets may be defined - each dataset generating a separate data file at the tabulation stage. Each data file will produce an influence line corresponding to one of the influence line datasets defined. See the Modeller user manual for further details.

The influence line produced in Modeller effectively replaces the traditional Anger's charts, giving the displacement or force ordinates at every node point.

The Modeller form to define an influence line dataset is as follows (version 13.6+).

Influence Line Parameters	Parameter Choices	Notes
Freedom type	*U, V, W, THX, THY, THZ*	*The degree of freedom (direction) for which the influence line will be generated*
Influence type	*Forces* *Reactions* *Displacemens*	*The required effect that the influence line is to show*
Influence axes type	*Element_Axes* *Local_Axes* *Global_Axes*	*The freedom type is relative to the axis system chosen. The choice is a matter of convenience. Typically, beam/gril elements would use local axes and plates/shells would use global axes*
Local coordinate dataset		*For Local_Axes* • *The local coordinate dataset name to be used to define the axis system* *For Global_Axes* • *The parameter is not used*
Displacement with respect to chosen axes	*Negative/* *Positive*	*Influence node deflection directions. This will be discussed more fully later*
Influence node number		*Node number for which the influence line is desired.*
Break away element numbers		*Element number(s) to one side of the chosen influence node to fully define the mesh separation position.* *For force effects (moment/shear) specify those element(s) to be disconnected from the structure. This input is not required for reaction and displacement effects since structural separation is not required. This will be discussed more fully later*
Influence line title		

4. General Example
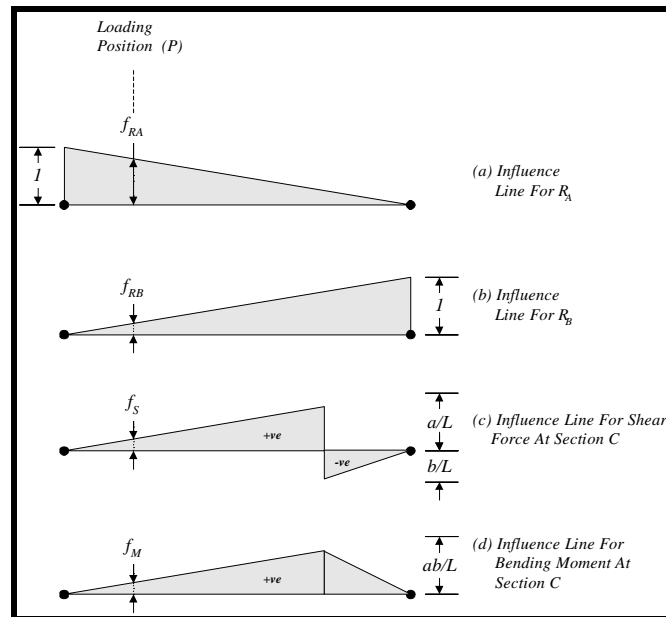
4.1 Theory

Consider the following simple case of a simply supported, two dimensional beam under the action of a concentrated load P (assumed a unit load). The object is to determine the position of this load (defined by the coordinate x) which will generate the worst case of bending and shear at point C.
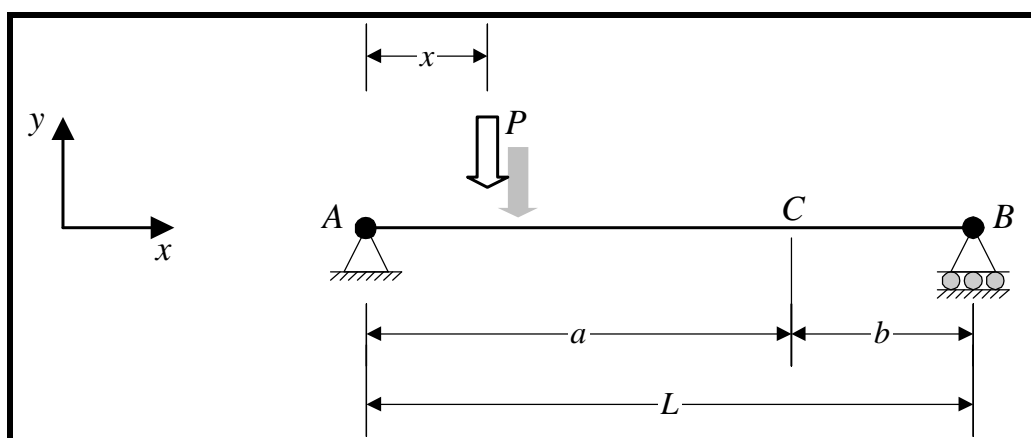
The influence lines in this instance are as described in the following diagram:

(a) Influence Line For R_A

(b) Influence Line For R_B

(c) Influence Line For Shear Force At Section C

(d) Influence Line For Bending Moment At Section C

Diagram (a) shows the influence line of the support reaction at point A due to the change of position of load P. The ordinate of this line at any position (e.g. f_{RA}), when multiplied by the force P, thus gives the magnitude of the reaction R_A when the force is at that position. That is:

$$R_A = f_R \cdot P$$

where:

$$f_{RA} = (L-x)/L$$

The influence lines for R_B and both the shear and bending are also shown in diagrams (b-d).

4.2 Modeller comparison

How is the previous example carried out in LUSAS? Firstly, define the beam geometry as:

L = $12\ m$
A = $0.25\ m^2$
I_{zz} = $0.00520833\ m^4$
Kt = $0.0104167\ m^4$ (Torsional constant, J)
A_{sy} = $250\ m^2$

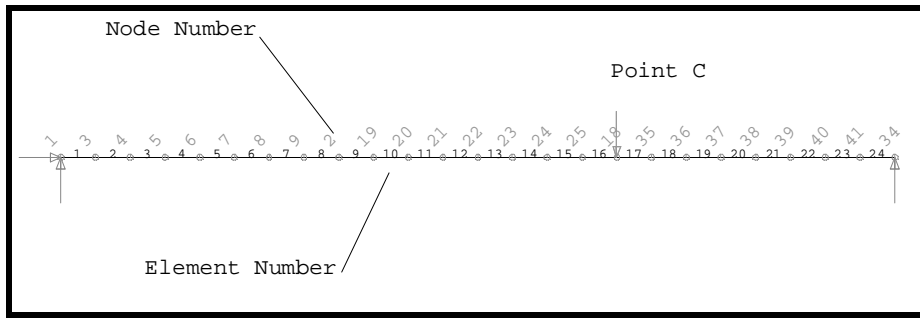and the material as:

Young's Modulus = $210E9\ N\ m^{-2}$
Poisson's Ratio = 0.3

Point C is located at a distance of 8m from end A.

The mesh used is shown in the following diagram:

where the node number corresponding to point C is 18 and the element number chosen to break the mesh is 16.

4.2.1 Example: Influence line for bending

The influence line dataset requires the following settings:

Influence Line Parameters	Parameter Choices
Freedom type	*THZ*
Influence type	*Forces*
Influence axes type	*Local*
Local coordinate dataset	*Local coordinate name*
Displacement with respect to chosen axes	*Positive*
Influence node number	*18*
Break away element number(s)	*16*
Influence line title	*Bending Moment (M_z)*

Plotting the deformed mesh in Modeller and annotating the *displacement* in the Y-direction gives the following diagram. This is the influence line for the M_z effect at point C.

Influence Line For Bending (Mz) At Point C
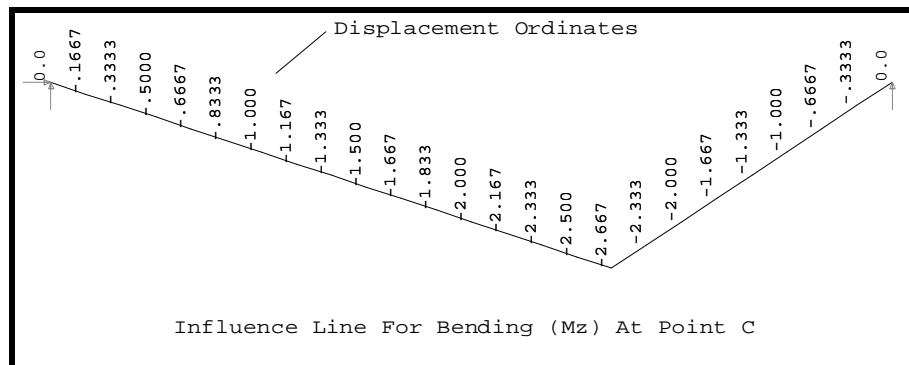
From before

$$M_z = f_C \cdot P$$

The maximum value of f_C occurs at point C and is -2.667, so that

$$M_z = -2.667 \cdot 1 = -2.667$$

This may be verified by running a standard analysis with the applied unit load at point C and inspecting the bending moment result. M_z at point C will be seen to be -2.667 as shown in the following diagram:

Bending moment diagram
(Unit load at Point C)

If the load P is in units of kN and the node coordinates were input in metres, the bending moment will be -2.667 kNm. Note that the influence load is specified in the direction of the expected *positive* ordinate of the influence line at the point of interest.

If the maximum bending effect at point C is required with a *two point* vehicle loading with wheel spacing of 0.5m, the position of the two load points needs to be located to maximise the summation of the ordinates. In this case the most obvious locations (given a nodal spacing of 0.5m) would be:

Loaded Nodes	*Ordinate Summation*	
25 & 18	2.500+2.667	= 5.167
18 & 35	2.667+2.333	= 5
Symmetrically over node 18	½ (2.500+2.667)+ ½ (2.667+2.333)	= 5.0835

Hence, placing the load over nodes 25 & 18 will produce the worst case bending moment (M_z) at point C. In general, to maximise the effect due to loading, it is necessary to obtain the maximum value of:

Effect = $\Sigma f_i P_i$

Where f_i is the ordinate of the influence line at the position of the point load P_i. For multiple loads, therefore

Effect = $u_1 P_1 + u_2 P_2 + \ldots\ldots + u_n P_n$

Modeller will not perform this optimisation automatically. For complicated three dimensional analyses the program AUTOLOADER is recommended to obtain the maximised load position(s) prior to the analysis.

4.2.2 Example: Influence line for shear

The influence line dataset requires the following settings:

Influence Line Parameters	Parameter Choices
Freedom type	*V*
Influence type	*Forces*
Influence axes type	*Local*
Local coordinate dataset	*Local coordinate name*
Displacement with respect to chosen axes	*Positive*
Influence node number	*18*
Break away element number(s)	*17*
Influence line title	*Shear Force (F_y)*

Plotting the deformed mesh in Modeller and annotating the *displacement* in the Y-direction gives the following diagram. This is the influence line for the F_y effect at point C.
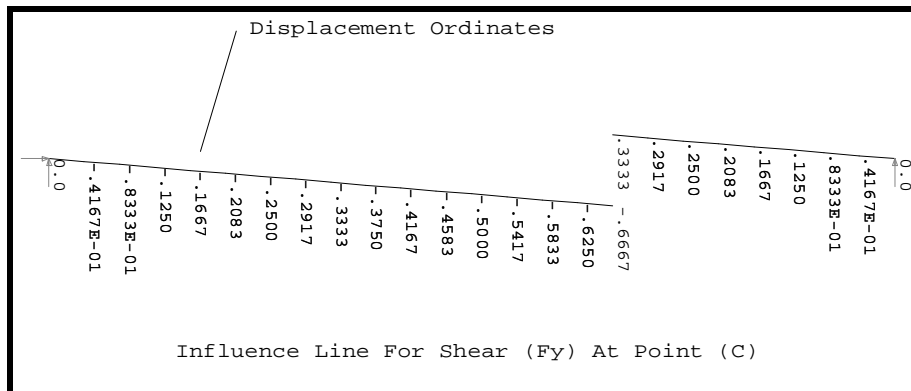
The maximum value of F_C occurs at point C and is -0.6667, so that

$F_y = -0.6667 . 1 = -0.6667$

This may be verified by tabulating without influence lines and with the applied unit load at point C and inspecting the shear force results. f_y at point C will be seen to be 0.667 as shown in the following diagram:
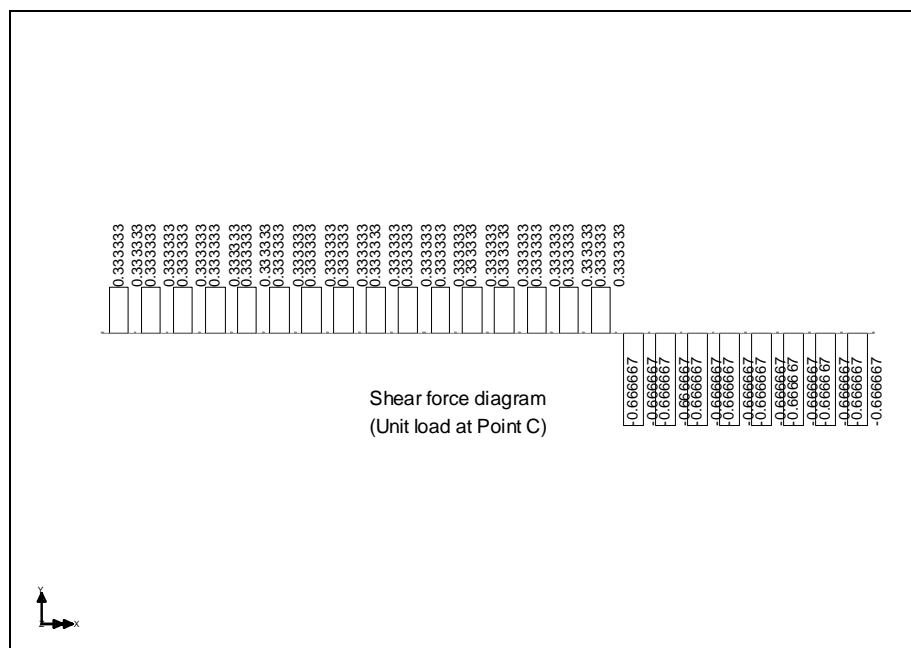
5. Muller-Breslau principle

5.1 Definition

The Muller-Breslau principle is one of the most generalised and effective methods to obtain influence effects and which has been implemented into LUSAS Modeller. A knowledge of this principle is required to select the following influence line dataset definition parameters appropriately:

- *Influence axes type*

- *Local coordinate dataset or element number*

- *Displacement direction with respect to chosen axes*

- *Element numbers (elements to separate)*

The Muller-Breslau principle may be stated, for force (bending, shear, etc.) effects, as follows:

> *"The ordinates of the influence line for any **action** at a point in a structure are equal to those of the deflection curve obtained by releasing the degree of freedom corresponding to this action and introducing a corresponding **unit distortion** at that point"*

and for displacement (translation, rotation) effects, as:

> *"The ordinates of the influence line for any **displacement** at a point in a structure are equal to those of the deflection curve obtained by introducing **a unit force** corresponding to this displacement in the structure at that point"*

To explain, consider the following two dimensional structure under the action of three point loads (x-direction is defined by the beam axis and the positive y-direction is vertically upwards).
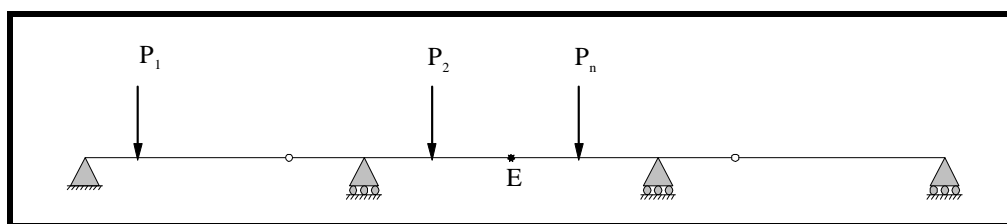
According to Muller-Breslau, the influence line for the *bending moment (M_z)* at point E is generated by introducing a θ_z hinge at E, thus releasing the bending moment at this section. Two equal and opposite couples are then applied to the two ends of the adjacent beams forming the hinge*, so as to obtain a unit rotation corresponding to the sign of the positive bending moment.* No other relative translational distortion at the ends of the beams may be permitted to occur at E.

Similarly the influence line for *shear force* at point E may be generated by introducing a mechanism whereby the ends of the beam may translate relatively in the *v* direction but not rotate or translate relative to one another in the *u* and θ_z directions. The shear influence may be generated by separating the structure at point E and introducing two vertical roller supports to the two adjacent nodes as shown in the following diagram.
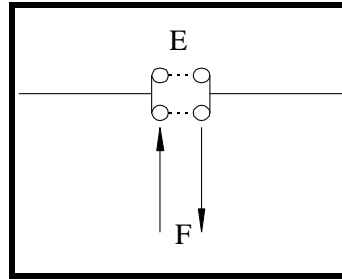
5.2 Modeller implementation

The mechanisms involved in the Muller-Breslau approach have been implemented in Modeller using the *constraint equation* technique (for further information see the Modeller user manual), whereby the structure is physically separated at the point of interest and an appropriate constraint equation introduced automatically between the disconnected elements.

Considering again the *bending moment* influence at point E, the structure is separated at that point as follows:
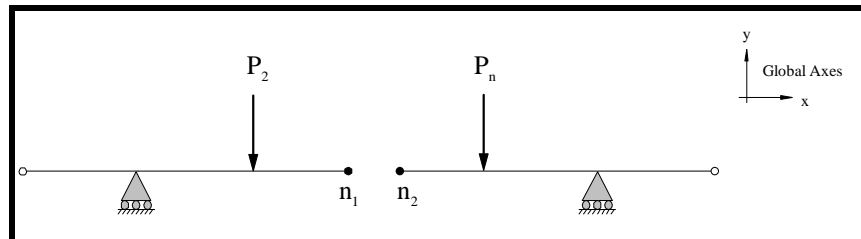
Where the ends of beam n_1 and n_2 must not translate relative to one another but must sustain a relative unit rotation. This behaviour is represented by three equations, one for each of the degrees of freedom represented by the elements chosen (in this case, say, *u, v, θ_z*)

Degree of freedom	Constraint Equation Number	Equation	Note
Translation (u)	*1*	$u_1 + u_2 = 0$	*To ensure no relative displacement in the u direction*
Translation (v)	*2*	$v_1 + v_2 = 0$	*To ensure no relative displacement in the v direction*
Rotation (θz)	*3*	$\theta_{z1} + \theta_{z2} = 1$	*To impose the required total rotation of one radian at the point. θ_{z1} and θ_{z2} are the rotations of nodes n_1 and n_2 respectively*

In general, there will be as many constraint equations as there are degrees of freedom at the point of consideration. In this instance the beam possesses three degrees of freedom and three constraint equations are required to fully describe the behaviour. For a six degree of freedom (three dimensional) beam element, the following six constraint equations would be necessary to describe the mechanism.

Degree of freedom	Constraint Equation Number	Equation	Note
Translation (u)	1	$u_1 + u_2 = 0$	To ensure no relative displacement in the u direction
Translation (v)	2	$v_1 + v_2 = 0$	To ensure no relative displacement in the v direction
Translation (w)	3	$w_1 + w_2 = 0$	To ensure no relative displacement in the w direction
Rotation (θ_x)	4	$\theta_{x1} + \theta_{x2} = 0$	To ensure no relative rotation about the θx axis θ_{x1} and θ_{x2} are the rotations of nodes n_1 and n_2 respectively
Rotation (θ_y)	5	$\theta_{y1} + \theta_{y2} = 0$	To ensure no relative rotation about the θy axis
Rotation (θ_z)	6	$\theta_{z1} + \theta_{z2} = 1$	To impose the required rotation of one radian at the point

Note that, although the physical connectivity in the mesh is maintained with constraint equations across the separation, stiffness continuity of the structure may be lost and pivot warnings/errors obtained in the output file. To circumvent these warnings/errors, both sides of the "break" should be inspected for the possibility of rigid body motion - which may be rectified by the use of additional sprung/rigid supports or the PENTLY system parameter (see the constraint equation documentation for further information).

5.3 Implementation rules

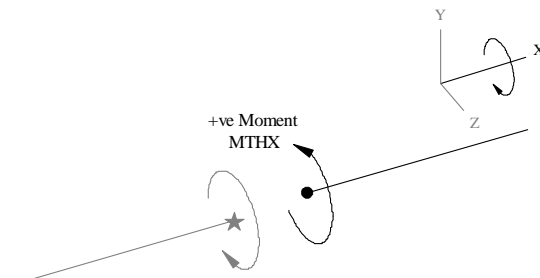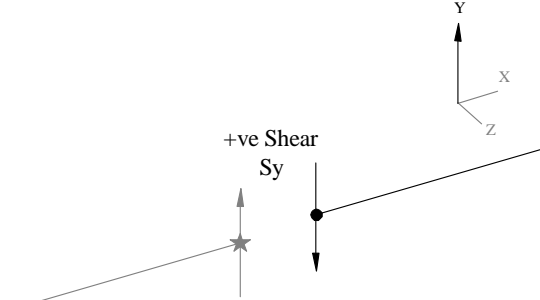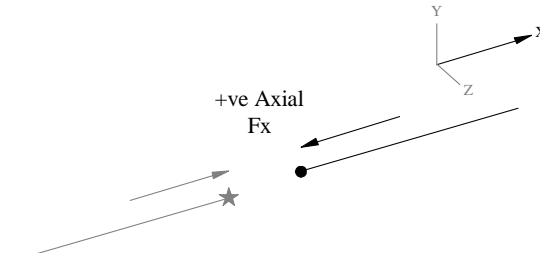When defining influence line datasets, there are three rules to be aware of:

1. The physical separation of the mesh at the influence node number specified creates two nodes at the same point - the original node, together with a new node number. The *element number* which is specified in defining the dataset is given the new node number. For multiple element connections, a new node number is generated for each connecting element that has been separated.

2. The original node on the other connecting element(s) gives the sign for influence.

3. The rotations and/or translations applied to the structure must induce an effect which is in the positive sense according to the axis system acting at the original node. That is, the imposed displacement of the original node must coincide with the positive sign of the action under consideration. For example, the bending moment influence constraint equations should induce a positive moment.

The selection of element(s) to separate is, therefore, of significance and will influence the choice of positive or negative effects. Consider the following example arrangement of new and original nodes (new node: ★, old Node: ●) in conjunction with the Element_Axis coordinates axis system specification (the Global_Axis specification would give the same results in this instance).

Diagram	*Notes*
	The positive moment convention at node (●) does not coincide with the positive axis rotation *Select the negative influence*
	The positive moment convention at node (●) coincides with the positive axis rotation *Select the positive influence*
	The positive shear convention at node (●) coincides with the positive axis translation *Select the positive influence*
	The positive axial convention at node (●) coincides with the positive axis translation *Select the positive influence*

Consider the example presented above but with alternative new and old nodes (as a result of a different element number).

Diagram	*Notes*
	The positive moment convention at node (●) coincides with the positive axis rotation *Select the positive influence*

Diagram	*Notes*
+ve Moment MTHX	*The positive moment convention at node (●) does not coincide with the positive axis rotation* *Select the negative influence*
+ve Shear Sy	*The positive shear convention at node (●) does not coincide with the positive axis translation* *Select the negative influence*
+ve Axial Fx	*The positive axial convention at node (●) does not coincide with the positive axis translation* *Select the negative influence*

## 6. Examples

6.1 Three dimensional beam

Consider the following structure:
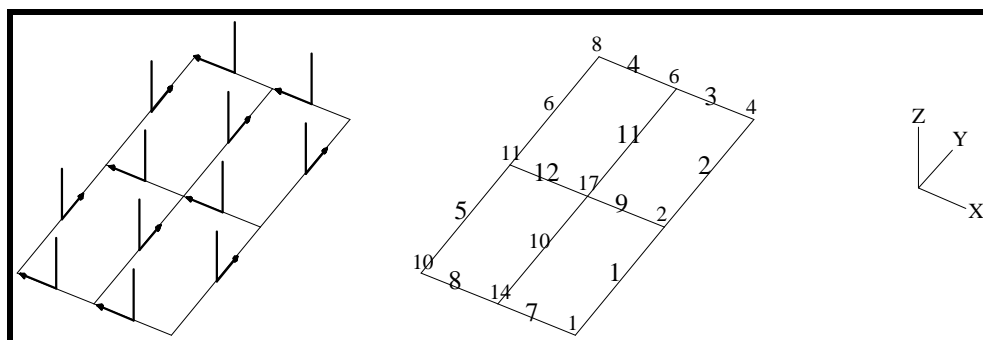
If the ***bending moment influence Mz at node 17 (with separating element number 11) is required,*** *t*here are a number of equally valid parameter specifications in the influence line dataset, as described in the follow table.

Method 1	*Specifying element 11 to separate the mesh identifies the element(s) in which the new node is to be generated and linked via constraint equations, then* 1. *An additional node (50) will be created with the same co-ordinates as Influence Node (17)* 2. *The topology of element 11 will be adjusted so that Node 50 is substituted for Node 17 (Figure 7)* 3. *Nodes 17 and 50 will be connected by constraint equations*
Method 2	*Specifying element 10 to separate the mesh introduces the new node connected to element number 10*
Method 3	*Specifying elements 9;10;12 to separate the mesh introduces the node connected to elements 9;10;12*
Method 4	*Specifying element 9;11;12 to separate the mesh introduces the node connected to elements 9;11;12*

The results of each method are illustrated as follows:
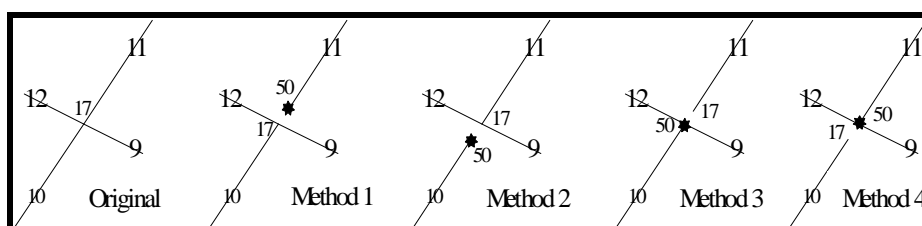
6.2 Three dimensional shell

The two cases of:

1. Bending moment influence M_x at node 17. Specified element number 4.

2. Bending *moment* influence M_y at Node 17. Specified element number 4.

will be considered for a three dimensional shell model. There are two ways to specify the separation of the mesh in each of these two cases which are shown respectively in the following tables:

Method 1	Specifying elements 2;4 to separate the mesh. This identifies the element(s) in which the new node is to be generated and linked via constraint equations, then 1. An additional node (50) will be created with the same coordinates as influence node (17) 2. The topology of element 2;4 will be adjusted so that node 50 is substituted for node 17 3. Nodes 17 and 50 will be connected by constraint equations
Method 2	Specifying elements 1;3 to separate mesh introduces the new node connected to element number 1;3

Method 1a	Specifying elements 1;2 to separate the mesh identifies the element(s) in which the new node is to be generated and linked via constraint equations, then 1. An additional node (50) will be created with the same co-ordinates as Influence Node (17) 2. The topology of element 1;2 will be adjusted so that Node 17 is substituted for Node 50 3. Nodes 17 and 50 will be connected by constraint equations
Method 2a	Specifying elements 3;4 to separate the mesh introduces the new node connected to element number 3,4

Note:Method 1 is preferred to method 2, method 1a is preferred to method 2a.

The results of each method are illustrated in the following figure:
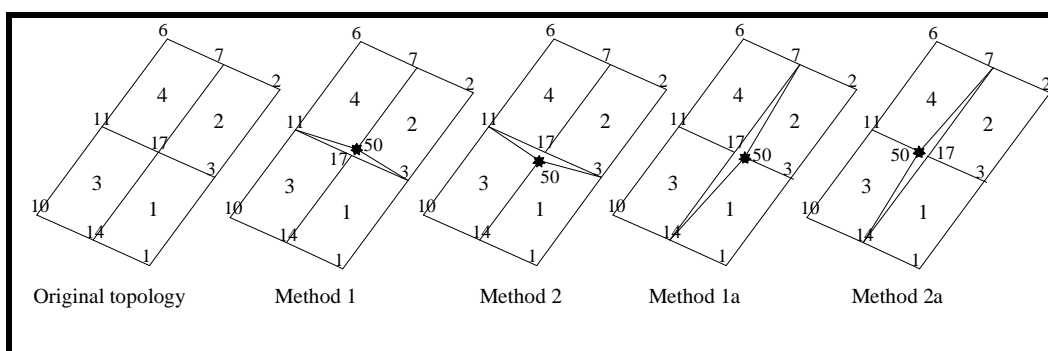
7. Final notes

- External loads applied in the model are ignored and are not tabulated in influence analysis.

- Remeshing the model may cause the node and element numbers to alter whereby all influence datasets defined to this point will be lost. It is suggested that the mesh is locked prior to any influence definition using *Meshview → Options → Lock Current Mesh*

- Do not use influence surface definitions with *Parabolic Elements* since the midside nodes are not constrained. This will lead to excessive local element distortions and incorrect results. It is suggested for use with Autoloader that the user carry out the influence dataset analysis with linear elements and bring back the loadcases onto parabolic elements

- The use of irregular meshes when specifying influence datasets may cause excessive over or under stiffening

- The influence directions should always be visualised. This indicates the positive sense of the axis system at each point and the constraints should be applied to coincide with the positive effect under examination

- Modeller allows the visualisation of the influence direction. This should be viewed to ensure that the element nodes are displacing in the correct direction