Concrete slab Design:

Yes, you can design the concrete slab using STAAD, with plate elements and meshing it appropriately.
But it is best practice to take the analysis results from the STAAD and do the manual design.
While meshing see to that your aspect ratio of the elements are close 1.0
While designing it is adviced to consider the torsional moment in addition with the major moments. **Mx = Mx + Mxy**
and similarily **My = My + Mxy for Rebar calculations**.
Shear stress shall be directly taken from the STAAD and can be checked with the allowable shear stresses based on the Pt provided.

PLATE/SHELL ELEMENT:

When a user chooses to model the surface component using plate elements, he/she is taking on the responsibility of meshing.

With the new Surface type of entity, the burden of meshing is shifted from the user to the program to some degree.

ELEMENT stress and moment output is available at the following locations:

A. Center point of the element.

B. All corner nodes of the element.

C. At any user specified point within the element.

Following are the items included in the ELEMENT STRESS output.

SQX, SQY	Shear stresses (Force/ unit len./ thk.)
SX, SY, SXY	Membrane stresses (Force/unit len./ thk)
MX, MY, MXY	Moments per unit width (Force x Length/length)
	(For Mx, the unit width is a unit distance parallel to the local Y axis.
	For My, the unit width is a unit distance parallel to the local X axis.
	Mx and My cause bending, while Mxy causes the element to twist out-of-plane.)
SMAX, SMIN	Principal stresses in the plane of the element (Force/unit area).

The 3rd principal stress is 0.0.

TMAX	Maximum 2D shear stress in the plane of the element (Force/unit area)

ANGLE Orientation of the 2D principal plane (Degrees)

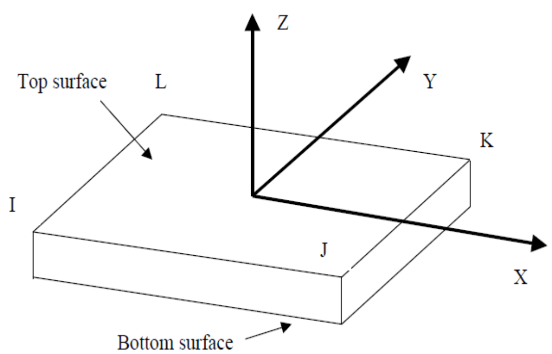
VONT, VONB 3D Von Mises stress, where

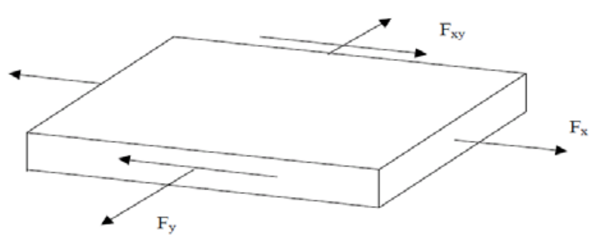
$$VM = 0.707\sqrt{(SMAX-SMIN)^2 + SMAX^2 + SMIN^2}$$

TRESCAT, TRESCAB Tresca stress, where

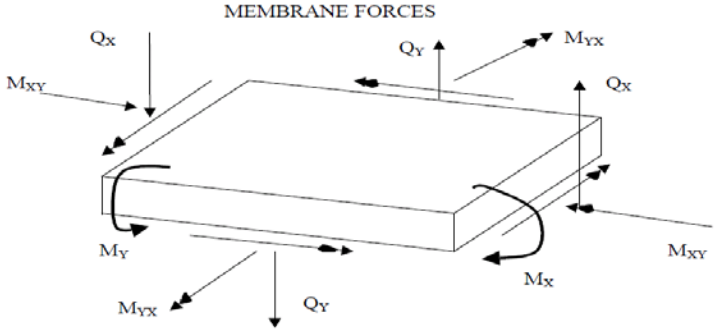
TRESCA = MAX[|(Smax-Smin)| , |(Smax)| , |(Smin)|]

The diagrams above show the positive direction of the forces relative to the following local coordinate system.





MEMBRANE FORCES



BENDING MOMENTS & TRANSVERSE SHEAR

ELEMENT FORCE outputs are available at the centre node of the element, all corner nodes of the element, and at any user-specified point within the element.
The items included in the ELEMENT FORCE output are:

QX, QY Transverse shear forces stated as force per unit length per unit element thickness.

FX, FY, FXY Membrane forces stated as force per unit length per unit element thickness.

MX, MY, MXY Bending moments stated as moment per unit length.

SMAX, SMIN Principal stresses stated as force per unit area.

TMAX Maximum in-plane shear stress stated as force per unit area.

ANGLE The orientation of the principal plane stated in degrees measured anti-clockwise from the local x-axis.
The top and bottom surfaces are identified on the basis of the direction of the local z-axis.

Units Kn met

```
10. ELEMENT PROPERTIES
11. 1 TO 9 THICK 0.25


         ELEMENT STRESSES    FORCE,LENGTH UNITS= KNS  MMS
         ----------------

                  STRESS = FORCE/UNIT WIDTH/THICK, MOMENT = FORCE-LENGTH/UNIT WIDTH

        ELEMENT  LOAD       SQX        SQY          MX          MY          MXY
                            VONT       VONB         SX          SY          SXY
                            TRESCAT    TRESCAB

            4      1        0.02      -0.02     2111.90    10726.39      4553.75
                            1.21       1.21        0.00        0.00         0.00
                            1.22       1.22
             TOP :  SMAX=     1.22  SMIN=     0.01  TMAX=     0.60  ANGLE= -23.3
             BOTT:  SMAX=    -0.01  SMIN=    -1.22  TMAX=     0.60  ANGLE= -23.3

                **** MAXIMUM STRESSES AMONG SELECTED PLATES AND CASES ****
                   MAXIMUM       MINIMUM       MAXIMUM       MAXIMUM       MAXIMUM
                  PRINCIPAL     PRINCIPAL       SHEAR        VONMISES      TRESCA
                   STRESS        STRESS        STRESS        STRESS        STRESS

                1.217974E+00 -1.217974E+00  6.017370E-01  1.210789E+00  1.217974E+00
       PLATE NO.       4             4             4             4             4
       CASE  NO.       1             1             1             1             1

         ********************END OF ELEMENT FORCES********************
```

Calculation of principal stresses for element 4

Calculations are presented for the top surface only.

SX = 0.0 kN/mm^2
SY = 0.0 kN/mm^2
SXY = 0.0 kN/mm^2
MX = 2111.84 kN-mm/mm
MY = 10726.22kN-mm/mm
MXY = 4553.97 kN-mm/mm

S = $1/6t^2 = 1/6*250^2 = 10416.67mm^2$ (Section Modulus)

$$\sigma x = SX + \frac{MX}{S} = 0.0 + \frac{2111.84}{10416.67}$$

$= 0.2027\ kN/mm^2$

$$\sigma y = SY + \frac{MY}{S} = 0.0 + \frac{10726.22}{10416.67}$$

$= 1.0297\ kN/mm^2$

$$\tau xy = SXY + \frac{MXY}{S} = 0.0 + \frac{4553.97}{10416.67}$$

$= 0.4372\ kN/mm^2$

$$TMAX = \sqrt{\frac{(\sigma_x - \sigma_y)^2}{4} + \tau_{xy}^2}$$

$$TMAX = \sqrt{\frac{(0.2027 \ -1.0297)^2}{4} + 0.4372^2}$$

$= 0.6018\ kN/mm^2$

-

$$SMAX= \frac{(\sigma_x + \sigma_y)}{2} + TMAX$$

$$= \frac{(0.2027 + 1.0297)}{2} + 0.6018$$

$= 1.218\ kN/mm^2$ say $1.22\ kN/mm^2$

$$SMIN= \frac{(\sigma_x + \sigma_y)}{2} - TMAX$$

$$= \frac{(0.2027 + 1.0297)}{2} - 0.6018$$

$= 0.0144\ kN/mm^2$ say $0.01\ kN/\ mm^2$

$$Angle = \frac{1}{2} \tan^{-1} \left\{ \frac{2\tau_{xy}}{\sigma_x - \sigma_y} \right\}$$

$$= \frac{1}{2} \tan^{-1} \left\{ \frac{2 * 0.4372}{0.2027 - 1.0297} \right\}$$

$= -23.3°$

When I perform concrete design on an element, the output contains expressions such as "LONG. REINF.", "TRANS. REINF.", "TOP", "BOTT.", etc. Can you explain what these terms mean?

The design of an element involves determination of the reinforcement for moments Mx and My at the centroid of the element. The reinforcement calculated to resist Mx is called longitudinal reinforcement, and is denoted in the output by the expression "LONG. REINF.".

The reinforcement calculated to resist My is called transverse reinforcement, and is denoted in the output by the expression "TRANS. REINF.".

The sign of Mx and My will determine which face of the element the steel has to be provided on. Every element has a "top" face, and a "bottom" face, as defined by the direction of the local Z axis of the elements. Mx will cause tension on one of those faces, and compression on the other. A similar effect will be caused by My. The output report of reinforcement provided on those faces contains the terms "TOP" for top face, and "BOTT" for the bottom face.

The procedure used by the program to arrive at these quantities is as follows :

For each element, the program first scans through all the active load cases, to find the following maxima :

Maximum positive Mx
Maximum negative Mx
Maximum positive My
Maximum negative My

The element is then designed for all those four quantities. If any of these moments happen to be zero, or if the reinforcement required to resist that moment is less than the capacity of the element with minimum reinforcement, only minimum reinforcement is provided. For the ACI code, the rules governing provision of reinforcement for shrinkage and temperature are used in calculating minimum reinforcement.

The rules applicable for design of a beam for flexure are used in calculating the steel areas. The width used in this calculation is a unit width of the element. For determination of the effective depth, the steel for longitudinal moment is assumed to be the outer layer, and the steel for transverse moment is the inner layer.

The output will consist of the steel area required for all of four maximas. As described earlier, they will be reported using the terms LONG, TRANSVERSE, TOP and BOTT.

When I perform concrete design on an element, the output reports reinforcement in terms of "SQ.MM/MM". Can you please explain why?

When you ask for an element design or a slab design using the commands

DESIGN ELEMENT ..

or

DESIGN SLAB ..

STAAD designs the element for the moments MX and MY at the centroid of the element. By definition, MX and MY are termed as Moments per Unit width, since that is what they are. They have units of Force-length/length, as in 43.5 KN-mm/mm, or 43.5 KN-m/m. In other words, if you take a one metre width of the slab at the centroid of the element in question, the moment over that one metre width on that element is equal to 43.5 KN-m.

The design of that element hence has to be done on the basis of a unit width. Thus, in order to design an element for a 43.5 KN-m/m moment, one needs to use a one metre width of slab. The reinforcement required for that element is thus reported in terms of unit width of the element. The results are hence in the form Area of steel/unit-width of element, as in, "SQ.MM/MM".

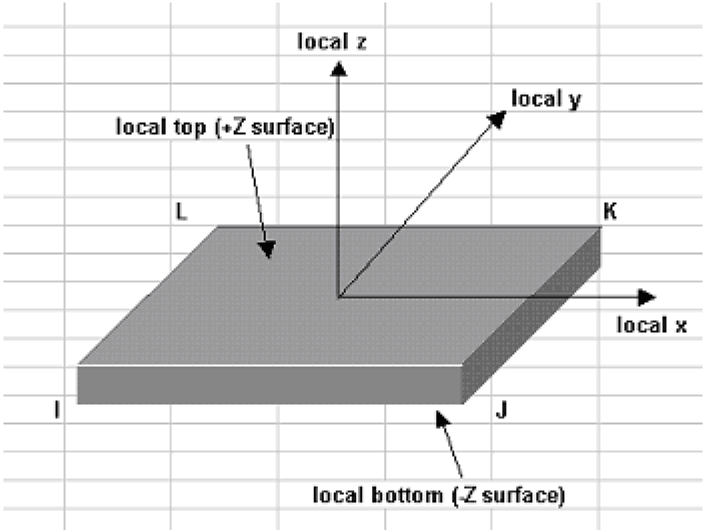
A floor slab has been modeled using 4-noded plate elements. The elements are subjected to pressure loading in the vertically downward direction. A concrete design has been performed on the elements. (See below for the reinforcement report for many of those elements.)
Why is it that the moments as well as reinforcement are appearing on the top and not on the bottom of the plates?

The reinforcement report for many of those elements looks like the following:

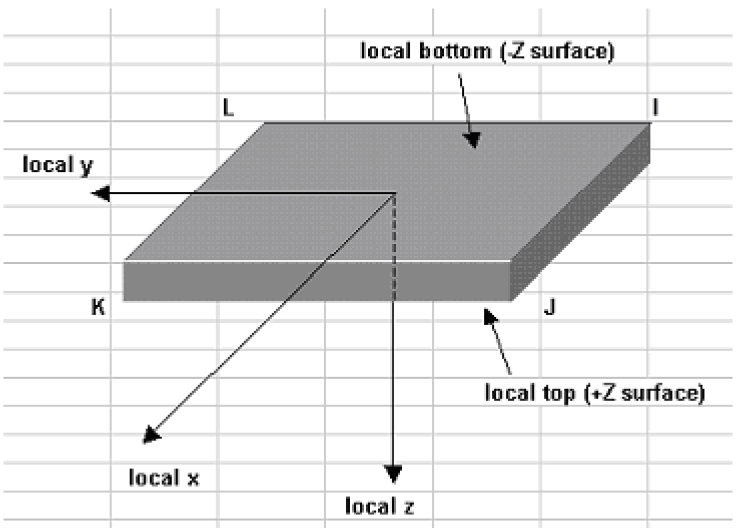
ELEMENT	LONG. REINF (SQ.IN/FT)	MOM-X /LOAD (K-FT/FT)	TRANS. REINF (SQ.IN/FT)	MOM-Y /LOAD (K-FT/FT)
134 TOP :	5.944	1474.13 / 12	6.914	1679.58 / 12
BOTT:	1.296	0.00 / 0	1.296	0.00 / 0

Solution:

In the above output, the word TOP and BOTTOM refer to the "local" top and bottom surfaces of the individual elements, and not in the global axis sense. The local top and bottom surfaces depend on the way an element is defined in its incidence statement.
TOP is defined as the surface which coincides with the positive side of the local Z axis. BOTTOM is defined as the surface which coincides with the negative side of the local Z axis.
Shown below are two examples in which the element incidence is numbered in two contrasting ways.
In the first figure, the local Z axis of the element points in the vertically upward direction. Consequently, the local top and bottom surfaces have the same sense as the global top and bottom.



In the next figure, the local Z axis of the element points in the vertically downward direction. Consequently, the local top and bottom surfaces have the opposite sense as the global top and bottom.



You can verify the direction of the local axes of the elements in your model by doing the following. Click the right mouse button and select Labels. Under the Plate category, switch on Plate Orientation. The local axes will be displayed as shown in these figures above.

PLATE AND SURFACE:

What is difference between plate element and Surface?

I am use to prepare model without including slab and assigning load on each beams by manual calculation of loads coming on each beams due to slab as per one way and two slab distribution rules. Now I want to create model with slabs by using floor area facility. Please advise what I have to do ?

1.use add surface button and surface cursor.

2.OR use Generate surface meshing and palate cursor.

Secondly, if some of my RC members have failed then what I have to do other than change size of failed member and re-analyzed and design whole structure.

3. If my all RC beams and columns has been passed but slab has been failed then what I have to do ?

4. In design of slab, is it necessary to convert 1- region into two or more regions. What is requirement?

If you want to **consider the in plane rigidity offered** by the slabs in the model then it is best to **model it as plates**.

Surfaces are special entities that need to be used for the **modeling of shear walls** and thereafter these shear walls need to be designed as per the concrete codes (ACI if you are using American code). So if you are modeling plates then the load can be provided as a pressure load on plates. But make sure that the plates are meshed properly so that there is proper load transfer from the plates to the adjacent beam members. If you don't want to consider the in plane rigidity offered by the slab against lateral displacements then the loading can be done using floor load options.

You can either **model the walls using plates** or you may manually calculate the load and provide them as a uniformly distributed member load.

A good rule of thumb for starting mesh sizes is the lesser of span/10 or 1000mm.

ELEMENT FORCE outputs are available at the centre node of the element, all corner nodes of the element, and at any user-specified point within the element.

The items included in the ELEMENT FORCE output are:

QX, QY Transverse shear forces stated as force per unit length per unit element thickness.

FX, FY, FXY Membrane forces stated as force per unit length per unit element thickness.

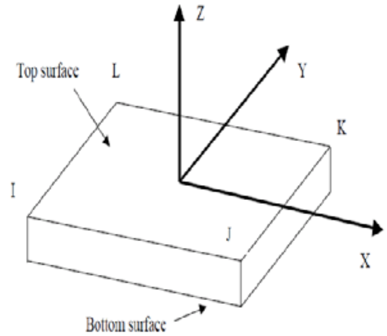
MX, MY, MXY Bending moments stated as moment per unit length.

SMAX, SMIN Principal stresses stated as force per unit area.

TMAX Maximum in-plane shear stress stated as force per unit area.

ANGLE The orientation of the principal plane stated in degrees measured anti-clockwise from the local x-axis.

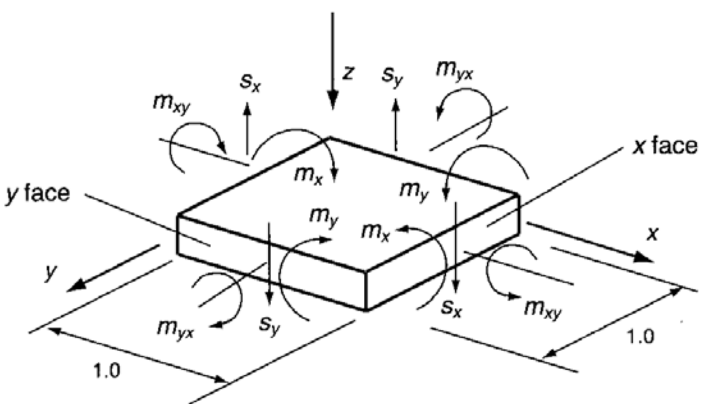
The top and bottom surfaces are identified on the basis of the direction of the local z-axis.



Output stresses and moments

Finite element programs normally provide output of stresses, internal moments and shears for plate bending elements.

The traditional notation (Timoshenko and Woinowsky-Krieger 1964) for internal moments and shears in plates is as shown in Fig.



Notation for plate bending.

Note the following features.

- All moments and shears are defined per unit width of plate.
- An 'x face' is defined as the face which is normal to the x axis. The 'positive x face' is the one from which the x axis points outwards. The notation for the internal moments are defined in relation to the faces on which they act rather than in relation to the axis about which they rotate.
- m_x and m_y are the direct moments per unit width, and are obtained by integrating the moments of the forces due to the σ_x and σ_y direct stresses, respectively. They are defined as sagging positive (in this case).
- m_{xy} and m_{yx} are the twisting moments per unit width, and are obtained by integrating the shear forces due to τ_{xy} and τ_{yx} stresses, respectively (note that, for example, τ_{xy} is the shear stress on the x plane in the y direction). The twisting moments are characterised as m_{xy} since $|m_{xy}| = |m_{yx}|$.
- m_x, m_y and m_{xy} form a 'triad' which is used to calculate principal moments, etc.
- s_x and s_y are the transverse shear forces per unit width on the x and y faces, respectively. They are the integrals of the τ_{xz} and τ_{yz} stresses, respectively.

Relationships between the internal force actions and stresses for a flat plate are given in Table *Stress components for flat plate bending*

Stress component	Maximum value	Position of maximum within the depth of the plate	Shape of stress block
Direct stress	$\sigma_x = 6m_x/t^2$ $\sigma_y = 6m_y/t^2$	Top and bottom	Linear
Shear stress due to twisting moments	$\tau_{xy} = 6m_{xy}/t^2$	Top and bottom	Linear
Shear stress due to transverse shear	$\tau_{xz} = 1.5s_x/t$	Centre	Parabolic

Notes: t – plate thickness.

Ref:Modern structural Analysis by Iain MacLeod.

Technical Reference

1.6.1 Plate and Shell Element

The Plate/Shell finite element is based on the hybrid element formulation. The element can be 3-noded (triangular) or 4-noded (quadrilateral). If all the four nodes of a quadrilateral element do not lie on one plane, it is advisable to model them as triangular elements. The thickness of the element may be different from one node to another.

"Surface structures" such as walls, slabs, plates and shells may be modeled using finite elements. For convenience in generation of a finer mesh of plate/shell elements within a large area, a MESH GENERATION facility is available. The facility is described in detail in Section 5.14.

The user may also use the element for PLANE STRESS action only. The ELEMENT PLANE STRESS command should be used for this purpose.

Geometry Modeling Considerations

The following geometry related modeling rules should be remembered while using the plate/shell element.

1) The program automatically generates a fictitious fifth node "O" (center node - see Fig. 1.8) at the element center.

2) While assigning nodes to an element in the input data, it is essential that the nodes be specified either clockwise or counter clockwise (Fig. 1.9). For better efficiency, similar elements should be numbered sequentially.

3) Element aspect ratio should not be excessive. They should be on the order of 1:1, and preferably less than 4:1.

4) Individual elements should not be distorted. Angles between two adjacent element sides should not be much larger than 90 and never larger than 180.

Load Specification for Plate Elements

Following load specifications are available:

1) Joint loads at element nodes in global directions.

2) Concentrated loads at any user specified point within the element in global or local directions.

3) Uniform pressure on element surface in global or local directions

4) Partial uniform pressure on user specified portion of element surface in global or local directions

5) Linearly varying pressure on element surface in local directions.

6) Temperature load due to uniform increase or decrease of temperature.

7) Temperature load due to difference in temperature between top and bottom surfaces of the element.

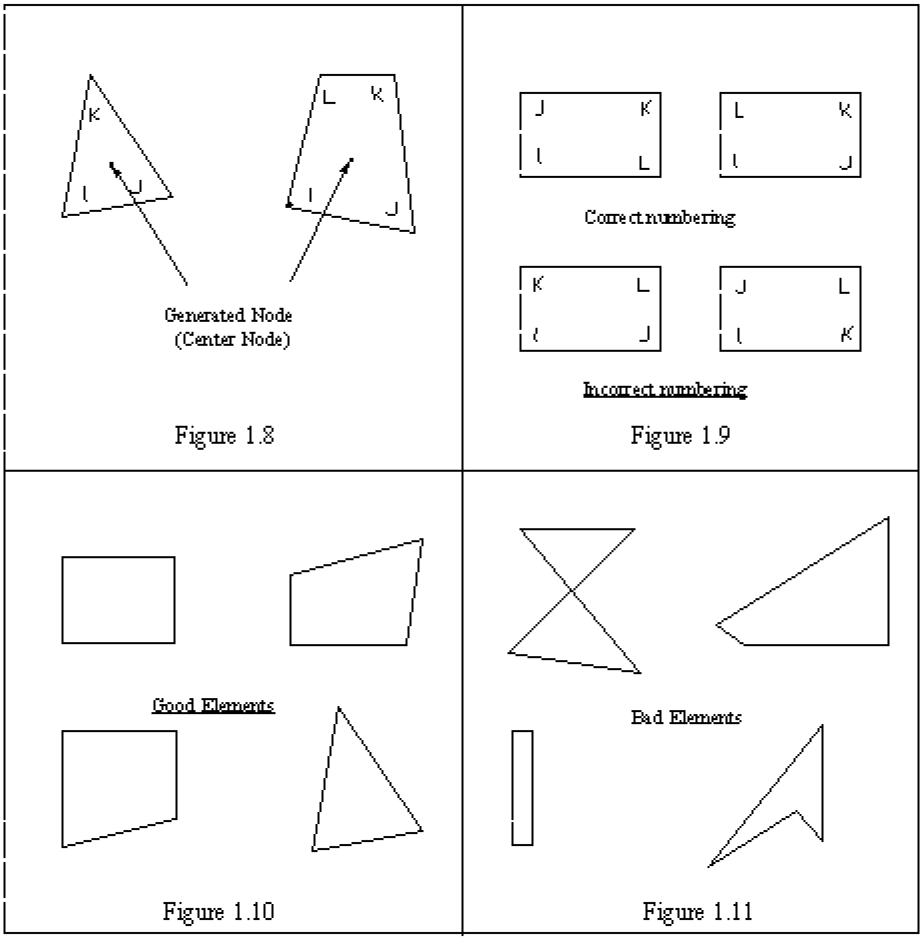


Figure 1.13

Theoretical Basis

The STAAD plate finite element is based on hybrid finite element formulations. A complete quadratic stress distribution is assumed. For plane stress action, the assumed stress distribution is as follows.

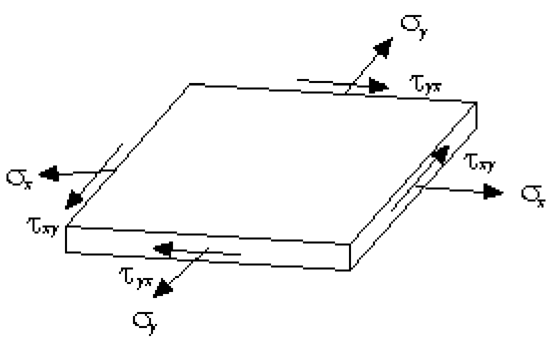


Figure 1.14

Complete quadratic assumed stress distribution:

$$\begin{Bmatrix} \sigma_x \\ \sigma_y \\ \tau_{xy} \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 2xy \\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & -x^2 \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ \vdots \\ a_{10} \end{Bmatrix}$$

a_1 through a_{10} = constants of stress polynomials.

The following quadratic stress distribution is assumed for plate bending action:

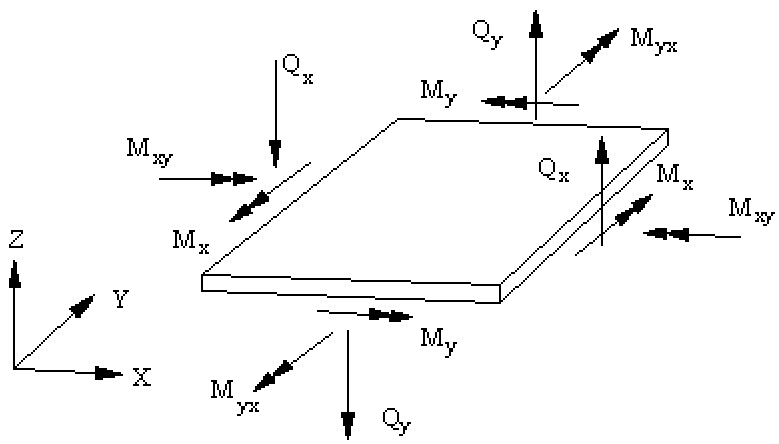


Figure 1.15

Complete quadratic assumed stress distribution:

$$\begin{Bmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{Bmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & xy & y^2 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & -xy \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & -x \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & x & y \end{bmatrix} \begin{Bmatrix} a_1 \\ a_2 \\ a_3 \\ \vdots \\ \vdots \\ a_{13} \end{Bmatrix}$$

a_1 through a_{13} = constants of stress polynomials.

The distinguishing features of this finite element are:

1) Displacement compatibility between the plane stress component of one element and the plate bending component of an adjacent element which is at an angle to the first (see Fig. below) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/plate elements.

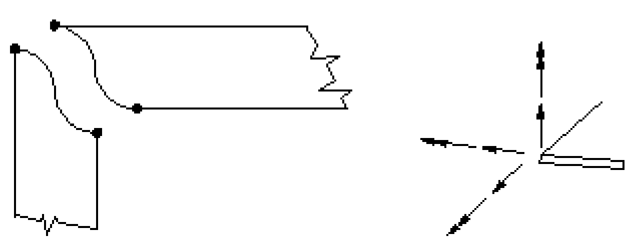


Figure 1.16

2) The out of plane rotational stiffness from the plane stress portion of each element is usefully incorporated and not treated as a dummy as is usually done in most commonly available commercial software.

3) Despite the incorporation of the rotational stiffness mentioned previously, the elements satisfy the patch test absolutely.

4) These elements are available as triangles and quadrilaterals, with corner nodes only, with each node having six degrees of freedom.

5) These elements are the simplest forms of flat shell/plate elements possible with corner nodes only and six degrees of freedom per node. Yet solutions to sample problems converge rapidly to accurate answers even with a large mesh size.

6) These elements may be connected to plane/space frame members with full displacement compatibility. No additional restraints/releases are required.

7) Out of plane shear strain energy is incorporated in the formulation of the plate bending component. As a result, the elements respond to Poisson boundary conditions which are considered to be more accurate than the customary Kirchoff boundary conditions

8) The plate bending portion can handle thick and thin plates, thus extending the usefulness of the plate elements into a multiplicity of problems. In addition, the thickness of the plate is taken into consideration in calculating the out of plane shear.

9) The plane stress triangle behaves almost on par with the well known linear stress triangle. The triangles of most similar flat shell elements incorporate the constant stress triangle which has very slow rates of convergence. Thus the triangular shell element is very useful in problems with double curvature where the quadrilateral element may not be suitable.

10) Stress retrieval at nodes and at any point within the element.

Plate Element Local Coordinate System

The orientation of local coordinates is determined as follows:

1) The vector pointing from I to J is defined to be parallel to the local x- axis.

2) The cross-product of vectors IJ and IK defines a vector parallel to the local z-axis, i.e., z = IJ x IK.

3) The cross-product of vectors z and x defines a vector parallel to the local y- axis, i.e., y = z x x.

4) The origin of the axes is at the center (average) of the 4 joint locations (3 joint locations for a triangle).

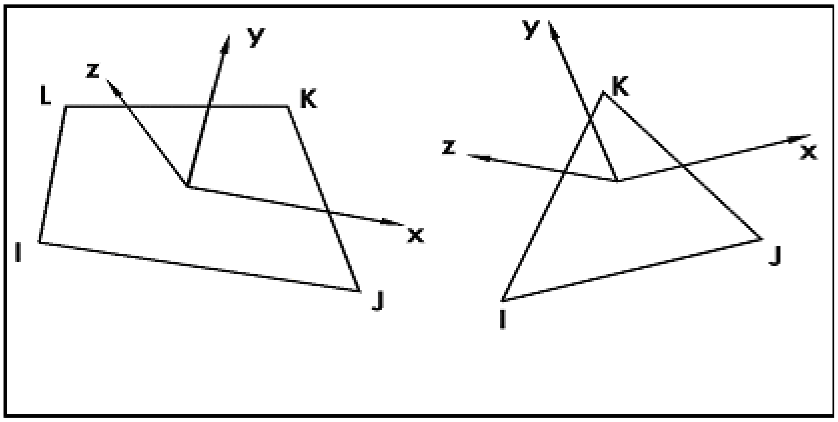


Fig. 1.17

Output of Plate Element Stresses and Moments

For the sign convention of output stress and moments, please see Fig. 1.13.

ELEMENT stress and moment output is available at the following locations:

A. Center point of the element.
B. All corner nodes of the element.
C. At any user specified point within the element.

Following are the items included in the ELEMENT STRESS output.

SQX, SQY	Shear stresses (Force/ unit len./ thk.)
SX, SY, SXY	Membrane stresses (Force/unit len./ thk)
MX, MY, MXY	Moments per unit width (Force x Length/length) (For Mx, the unit width is a unit distance parallel to the local Y axis. For My, the unit width is a unit distance parallel to the local X axis. Mx and My cause bending, while Mxy causes the element to twist out-of-plane.)
SMAX, SMIN	Principal stresses in the plane of the element (Force/unit area). The 3rd principal stress is 0.0.
TMAX	Maximum 2D shear stress in the plane of the element (Force/unit area)
ANGLE	Orientation of the 2D principal plane (Degrees)
VONT, VONB 3D	Von Mises stress, where

$$VM = 0.707\sqrt{(SMAX - SMIN)^2 + SMAX^2 + SMIN^2}$$

TRESCAT, TRESCAB Tresca stress, where
TRESCA = MAX[|(Smax-Smin)| , |(Smax)| , |(Smin)|]

Notes:

1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in Fig. 1.13.
2. To obtain element stresses at a specified point within the element, the user must provide the location (local X, local Y) in the coordinate system for the element. The origin of the local coordinate system coincides with the center of the element.
3. The 2 nonzero Principal stresses at the surface (SMAX & SMIN), the maximum 2D shear stress (TMAX), the 2D orientation of the principal plane (ANGLE), the 3D Von Mises stress (VONT & VONB), and the

3D Tresca stress (TRESCAT & TRESCAB) are also printed for the top and bottom surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.

4. The third principal stress is assumed to be zero at the surfaces for use in Von Mises and Tresca stress calculations. However, the TMAX and ANGLE are based only on the 2D inplane stresses (SMAX & SMIN) at the surface. The 3D maximum shear stress at the surface is not calculated but would be equal to the 3D Tresca stress divided by 2.0.

Sign Convention of Plate Element Stresses and Moments

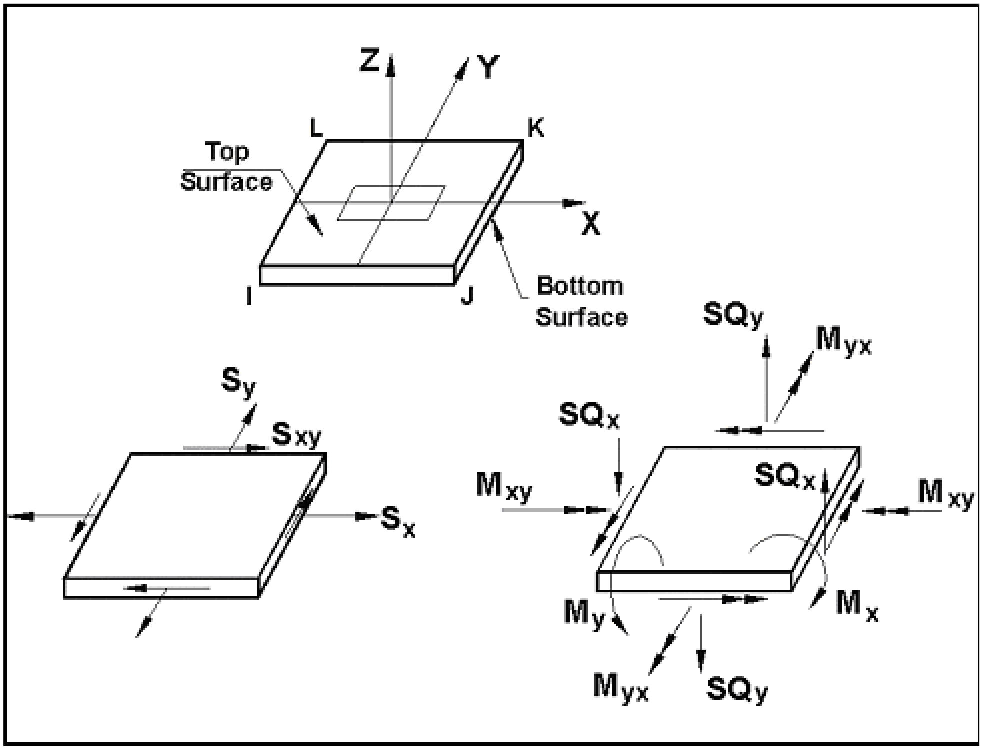


Figure 1.18

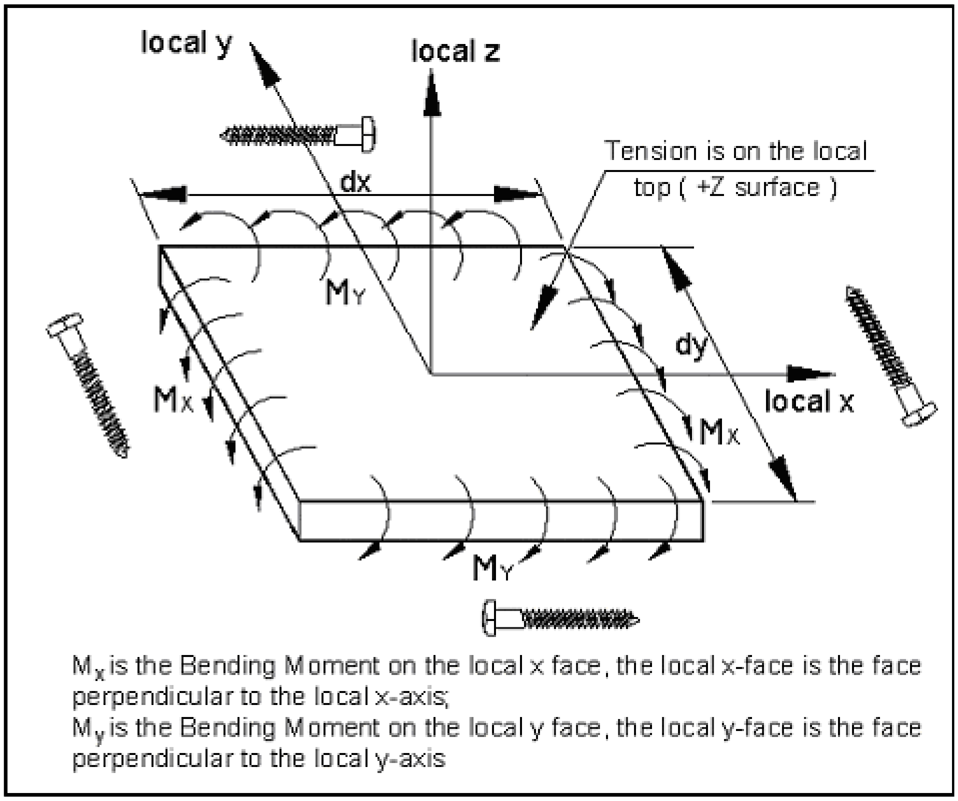


Figure 1.19

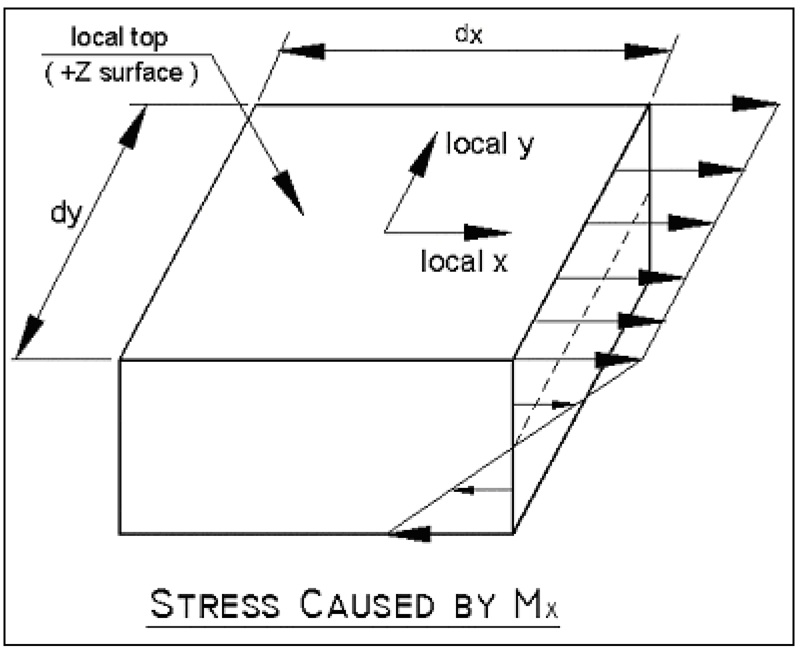


Figure 1.20

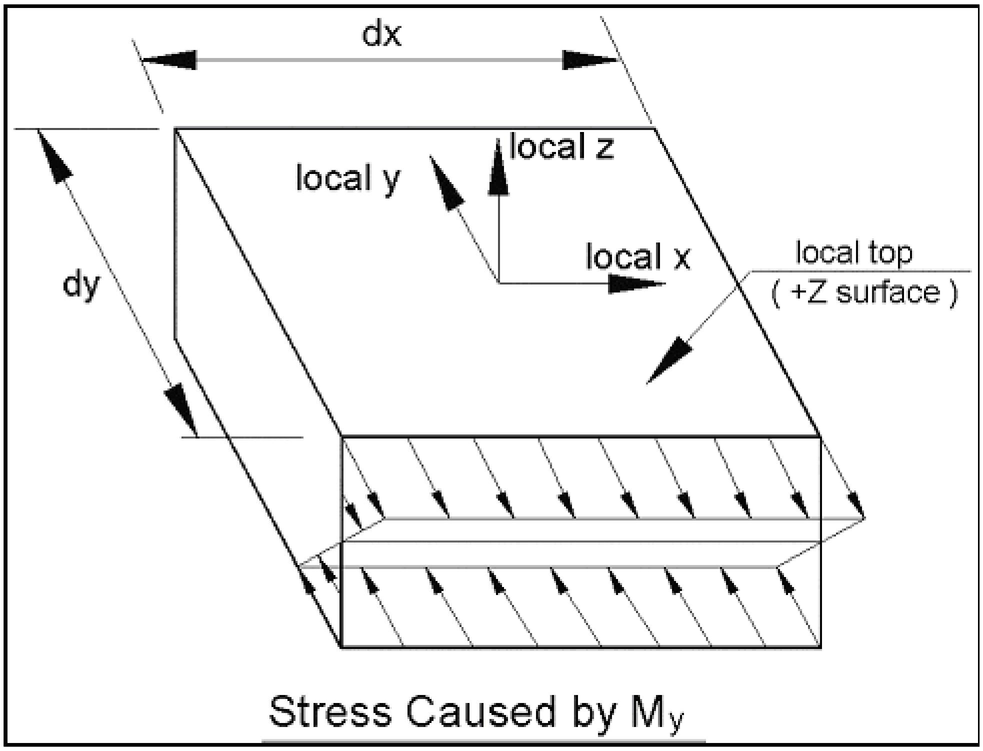


Figure 1.21

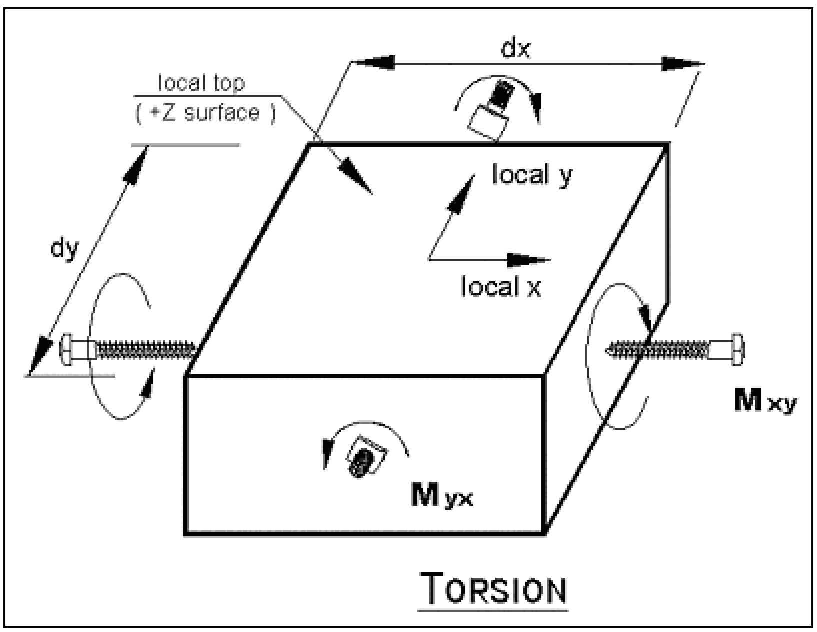


Figure 1.22

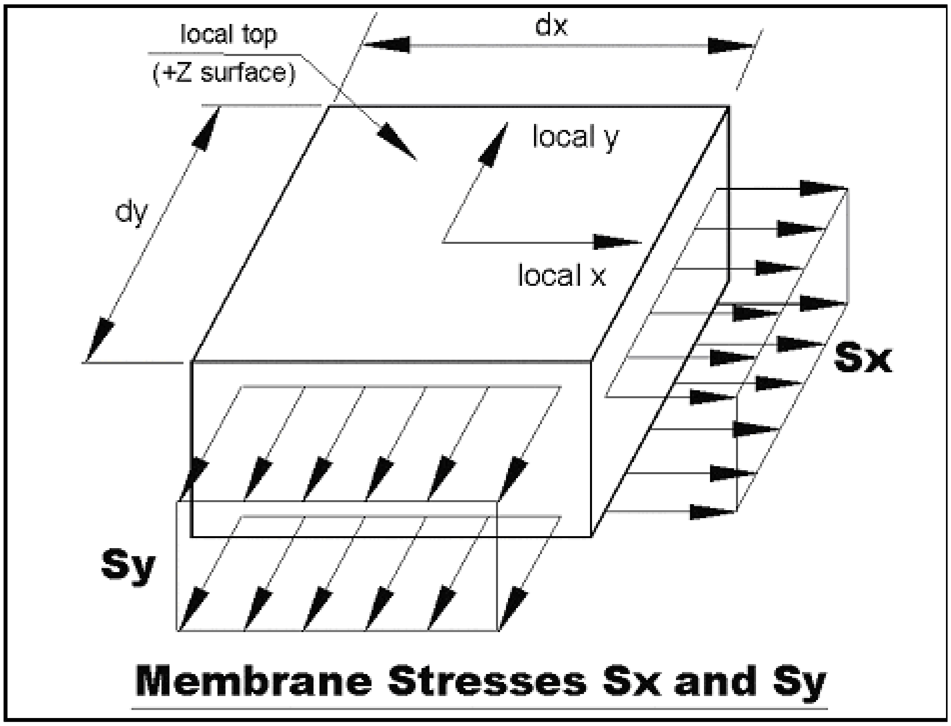


Figure 1.23

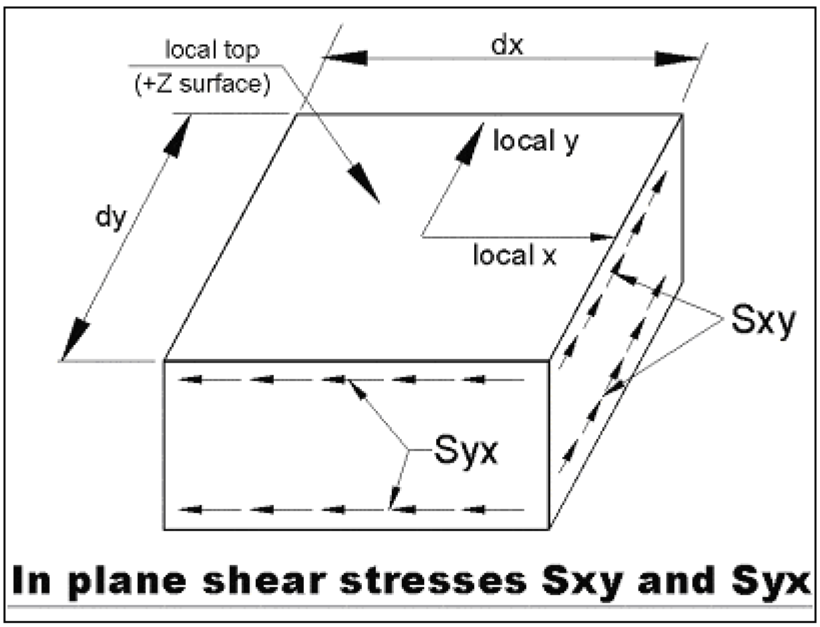


Figure 1.24

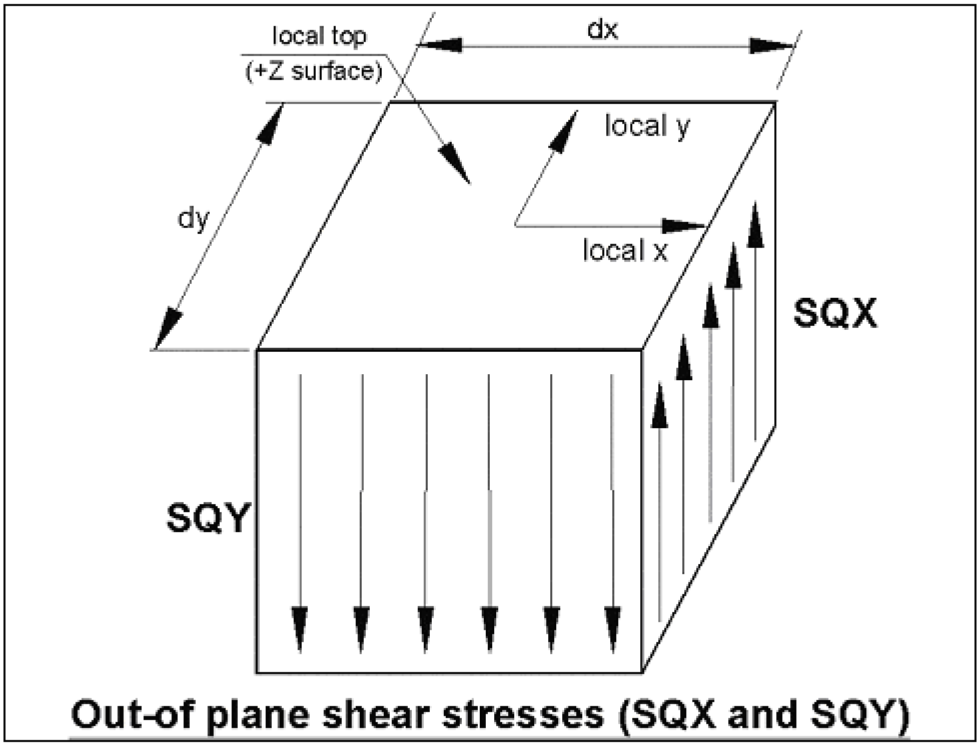


Figure 1.25

Please note the following few restrictions in using the finite element portion of STAAD:

1) Both frame members and finite elements can be used together in a STAAD analysis. The ELEMENT INCIDENCES command must directly follow the MEMBER INCIDENCES input.

2) The selfweight of the finite elements is converted to joint loads at the connected nodes and is not used as an element pressure load.

3) Element stresses are printed at the centroid and joints, but not along any edge.

4) In addition to the stresses shown in Fig 1.18, the Von Mises stresses at the top and bottom surface of the element are also printed.

Plate Element Numbering

During the generation of element stiffness matrix, the program verifies whether the element is same as the previous one or not. If it is same, repetitive calculations are not performed. The sequence in which the element stiffness matrix is generated is the same as the sequence in which elements are input in element incidences.

Therefore, to save some computing time, similar elements should be numbered sequentially. Fig. 1.14 shows examples of efficient and non-efficient element numbering.

However the user has to decide between adopting a numbering system which reduces the computation time versus a numbering system which increases the ease of defining the structure geometry.

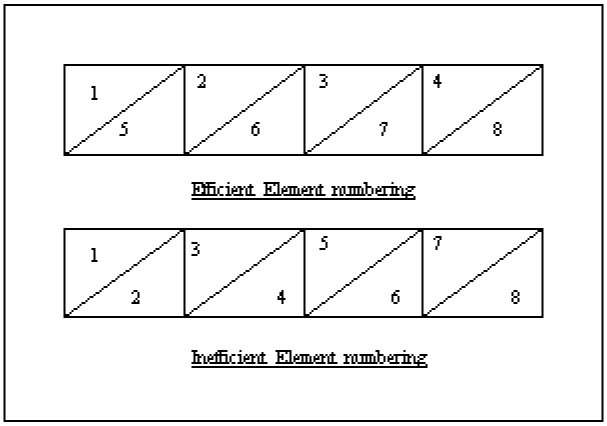

1
5
2
6
3
7
4
8
Efficient Element numbering
1
2
3
4
5
6
7
8
Inefficient Element numbering

Figure 1.26