Republic of Iraq
Ministry of Higher Education and Scientific Re
University of Technology
Building and Construction Engineering
Department

Analysis of Rectangular Concrete Tanks Using STAAD Pro Program

Scientific Project Submitted to the Department Of
(Building and construction)
University of Technology

In Partail Fulfillment of The Requirement
For The Degree of Bachlor
of Scienc In WaterResources and Dams Engineering

Prepared by :
Thoalfaqar Salim Hussein
Amenna Hamed Alwan
Supervisor by :
Dr.Ammar A. Ali

بسـم الله الرحمن الرحيم

"قالوا سبحانك لا علم لنا إلا ما علمتنا انك أنت العليم الحكيم"

صدق الله العظيم

عن الإمام علي عليه السلام

"اعمل لدنياك كأنك تعيش أبدا واعمل لآخرتك كأنك تموت غدا"

Contents

Introduction

1-1 General

Water storage tank are very important public structures. These tanks may be used for storage of water for drinking and washing, domestic use, swimming pools, sewage sedimentation …etc.

Water retaining structures commonly known as "tanks" are classified according to it is position as follows:-

1- Tanks resting on ground.
2- Tanks constructed underground.
3- Elevated tanks.

All the above three types of tanks may be of circular, rectangular, hexagonal or any other shape in plane. These tanks may be with or without a roof covering.

The side wall of tanks are subjected to water pressure from water in the case of tanks resting on ground while they may also subjected to earth pressure (saturated or unsaturated) from outside in case of underground tanks.

While designing underground tanks, both conditions of full of water or empty have to be considered. In case of tanks full, water pressure of side walls has to be considered assuming there is no soil outside of the wall. However, in case when tank is empty, soil pressure is to be considered from outside and also the stability of tanks against up lift pressure due to ground water has to be maintained.

Elevated tanks are supported on staging which may consist of solid or perforated masonry walls or reinforced concrete columns braced together or a thin hollow shaft.

The tanks portion is designed for water pressure, live load and self weight of different parts, while the staging is design to resist wind forces and earthquake forces. In addition, slab in such cases is generally provided as raft or piles depending upon soil conditions. Leakage or seepage is a common problem in water retaining structures, to minimize it, impervious concrete must be used. Also we have to consider a very important three factors in designing a reinforced concrete tanks, which are they :

1- Strength
2- Water tightness
3- Overall stability

It is common practice to use reinforced or pre-stressed concrete structure for the retention exclusion or storage of water and other aqueous liquids. Concrete is generally the most economical material of construction (the word concrete has come to symbolize strength, stability and the image of being set as stone) and when correctly designed and constructed will provide long life and low maintenance costs so not surprising the concrete is the most common solution for storage tanks in the water and waste water industry. Beyond the inherent durability of concrete and the variety of post-tension and corrosion protection techniques are providing even longer- lasting and economical solution for storage tanks.

The following figures are simple drawing for the three general types of tanks mentioned above .

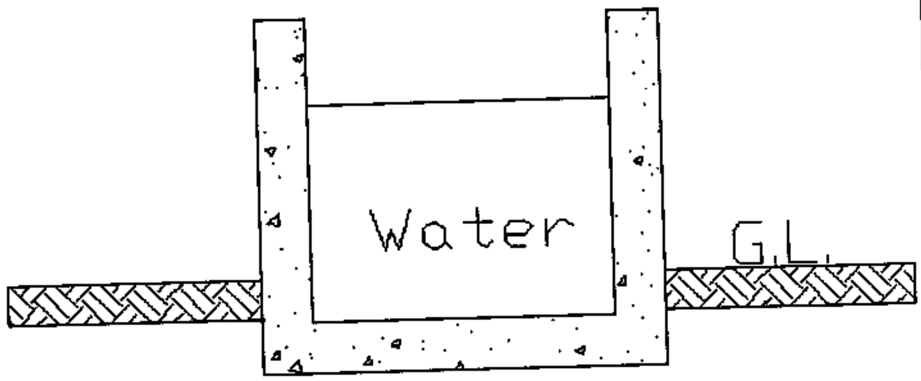


Fig. (1-1) Ground Tank

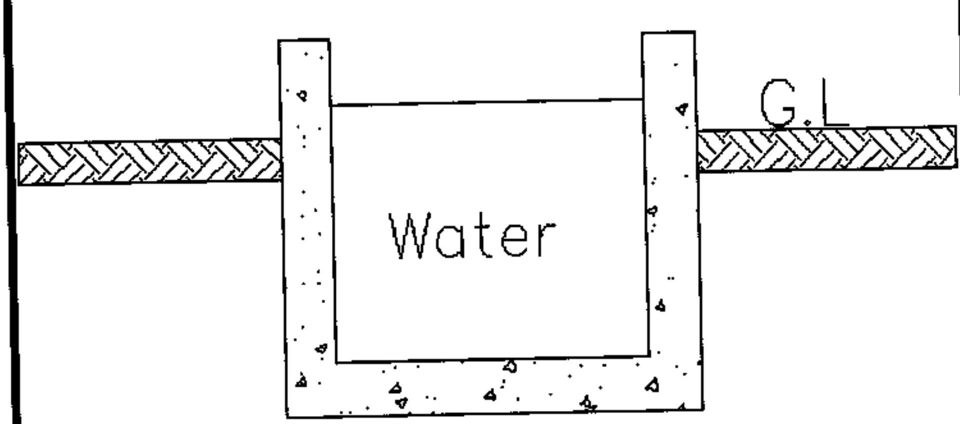


Fig. (1-2) Under Ground Tank

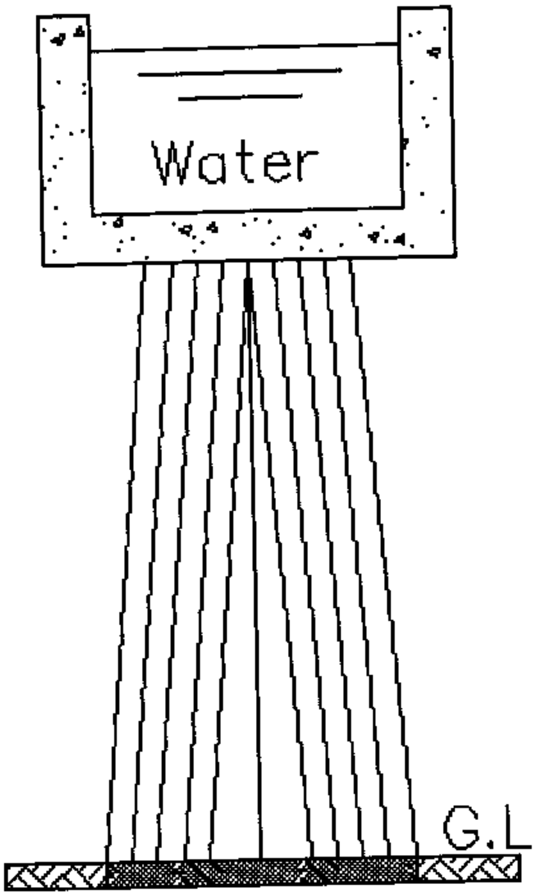


Fig. (1-3) Elevated Tank

On the other hand, there is the Spherical shaped storage tanks (Fig.1-4) which are generally used for storing products at pressures above 35 kPa (ga).

A spheroidal tank (Fig.1-5) is essentially spherical in shape except that it is somewhat flattened. Hemispheroidal tanks have cylindrical shells with curved roofs and bottoms. Noded spheroidal tanks are generally used in the larger sizes and have internal ties and supports to keep shell stresses low. These tanks are generally used for storing products above 35 kPa.

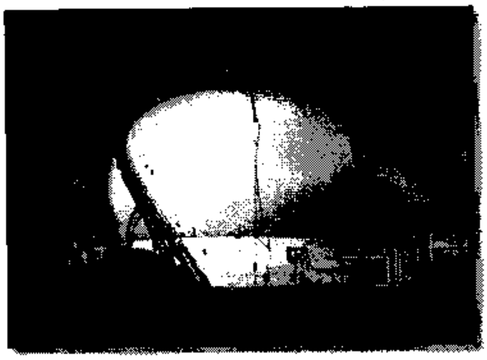


Fig.(1-4) Spherical tank

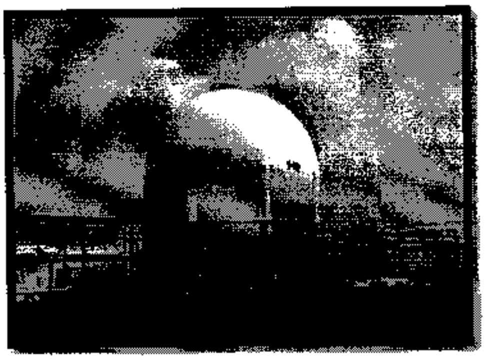


Fig.(1-5) Spheroidal tank

Also Underground storage is most advantageous when large volumes are to be stored. Underground storage is especially advantageous for high vapor pressure products.

Types of underground storage are:

1- Caverns constructed in salt by solution mining or conventional mining.

2- Caverns constructed in nonporous rock by conventional mining.

3- Caverns developed by conversion of depleted coal, limestone, or salt mines to storage e.

1-2 Features

1-Rectangular shape makes best use of available space

2- May be installed at or below ground level or on towers

3-Clear interior allows easy inspection, draining and painting

1-3 Aims of The Study

The aim of this project is to study the structure behavior of ground water storage tanks . The project discusses the details of the ground water tank. Also we have presets a simple example to design small a rectangular water tank with its details. By using STAAD.Pro program we aim to design a rectangular tank by (finite element method) and study the results obtained.

Problem Development

2-1 Design Methods of Concrete Tanks

many methods are available to deal with designing of rectangular water tanks. But these methods are different with how much they are simple or how accurate the results which are ordained. Some of these methods are given below.

2-1-1 Using Charts and Tables

Tanks have traditionally been designed by reference to published tables derived from elastic thin plate theory.

These tables cover isolated rectangular panels with various proportions, and edge conditions and loading as shown in Fig.(2-1). Interpolation is required and the values cannot represent the real interaction between adjacent walls and the base. The base rarely provides the assumed fixity and it is not easy to calculate these affects accurately .

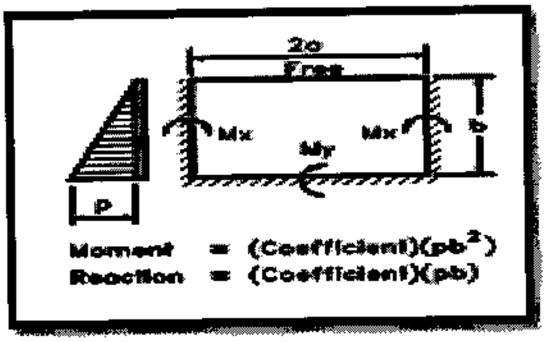


Fig.(2-1) Rectangular wall of a tank with fixed edges

Engineers now have access to powerful computers and suitable software which should enable them to design tanks more accurately as complete structures and show worthwhile savings. This has not become a routine method however because the time spent generating the structure and loadings and dealing with the output can be such that

many tanks can still be designed economically and quicker by hand.

It is pointless to aim for the extreme accuracy of a "Perfect Analysis" for a concrete structure when the variability of factors such as ground conditions and panel thickness can affect the results by as much as 20% in some cases. The aim, therefore, is to develop a practical and quick theoretical analysis procedure that can give results to between 5% and 10% of a "Perfect Analysis".

The following methods can meet these criteria and show that a balance can be struck between accuracy and practicality.

2-1-2 Grillage Method

This method subdivides each wall and slab panel of the tank into a grillage of rectangular beams which are connected at the panel junctions Fig.(2-2). The beams are given properties relating to their orientation, spacing and the panel thickness. The support conditions are applied at the base panel nodes as springs which can be specified to suit piles or the stiffness of the ground. The loads are applied onto the beams.

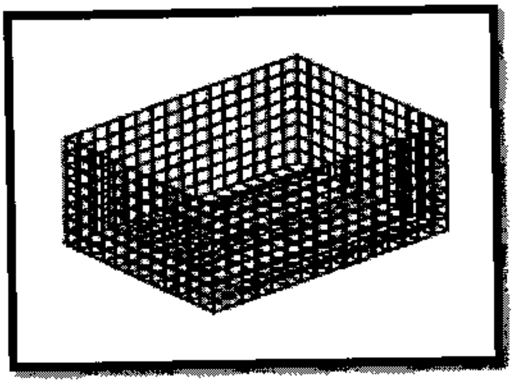


Fig.(2-2) Grillage modeling of rectangular tank

The above model produces acceptably smooth moment diagrams and the results have an error of less than 4%. They can be plotted as

shown in the cut-away view Fig.(2-3) or against sections through the structure. The values must be divided by the beam width to produce values per meter width.

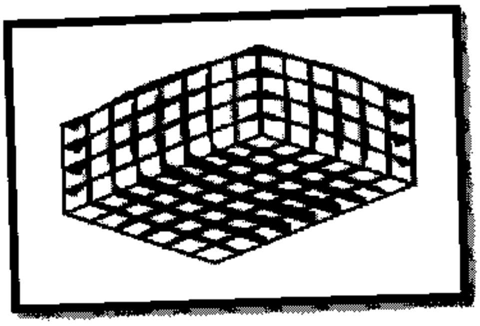


Fig.(2-3) Moment diagram for grillage beams of rectangular concrete tank

The grillage pattern can be made coarser to reduce the input time and model size but at the expense of some loss of accuracy and smoothness of the results plot. This can be more practical for large multiple tank structures.

2-1-3 Finite Element Method

This method subdivides each panel into a mesh of small elements. The element thickness is specified and the supports or springs can be added at nodes in the same way as for the grillage. The pressure loads on the panels can be applied by specifying the intensity on the panel as a whole Finite Element Analysis (FEA) programs can produce a colored results value contour plot as shown in Fig.(2-4).

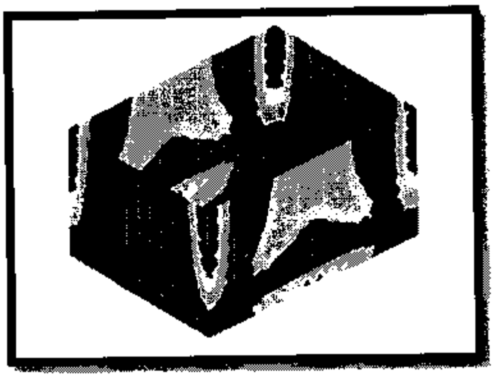


Fig (2-4) Contour plot of rectangular tank

The results from FEA can improve on the accuracy and smoothness of the grillage method and can be presented with values per meter width. In some FEA programs the results plots are based on the average or centre of element value so it is important in these cases to use small elements at panel joints. The coarse mesh option therefore is not always appropriate with FEA but the speed of mesh generation and loading input outweighs this. The FEA method can show very high local force values at point loads such as at pile supports so these need to be modeled carefully to reflect their real width and load spread.

2-1-4 Comparisons

The grillage method uses general purpose 3-D space frame analysis software which is used regularly by engineers in many design offices, whereas the FEA programs are more specialized and are not used as generally. So this can limit its use to the larger organizations. The preparation of the grillage model is not as fast as FEA mesh generation because the beams must be defined for each beam spacing and panel thickness condition.

The application of uniform loading is equally fast by both methods but hydrostatic loading is much faster by FEA by virtue of its full panel loading facility. A Pentium computer can now analyze the above

FEA model in less than 5 minutes so run times are no longer a real issue. The output from the grillage model is more familiar to structural engineers but it does need to be converted manually to show the results per meter width. This requires a degree of vigilance by the engineer.

2-1-5 Developments

The grillage method needs to be automated further so that the designer can specify the panel width and beam spacing and let the computer calculate the beam properties and output the results in a per meter width format.

The loading input method also needs to be enhanced to allow globally varying loads to be applied to the panel as a whole.

FEA programs like Staad Pro which are to be used for tank analysis should be able to access and plot the results at the element boundaries and plot moment diagrams. Both analysis methods would benefit from a library of tank models which could be modified or multiplied by simple input from the engineer. The most valuable benefit however, would be the automation of the reinforcement detailing process. The results could easily be fed through a post processor to calculate the requirements to specified code. The engineer could specify the main parameters and preferred spacing and the computer could then produce reinforcement proposal diagrams for review. The information could then be passed to the detailing program to produce drawings to a standard format similar to the wall elevation shown in Fig.(2-5).

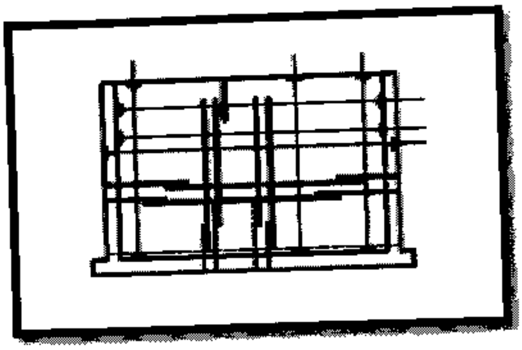


Fig.(2-5) Detail drawing

The details would then be checked and edited where necessary for holes and items not covered by the main tank analysis. It is quite possible therefore, to automate the design process for many tanks and bring cost benefits to the water industry.

2-2 Basic Relations of Plate Analysis

Plates are flat structures in which the thickness (t) is small compared to the other two dimensions, viz., the length a and breadth b. An infinitesimal element under a system of normal and shearing group of stresses isolated from a loaded plate is shown in Fig.(2-6). The thickness (t) of the plate is along z-axis

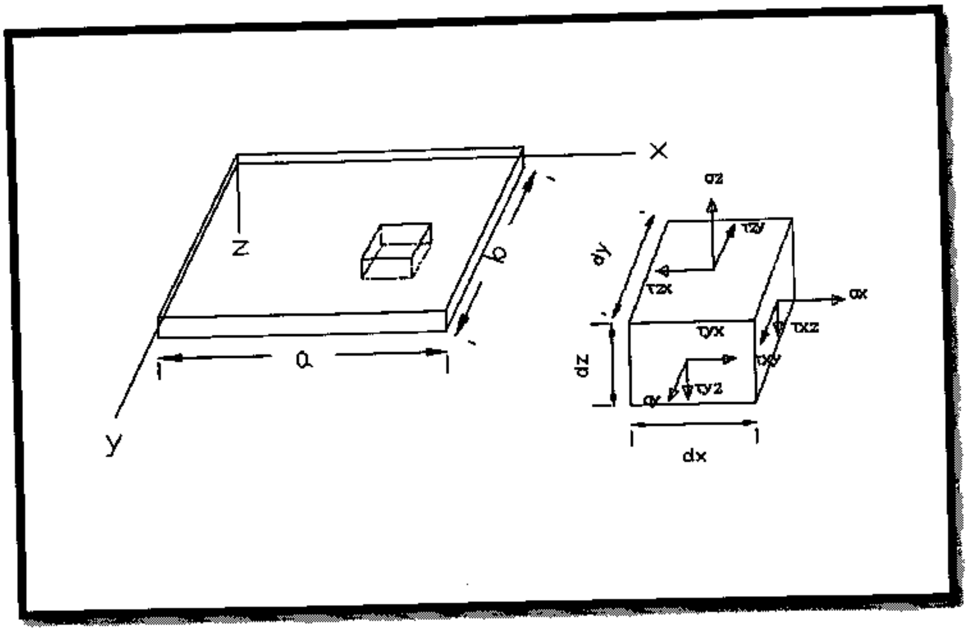


Fig.(2-6) system of normal and shearing group of

stresses isolated from a loaded plate

2-2-1 Thin and Thick Plates

Geometrically, a plate is said to be thin if the thickness ratio t' (=t/b, where b is the least dimension of the plate, b<a) remains normally less than 1/20. Plates having t' value greater than 1/20 are normally called thick .

In thin plate theory, the stresses σz , τxz (=τzx) and τyz (=τzy) which are directed towards the z-axis across the thickness, are considered to be of negligible magnitude compared to the other stresses viz., σx, σy and τxy (=τyx), and are disregarded. For all practical purposes, the plane of the thin plates remains as unstressed or neutral and serve as the reference plane.

In thick plate theory, the stress components σz, τxz and τyz are taken into consideration since they are of comparable magnitudes and cannot be neglected. The various in-plane stresses σx, σy and τxy(=τyx) do not remain proportional to the distance z measured from the neutral plane through the variation of the stresses τxz as well as τyz across the thickness can still be assumed to be parabolic (as assumed in the case of straight beams of

rectangular section). In essence, the analysis of thick plates are to be carried out using a 3-D analysis whereas that of thin plates can handled with a 2-D analysis.

2-2-2 Stress-Strain Relations

In 2-D problems, Hooke's law expressing strains in terms of stresses can be expressed as : $\epsilon = D\,\sigma$ in which elements of the D matrix can be written as :

$$D11 = D22 = \frac{1}{E}$$

$$D12 = D21 = -\frac{\nu}{E} \qquad (2.1)$$

$$D44 = \frac{2(1+\nu)}{E} = \frac{1}{G}$$

That is

$$\varepsilon x = \frac{1}{E}[\sigma x - \nu\sigma y] \qquad (2.2a)$$

$$\varepsilon y = \frac{1}{E}[\sigma y - \nu\sigma x] \qquad (2.2b)$$

$$\gamma xy = \frac{2(1+\nu)}{E}[\tau xy] \qquad (2.2c)$$

solving σx, σy in terms of εx and εy and τxy in terms of γxy, yields

$$\sigma x = \frac{E}{1-\nu^2}\ [\epsilon x + \nu\ \epsilon y] = \mathrm{C11}\varepsilon\mathrm{x} + \mathrm{C12}\ \epsilon y \qquad (2.3a)$$

$$\sigma y = \frac{E}{1-\nu^{2\nu}}[\epsilon y + \varepsilon x] = c12\ \varepsilon x + c22\ \varepsilon y \qquad (2.3b)$$

$$\tau xy = \frac{E}{2(1+\nu)}[\gamma xy] = c44\ \gamma xy \qquad (2.3c)$$

Where the Cij coefficients are the elements of the generalized modulus of elasticity matrix C (C = D^{-1}) so that

$C11=C22= E/(1-\nu^2)$

$$C12 = C21 = \frac{\nu E}{(1-\nu^2)} \quad (2.4)$$

$C44=E/[2(1+\nu)]$

The strain-curvature relations are :

$$\varepsilon x = -z\frac{\partial^2 \omega}{\partial x^2} = z\,K_x \quad (2.5a)$$

$$\varepsilon y = -z\frac{\partial^2 \omega}{\partial y^2} = z\,K_y \quad (2.5b)$$

$$\gamma xy = -2z\frac{\partial^2 \omega}{\partial x \partial y} = -2\,z\,K_{xy} \quad (2.5c)$$

On introduction (2.5) in (2.3), the stresses expressed in terms of the curvatures can be given as :

$$\sigma x = -\frac{E\,Z}{1-\nu^2}\left[\frac{\partial^2 \omega}{\partial x^2} + \nu\frac{\partial^2 \omega}{\partial y^2}\right] \quad (2.6a)$$

$$\sigma y = -\frac{E\,Z}{1-\nu^2}\left[\frac{\partial^2 \omega}{\partial y^2} + \nu\frac{\partial^2 \omega}{\partial x^2}\right] \quad (2.6b)$$

$$\tau xy = -\frac{E(1-\nu)Z}{(1-\nu^2)}\left[\frac{\partial^2 \omega}{\partial x \partial y}\right] \quad (2.6c)$$

2-2-3 Moment Curvature Relations

The magnitude of the various bending forces can be evaluated from the stresses referring to Fig.1.2 (c) as : (1.4a),(1.4b)and(1.4c).

Introducing the expressions of σx,σy and τxy from (2.6) , the moment –curvature relations can be given as :

$$Mx = \int_{-t/2}^{t/2} \sigma x \, z \, dz = -\frac{E}{1-\nu^2}\left[\frac{\partial^2\omega}{\partial x^2} + \nu\frac{\partial^2\omega}{\partial y^2}\right]\int_{-t/2}^{t/2} z^2 \; dz$$

$$= -D\left[\frac{\partial^2\omega}{\partial x^2} + \nu\frac{\partial^2\omega}{\partial y^2}\right] = D[Kx + \nu\, Ky] \qquad (2.7a)$$

$$My = \int_{-t/2}^{t/2} \sigma y \, z \, dz = -\frac{E}{1-\nu^2}\left[\frac{\partial^2\omega}{\partial y^2} + \nu\frac{\partial^2\omega}{\partial x^2}\right]\int_{-t/2}^{t/2} z^2 \; dz$$

$$= -D\left[\frac{\partial^2\omega}{\partial y^2} + \nu\frac{\partial^2\omega}{\partial x^2}\right] = D[Ky + \nu\, Kx] \qquad (2.7b)$$

with τxz=τyx,

$$Mxy = Myx = \int_{-t/2}^{t/2} -\tau xy \, z \, dz = \frac{E(1-\nu)}{1-\nu^2}\left[\frac{\partial^2\omega}{\partial x \partial y}\right]\int_{-t/2}^{t/2} z^2 \; dz$$

$$= D(1-\nu)\left[\frac{\partial^2\omega}{\partial x \partial y}\right] = D(1-\nu)[Kxy] \qquad (2.7c)$$

Where $D = \frac{E\,.t^3}{12(1-\nu^2)} = plate\ rigidity$ (2.7d)

2-2-4 Equilibrium Equation

Noting that the transverse shear and the moments act per unit length of the element (Fig. 2-7), the Equilibrium equations for the loaded flat plate elements are :

1. $\sum Fz$ (sum for forces along z) = 0 gives:

from $x^{+} and\ x^{-}$ faces

$$\left[Qx + \frac{\partial Qx}{\partial x}\, dx\right] dy - [Qx]dy \quad \text{or} \quad \left[\frac{\partial Qx}{\partial x}\right] dxdy$$

from $y^{+} and\ y^{-}$ faces

$$\left[Qy + \frac{\partial Qy}{\partial y}\, dy\right] dx - [Qy]dx \quad \text{or} \quad \left[\frac{\partial Qy}{\partial y}\right] dxdy$$

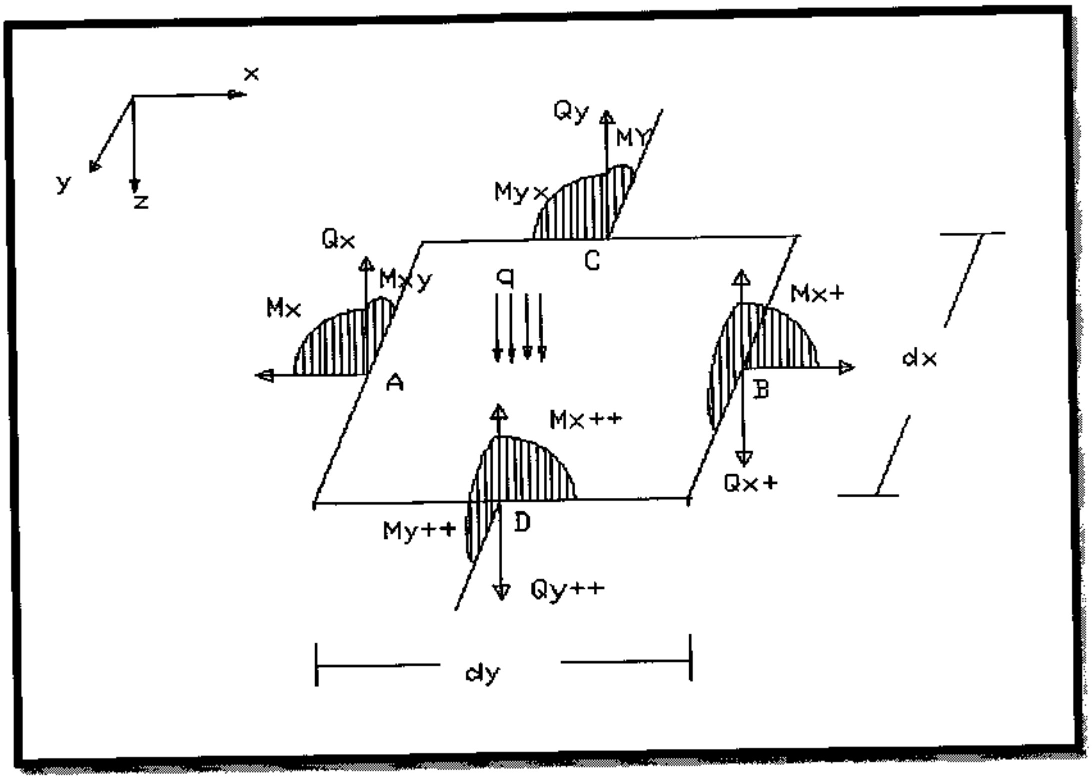


Fig.(2-7) transverse shear and the moments act per unit length of the element

From external loading

$[q]dxdy$

Adding these together with dx dy as common

$$\frac{\partial Qx}{\partial x} + \frac{\partial Qy}{\partial y} + q = 0 \qquad (2.8a)$$

2. $\sum My\ (sum\ of\ moment\ about\ AB\ ) = 0$ given:

From $x^{+} and\ x^{-}$ faces

$\left[Mxy + \frac{\partial Mxy}{\partial x} dx\right] dy - [Mxy]dy$ or $\left[\frac{\partial Mxy}{\partial x}\right] dx\, dy$

The contribution from Qx and Mx are absent.

From $y^{+} and\ y^{-}$ faces

$-\left[My + \frac{\partial My}{\partial y} dy\right] dx + [My]dx + \left[Qy + \frac{\partial Qy}{\partial y}\ dy + Qy\right] dx \frac{dy}{2}$

Or $\left[-\frac{\partial My}{\partial y} + Qy\right] dxdy$

The contribution from the external loading q and Mxy(=Mxy) are

absent and the product of smaller quantities larger than second order is neglected.

Adding these together with dx dy as common,

$$\left[Qy - \frac{\partial My}{\partial y} + \frac{\partial Mxy}{\partial x} = 0\right] \quad (2.8b)$$

3. $\sum Mx\ (sum\ of\ moment\ about\ CD\ ) = 0$ given:

From $x^{+} and\ x^{-}$ faces

$$-\left[Mx + \frac{\partial Mx}{\partial x}dx\right]dy + [Mx]dy + \left[Qx + Qx + \frac{\partial Qx}{\partial x}\ dx\right]dy\frac{dx}{2}$$

Or $\left[-\frac{\partial Mx}{\partial x} + Qx\right]dxdy$

The contribution of the external loading q and Mxy are absent are the product of smaller quantities larger than second order is neglected.

Adding these together with dx dy as common,

$$Qx - \frac{\partial Mx}{\partial x} + \frac{Myx}{\partial y} = 0 \quad (2.8c)$$

(3.12 a-c) are the three required equilibrium equations.

2-2-5 Plate Equation

The three equilibrium equations can be combined in to a single equation on elimination Qx and Qy in (2.8a) using (2.8c) and (2.8b) as follows:

From (2.8c)

$$Qx = \frac{\partial Mx}{\partial x} - \frac{Myx}{\partial y}$$

$$\frac{\partial Qx}{\partial x} = \frac{\partial^2 Mx}{\partial x^2} - \frac{\partial^2 Myx}{\partial x \partial y} \quad (2.9a)$$

$$Qy = \frac{\partial My}{\partial y} - \frac{Mxy}{\partial x}$$

$$\frac{\partial Qy}{\partial y} = \frac{\partial^2 My}{\partial y^2} - \frac{\partial^2 Mxy}{\partial x \partial y} \quad (2.9b)$$

Using (2.9 a,b) in (2.8a) and noting that Mxy=Myx :

$$\frac{\partial^2 Mx}{\partial x^2} + \frac{\partial^2 My}{\partial y^2} - 2\frac{\partial^2 Mxy}{\partial x \partial y} + q = 0 \quad (2.10)$$

This is the single equilibrium equation of rectangular plates of uniform thickness under continuous loading q=q(x,y) which can be expressed in terms of the displacement ω using the moment curvature relation (2.7a-c).

Now $\frac{\partial^2 Mx}{\partial x^2} = -D[\frac{\partial^4\omega}{\partial x^4} + \nu\frac{\partial^4\omega}{\partial x^2\partial y^2}]$

$$\frac{\partial^2 My}{\partial y^2} = -D[\frac{\partial^4\omega}{\partial y^4} + \nu\frac{\partial^4\omega}{\partial x^2\partial y^2}]$$

$$-2\frac{\partial^4 Mxy}{\partial x^2\partial y^2} = -D[(2-2\nu)\frac{\partial^4\omega}{\partial x^2\partial y^2}]$$

Adding these together and substituting in (2.10) :

$$-D\left[\frac{\partial^4\omega}{\partial x^4} + 2\frac{\partial^4\omega}{\partial x^2\partial y^2} + \frac{\partial^4\omega}{\partial y^4}\right] + q = 0 \qquad (2.11a)$$

Or $\nabla^4\omega = q/D$ (2.11b)

This equation is known as Sophie Germaine's equation(1815).

In non-dimensional form using

$$\bar{x} = \frac{x}{a}, \bar{y} = \frac{y}{b} \qquad (2.11c)$$

Where, a and b are the dimensions of the plate along x and y respectively, so that

$$\frac{\partial^4\omega}{\partial x^4} = \frac{1}{a^2}\frac{\partial^4\omega}{\partial \bar{x}^4}$$

$$\frac{\partial^4\omega}{\partial x^2\partial y^2} = \frac{1}{a^2b^2}\frac{\partial^4\omega}{\partial \bar{x}^2\partial \bar{y}^2} \qquad (2.11d)$$

$$\frac{\partial^4\omega}{\partial y^4} = \frac{1}{b^4}\frac{\partial^4\omega}{\partial \bar{y}^4}$$

We get the plate equation (2.11a) is

$$\frac{\partial^4\omega}{\partial \bar{x}^4} + 2c^2\frac{\partial^4\omega}{\partial \bar{x}^2\partial \bar{y}^2} + c^4\frac{\partial^4\omega}{\partial \bar{y}^4} = \frac{qa^4}{D}$$

Where c^2=constant=a^2/b^2 (2.11e)

In (2.11b) the operator ∇^4 is known as biharmonic operator.

$$\nabla^4 = \frac{\partial^4}{\partial x^4} + 2\frac{\partial^4}{\partial x^2 \partial y^2} + \frac{\partial^4}{\partial y^4} = \nabla^2(\nabla^2) \qquad (2.12a)$$

Where $\nabla^2 = laplace\ operator$

$$= \frac{\partial^2}{\partial x^2} + \frac{\partial^2}{\partial y^2} \qquad (2.12b)$$

2-3 Finite Element Approach

In the finite element method, the plate is divided into a series of small elements. These elements are then joined at their nodal points only and continuity, together with equilibrium, is established at these points (Ref. 5).

To derive the stiffness characteristics for the rectangular element, the co-ordinate and node numbering systems shown in Fig.(2-8.a), are used. The element has three degrees of freedom at each node, namely two rotations and the transverse deflection, i.e. $\theta x, \theta y$ *and* w. the positive directions of these rotation are defined according to the right-hand corkscrew rule. The element then has a total of twelve degrees of freedom as shown in Fig.(2-8.b). the corresponding moments and forces consist of two moments Tx and Ty and a force Fz at each node, see Fig.(2-8.c).

For example, for node (1) we can write the displacements as:-

$$\{\delta 1\} = \begin{Bmatrix} \theta x1 \\ \theta y1 \\ w1 \end{Bmatrix}$$

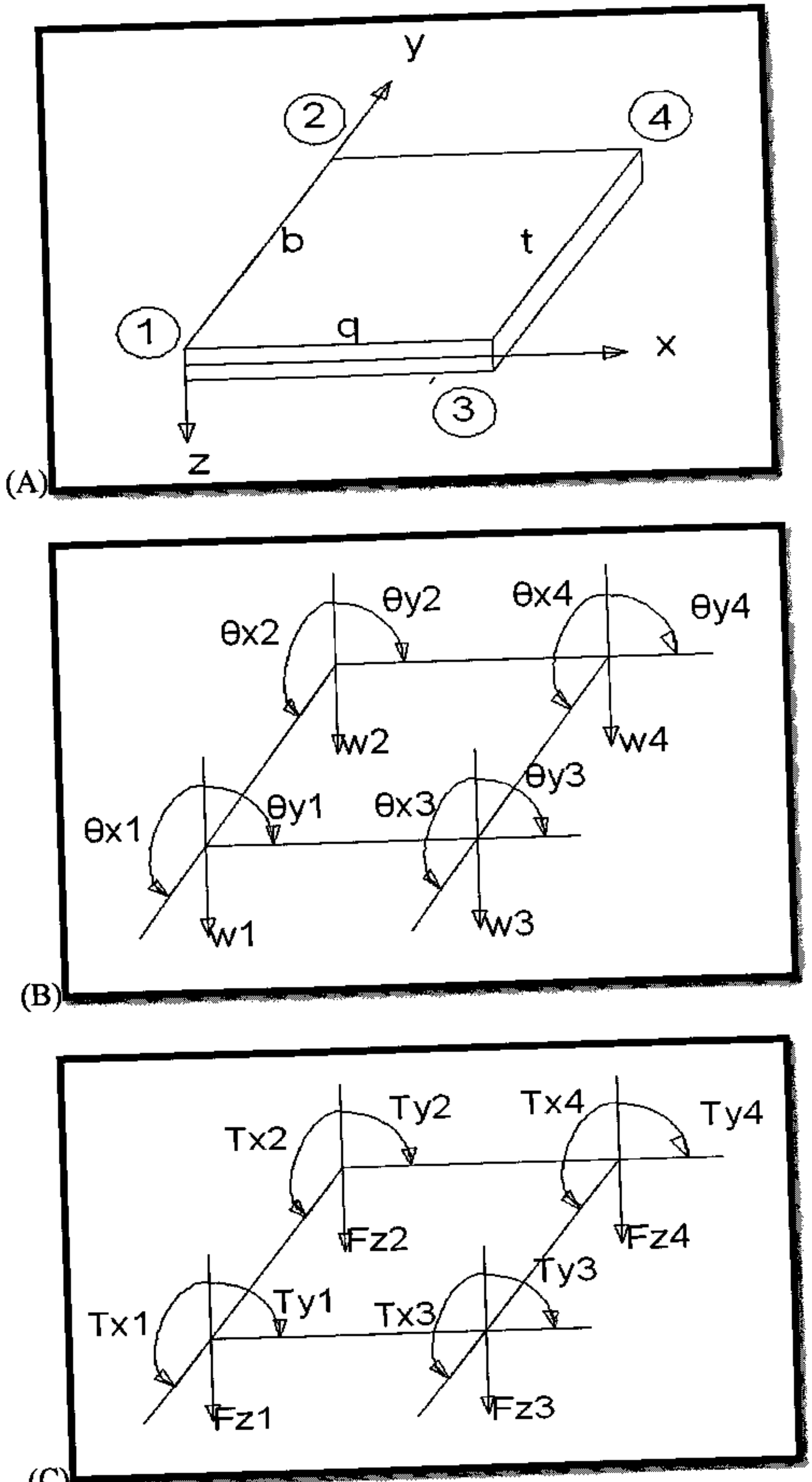
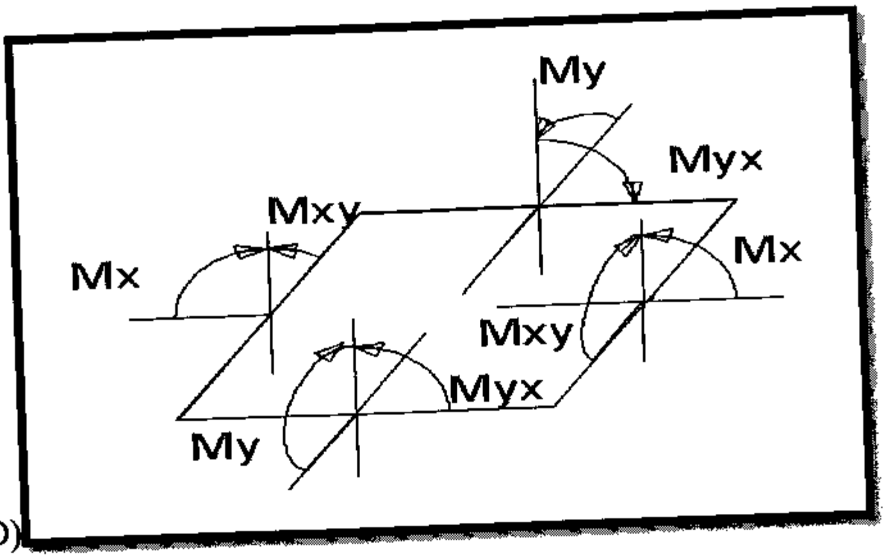


Fig.(2-8) continued



(D)

Fig.(2-8) rectangular plate bending element:-

A) Co-ordinate system.

B) Nodal displacement.

C) Nodal forces.

D) Internal moments.

and the corresponding moments and forces as :-

$$\{F1\} = \begin{Bmatrix} Tx1 \\ Ty1 \\ Tz1 \end{Bmatrix}$$

So that the complete displacement and force vectors for the element can be written as :-

And $\{\delta^e\} = \begin{Bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \\ \delta_4 \end{Bmatrix} = \begin{Bmatrix} \theta x_1 \\ \theta y_1 \\ w_1 \\ \theta x_2 \\ \theta y_2 \\ w_2 \\ \theta x_3 \\ \theta y_3 \\ w_3 \\ \theta x_4 \\ \theta y_4 \\ w_4 \end{Bmatrix}$

$$\text{And } \{F^e\} = \begin{Bmatrix} F1 \\ F2 \\ F3 \\ F4 \end{Bmatrix} = \begin{Bmatrix} Tx1 \\ Ty1 \\ Fz1 \\ Tx2 \\ Ty2 \\ Fz2 \\ Tx3 \\ Ty3 \\ Fz3 \\ Tx4 \\ Ty4 \\ Fz4 \end{Bmatrix}$$

$$\{F^e\} = [K^e]\{\delta^e\} \tag{2.13}$$

Where K^e is a 12x12 stiffness matrix of the element. Since the element has twelve degrees of freedom, a suitable function is chosen which has twelve undetermined

$$w = \propto_1 + \propto_2 \ x + \propto_3 \ y + \propto_4 \ x^2 + \propto_5 \ xy + \propto_6 \ y^2 + \propto_7 \ x^3 + \propto_8 x^2y + \propto_9 x\,y^2 + \propto_{10} \ y^3 + \propto_{11} \ x^3y + \propto_{12} xy^3 \tag{2.14}$$

The displacement function gives the following expressions for the rotations :-

$$\theta x = -\frac{\partial w}{\partial y} = -(\propto_3 + \propto_5 \ x + 2 \propto_6 \ y + \propto_8 \ x^2 + 2 \propto_9 \ xy + 3 \propto_{10} \ y^2 + \propto_{11} \ x^3 + 3 \propto_{12} xy^2)$$

$$\theta y = \frac{\partial w}{\partial x} = \propto_2 + 2 \propto_4 \ x + \propto_5 \ y + 3 \propto_7 \ x^2 + 2 \propto_8 xy \ + \propto_9 \ y^2 + 3 \propto_{11} x^2y + \propto_{12} y^3)$$

Equation (2.14) can be written in a matrix from as :-

$$\begin{Bmatrix} \theta x \\ \theta y \\ w \end{Bmatrix} = \begin{bmatrix} 0 & 0 & -1 & 0 & -x & -2y & 0 & -x^2 & -2xy & -3y^2 & -x^3 & -3xy^2 \\ 0 & 1 & 0 & 2x & y & 0 & 3x^2 & 2xy & y^2 & 0 & 3x^2y & y^3 \\ 1 & x & y & x^2 & xy & y^2 & x^3 & x^2y & xy^2 & y^3 & x^3y & xy^3 \end{bmatrix} \begin{Bmatrix} \propto_1 \\ \propto_2 \\ \propto_3 \\ \propto_4 \\ \propto_5 \\ \propto_6 \\ \propto_7 \\ \propto_8 \\ \propto_9 \\ \propto_{10} \\ \propto_{11} \\ \propto_{12} \end{Bmatrix} \tag{2.15}$$

Which can be summarized in generalmas:-

$$\{\delta(x,y)\} = [f(x,y)]\{\alpha\} \tag{2.16}$$

Then, the substitution of the nodal co-ordinate values into equation (2.16) leads to the formation of the [A] matrix as :-

$$\{\delta^e\} = [A]\{\alpha\} \tag{2.17}$$

This [A] matrix is a 12x12 matrix for the element has a total of twelve degrees of freedom:-

$$[A] = \begin{bmatrix} 0 & 0 & -1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 1 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ \\ 0 & 0 & -1 & 0 & 0 & -2b & 0 & 0 & 0 & -3b^2 & 0 & 0 \\ 0 & 1 & 0 & 0 & b & 0 & 0 & 0 & b^2 & 0 & 0 & b^3 \\ 1 & 0 & b & 0 & 0 & b^2 & 0 & 0 & 0 & b^3 & 0 & 0 \\ \\ 0 & 0 & -1 & 0 & -a & 0 & 0 & -a^2 & 0 & 0 & -a^3 & 0 \\ 0 & 1 & 0 & 2a & 0 & 0 & 3a^2 & 0 & 0 & 0 & 0 & 0 \\ 1 & a & 0 & a^2 & 0 & 0 & a^2 & 0 & 0 & 0 & 0 & 0 \\ \\ 0 & 0 & -1 & 0 & -a & -2b & 0 & -a^2 & -2ab & -3b^2 & -a^3 & -3ab^2 \\ 0 & 1 & 0 & 2a & b & 0 & 3a^2 & 2ab & b^2 & 0 & 3a^2b & b^3 \\ 1 & a & b & a^2 & ab & b^2 & a^3 & a^2b & ab^2 & b^3 & a^3b & ab^3 \end{bmatrix} \tag{2.18}$$

Node1 x=0 , y=0   Node2 x=0 , y=b   Node3 x=a , y=o   Node4 x=a , y=b

From equation (2.17)

$$\{\alpha\} = [A]^{-1}\{\delta^e\}$$

Substituting in to equation (2.16):-

$$\{\delta(x,y)\} = [f(x,y)][A]^{-1}\{\delta^e\} \tag{2.19}$$

Then, the state of strain in the element can be represented as, from theory of plates:-

$$\{\varepsilon(x,y)\} = \begin{Bmatrix} -\partial^2 w/\partial x^2 \\ -\partial^2 w/\partial y^2 \\ 2\partial^2 w/\partial x \partial y \end{Bmatrix} \qquad (2.20)$$

And substituting for w from equation (2.14) gives

{ε(x,y)}=

$$\begin{Bmatrix} -\partial^2 w/\partial x^2 \\ -\partial^2 w/\partial y^2 \\ 2\partial^2 w/\partial x \partial y \end{Bmatrix} = \begin{Bmatrix} -(2\propto_4 + 6\propto_7\ x + 2\propto_8\ y + 6\propto_{11}\ xy) \\ -(2\propto_6 + 2\propto_9\ x + 6\propto_{10}\ y + 6\propto_{12}\ xy) \\ 2(\propto_5 + 2\propto_8 x + 2\propto_9 y + 3\propto_{11} x^2 + 3\propto_{12} y^2) \end{Bmatrix}$$

$$= \begin{bmatrix} 0 & 0 & 0 & -2 & 0 & 0 & -6x & -2y & 0 & 0 & -6xy & 0 \\ 0 & 0 & 0 & 0 & 0 & -2 & 0 & 0 & -2x & -6y & 0 & -6xy \\ 0 & 0 & 0 & 0 & 2 & 0 & 0 & 4x & 4y & 0 & 6x^2 & 6y^2 \end{bmatrix} \begin{Bmatrix} \propto_1 \\ \propto_2 \\ \propto_3 \\ \propto_4 \\ \propto_5 \\ \propto_6 \\ \propto_7 \\ \propto_8 \\ \propto_9 \\ \propto_{10} \\ \propto_{11} \\ \propto_{12} \end{Bmatrix} \qquad (2.21)$$

This above equation can be summarized as :-

$$\{\varepsilon(x,y)\} = [C]\{\alpha\} \qquad (2.22)$$

But from equation (2.17):-

$$\{\propto\} = [A]^{-1}\{\delta^e\}$$

Then substituting in to equating (2.22):-

$$\{\varepsilon(x,y)\} = [C][A]^{-1}\{\delta^e\}$$

Let [B] = $[C][A]^{-1}$

$$\therefore \{\varepsilon(x,y)\} = [B]\{\delta^e\} \qquad (2.23)$$

Next step, we must relate internal stresses {σ(x,y)} to strains {ε(x,y)} and to nodal displacement $\{\delta^e\}$. in a plate flexure solution the internal

stresses are :-

$$\{\sigma(x,y)\} = \begin{Bmatrix} M_x \\ M_y \\ M_{xy} \end{Bmatrix} \qquad (2.24)$$

Where Mx and My are the internal bending moments per unit length and Mxy is the internal twisting moment per unit length set up within the element, see Fig. (2-8 d).

From plate bending theory, the stress-strain relationships are:-

$$Mx = -\left(Dx\frac{\partial^2 w}{\partial x^2} + D1\frac{\partial^2 w}{\partial y^2}\right)$$

$$My = -\left(Dy\frac{\partial^2 w}{\partial y^2} + D1\frac{\partial^2 w}{\partial x^2}\right)$$

$$Mxy = 2Dxy\frac{\partial^2 w}{\partial x\, \partial y}$$

Where Dx and Dy are the flexural rigidities in the x and y directions respectively, D1 is a coupling rigidity representing a Poisson's ratio type of effect and Dxy is the torsional rigidity.

But for an isotropic plate:-

$$Dx = Dy = D = \frac{E\, t^3}{12\,(1-\nu^2)}$$

$$D1 = \nu\, D$$

And Dxy= $\frac{1}{2}(1-\nu)D$

Then we can write equation (2.24)and(2.25) in a matrix form as:-

$$\{\sigma(x,y)\} = \begin{Bmatrix} Mx \\ My \\ Mxy \end{Bmatrix} = \begin{bmatrix} Dx & D1 & 0 \\ D1 & Dy & 0 \\ 0 & 0 & Dxy \end{bmatrix} \begin{Bmatrix} -\dfrac{\partial^2 w}{\partial x^2} \\ -\dfrac{\partial^2 w}{\partial y^2} \\ -\dfrac{2\partial^2 w}{\partial x \partial y} \end{Bmatrix} \qquad (2.26)$$

$$\{\sigma(x,y)\} = [D]\{\varepsilon(x,y)\} \qquad (2.27)$$

This defining the [D] matrix .substituting for {ε(x,y)} from general equation (2.23) gives the required relationship between element stresses and nodal displacement as :- $\{\sigma(x,y)\} = [D][B]\{\delta^e\}$

The internal stresses $\{\sigma(x,y)\}$ are now replaced by statically equivalent nodal forces $\{F^e\}$. the principle of virtual work is used, where during any arbitrary virtual displacements imposed on the element, the total external work done by the nodal loads must equal to the total internal work done by the stresses . selecting a set of virtual nodal displacements: $\{\delta^{\star e}\} = \{I\}$ (the identity column matrix).

The external work done by the nodal loads is (W_{ext}),

$$W_{ext} = \{\delta^{\star e}\}^T\{F^e\} = \{I\}\{F^e\} = \{F^e\}$$

If the imposed virtual displacements causes strains $\{\varepsilon^{\star}(x,y)\}$ (curvatures) at a point within the element where the actual stresses are $\{\sigma(x,y)\}$ (bending moments per unit length), then the total internal work (W_{int}) done will be :

$$W_{int} = \int_0^b \int_0^a \{\varepsilon^{\star e}(x,y)\}^T\{\sigma(x,y)\}\,\mathrm{dx}\,\mathrm{dy}.$$

Substituting,

$$\{\varepsilon^{*}(x,y)\} = [B]\,\{\delta^{*e}\} \;= [B]\{I\} = [B]$$

And $\{\sigma(x,y)\} = [D][B]\,\{\delta^{e}\}$

Then

$$W_{int} = \int_0^b \int_0^a [B]^T[D][B]\,\mathrm{dx\,dy}.\{\delta^{e}\}$$

equation

$$W_{ext}{=}W_{int}$$

Then,

$$\{F^e\} = (\int_0^b \int_0^a [B]^T[D][B]\,\mathrm{dx\,dy}\,)\delta^{e} \qquad (2.28)$$

And the element stiffness matrix $[K^e]$ is defined as :-

$$[K^e] = \int_0^b \int_0^a [B]^T[D][B]\,\mathrm{dx\,dy} \qquad (2.29)$$

Thus the final value of the stiffness matrix $[K^e]$Obtained from this above equation is as given in the following equation (2.30) for general orthotropic case :-

$$[K^e] = \frac{1}{15\,ab}\begin{bmatrix} SA & -SB & -SD & SG & 0 & -SH & SN & 0 & SO & SP & 0 & SQ \\ -SB & SC & SE & 0 & SI & SJ & 0 & SR & -SS & 0 & ST & -SU \\ -SD & SE & SF & SH & SJ & SM & SO & SS & SX & -SQ & SU & SY \\ SG & 0 & SH & SA & SB & SD & SP & 0 & -SQ & SN & 0 & -SO \\ 0 & SI & SJ & SB & SC & SE & 0 & ST & -SU & 0 & SR & -SS \\ -SH & SJ & SM & SD & SE & SF & SQ & SU & SY & -SO & SS & SX \\ SN & 0 & SO & SP & 0 & SQ & SA & SB & -SD & SG & 0 & -SH \\ 0 & SR & SS & 0 & ST & SU & SB & SC & -SE & 0 & SI & -SJ \\ SO & -SS & SX & -SQ & -SU & SY & -SD & -SE & SF & SH & -SJ & SM \\ SP & 0 & -SQ & SN & 0 & -SO & SG & 0 & SH & SA & -SB & SD \\ 0 & ST & SU & 0 & SR & SS & 0 & SI & -SJ & -SB & SC & -SE \\ SQ & -SU & SY & -SO & -SS & SX & -SH & -SJ & SM & SD & -SE & SF \end{bmatrix} \qquad (2.30)$$

Where:

$$P = \frac{a}{b}$$

$$SA = 20\,a^2\,Dy + 8\,b^2 Dxy$$

$$SB = 15\,a\,b\,D_1$$

$$SC = 20\,b^2\,Dx + 8\,a^2\,Dxy$$

$$SD = 30\,a\,p\,Dy + 15\,b\,D_1 + \ 6\,b\,Dxy$$

$$SE = 30\,b\,p^{-1} Dx + 15\,a\,D_1 + 6\,a\,Dxy$$

$$SF = 60\,p^{-2} Dx + 60\,p^2 Dy + 30\,D_1 + 84\,Dxy$$

$$SG = 10\,a^2\,Dy - 2b^2 Dxy$$

$$SH = -30\,a\,p\,Dy - 6\,b\,Dxy$$

$$SI = 10\,b^2\,Dx - 8\,a^2\,Dxy$$

$$SJ = 15\,b\,p^{-1} Dx - 15\,a\,D_1 - 6\,a\,Dxy$$

$$SM = 30\,p^{-2}\,Dx - 60\,p^2\,Dy - 30\,D_1 - 84\,Dxy$$

$$SN = 10\ \ a^2\,Dy - 8\,b^2\,Dxy$$

$$SO = -15\,p\,a\,Dy + 15\,b\,D_1 + 6\,b\,Dxy$$

$$SP = 5\,a^2\,Dy + 2\,b^2\,Dxy$$

$$SQ = 15\,a\,p\,Dy - 6\,b\,Dxy$$

$$SR = 10\,b^2\,Dx - 2\,a^2\,Dxy$$

$$SS = 30\,b\,p^{-1}\,Dx + 6\,a\,Dxy$$

$$ST = 5\,b^2\,Dx + 2\,a^2\,Dxy$$

$$SU = 15\,b\,p^{-1}\,Dx - 6\,a\,Dxy$$

$$SX = -60\,p^{-2}\,Dx + 30\,p^2\,Dy - 30D_1 - 84\,Dxy$$

$$SY = -30\,p^{-2}\,Dx - 30\,p^{2}\,Dy + 30D_1 - 84\,Dxy$$

2-3-1 Assembly of Structure Stiffness Matrix :-

After establishing the stiffness matrices of all the elements in the structure, they are assembled together so that a relationship is obtained between the external forces acting at the nodal points of the structure and the displacements of these points.

Denoting all the external nodal forces by :-

$$N = \begin{Bmatrix} N_1 \\ , \\ , \\ N_i \\ , \\ , \end{Bmatrix}$$

And all the nodal displacement as:-

$$U = \begin{Bmatrix} U_1 \\ , \\ , \\ U_i \\ , \\ , \end{Bmatrix}$$

With N_i and U_i representing the three components of external forces and the three corresponding displacement at the*I the node respectively . thus:

[s] {U}={N}

Where S is the overall structure stiffness matrix . The elements of matrix S are built up by adding the stiffness of elements adjacent to a node ,as follows:

$$S_{ij}=\sum_{e=1,2,\dots} k_{ij}^{e}$$

2-3-2 Nodal Forces

In most of the problems with in this work , the plate is subjected to a uniformly distributed load of value "q", whereas in the derivation of the stiffness matrix , concentrated external nodal loads are assumed . Thus, the lumped load representation is used to replace the uniformly distributed load to equivalent concentrated load at the nodes.

2-3-3 Boundary Conditions

Three types of boundary condition involved in this work, they are: the fixed edge in which, the deflection and slope normal to that edge are equal zero, and the simply supported edge in which the deflection and the rotation about axis normal to the edge are equal to zero , and the free edge.

Staad Pro Program

3.1 Introduction

Staad offers general purpose structural analysis and design along with extensive model generation and post-processing facilities. All these features are integrated in one common Graphical User Interface (GUI). This manual describes the Staad GUI in detail.

The Staad GUI has a concept called Page Control. When it is switched on (using the Page Control option from the Mode menu), a tabbed menu appears along the left side of the screen as a guide to the process of creating a structure. Every "Page" serves a specific purpose. For example, the General | Load Page offers facilities to define different types of loads. There are two levels of tabs indicating pages and subpages. You should start from the top of the tabbed menu and gradually work down through the pages and subpages to input geometry, supports, loads, and other parameters.

In addition to the GUI, STAAD also offers an Input Command File interface for specifying the Input, Analysis, and Output commands. This file is a text file consisting of simple English-like commands. You develop or modify the model using either the Staad GUI or the Input Command File. When a model is created using the graphical tools, the input command file is automatically generated.

*This chapter is adopted from Staad Pro Help

شكر و تقدير

يسرنا بعد إن انهينا بحثنا هذا أن نتقدم بكل فخر واعتزاز وعظيم الامتنان إلى

الأستاذ الفاضل ((عمار عباس)) لما قدمه لنا من مساعدة كبيرة و توجيهات قيمة

ومتواصلة إثناء فترة المشروع . .

كما نود أن نتوجه بالشكر الجزيل إلى كل من الأساتذة الدكتور نمير خورشيد

رئيس القسم والدكتور مهند القزويني رئيس الفرع والى كل أساتذتنا الكرام

في قسم البناء والإنشاءات

هذا وأسأل الله أن يوفق الجميع

طلبة المشروع

ذوالفقار سالم حسين

آمنة حامد علوان

3-2 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 user may also use the element for plane stress action only. The element plane stress command should be used for this purpose.

3-3 Geometry Modeling Considerations

The following geometry related modeling rules should be remembered while using the Plate / Shell element.

1- The program automatically generates a fifth node "O" (center node - see Fig.3.1) 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.3.2). 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.

*This chapter is adopted from Staad Pro Help

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 see Fig(3.4).

3-4 Element Load Specification

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.

*This chapter is adopted from Staad Pro Help

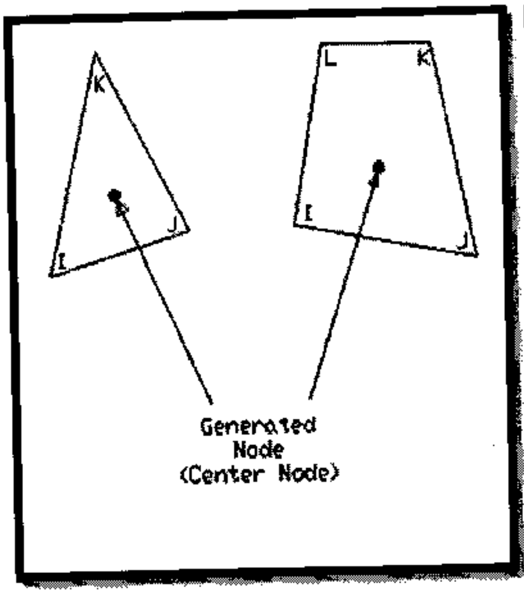


Fig.(3.1)Fifth node generation

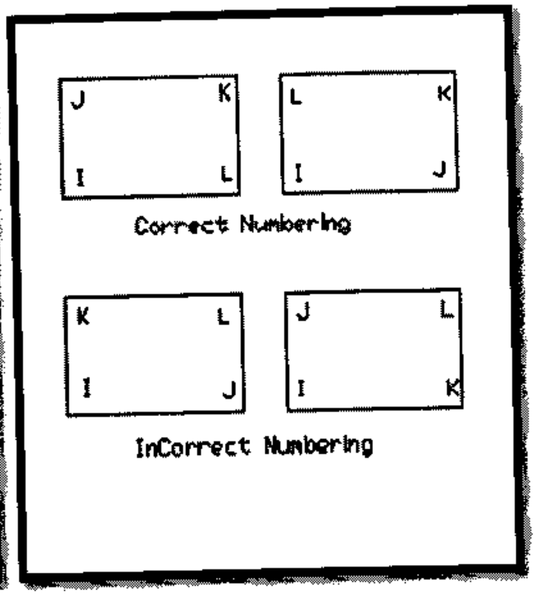


Fig.(3.2)Correct element numbering

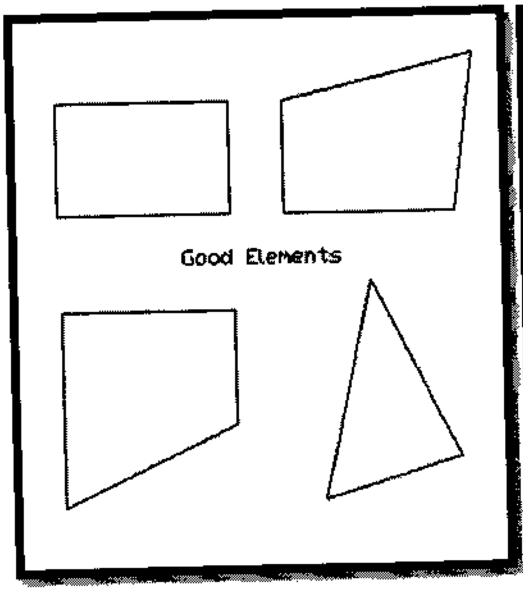


Fig.(3.3) Good element types

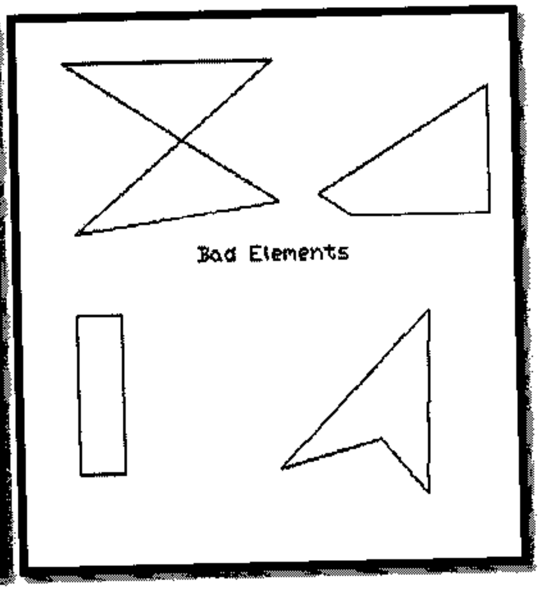


Fig.(3.4) Bad element types

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 Fig(3.5).

*This chapter is adopted from Staad Pro Help

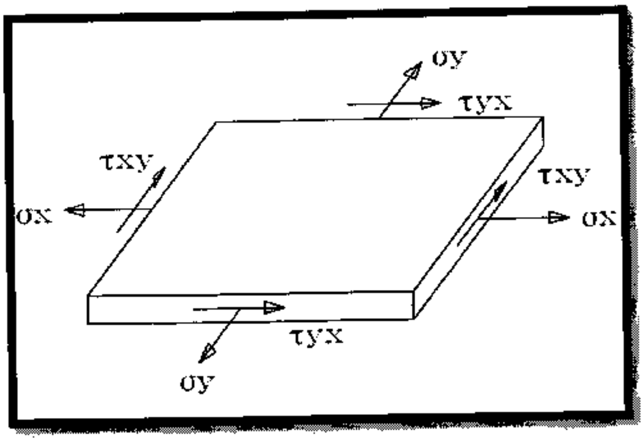


Fig.(3.5) Plane stresses

Complete quadratic assumed stress distribution:

$$\begin{pmatrix}\sigma_x\\ \sigma_y\\ \sigma_{xy}\end{pmatrix}$$

$$=\begin{bmatrix}1 & x & y & 0 & 0 & 0 & 0 & x^2 & 2xy & y^2 & 0 & 0\\ 0 & 0 & 0 & 1 & x & y & 0 & y^2 & 0 & 0 & x^2 & 2xy\\ 0 & -y & 0 & 0 & 0 & -x & 1 & -2xy & -y^2 & 0 & 0 & -x^2\end{bmatrix}\begin{pmatrix}a_1\\ a_2\\ a_3\\ .\\ .\\ a_{12}\end{pmatrix}$$

a_1 through a_{12} =constants of stress polynomials.

The following quadratic stress distribution is assumed for plate bending action Fig.(3.6):

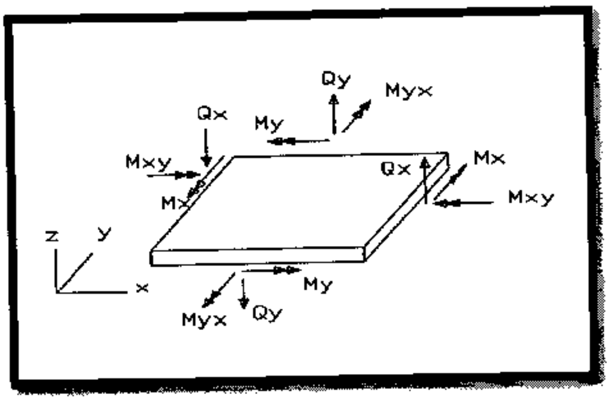


Fig.(3.6) Out-of- plane action

*This chapter is adopted from Staad Pro Help

Complete quadratic assumed stress distribution:

$$\begin{pmatrix} M_x \\ M_y \\ M_{xy} \\ Q_x \\ Q_y \end{pmatrix} = \begin{bmatrix} 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & y^2 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & 0 & 0 & 0 & x^2 & xy & y^2 & 0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & -xy & 0 & 0 & 0 & 0 & -xy & x^2 & y^2 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & 0 & 1 & x & y & 0 & 0 & 0 & -x & 0 & 2y \\ 0 & 0 & 0 & 0 & 0 & 1 & 0 & 1 & 0 & -y & 0 & 0 & 0 & x & y & 2x & 0 \end{bmatrix} \begin{pmatrix} a_1 \\ a_2 \\ a_3 \\ . \\ . \\ a_{17} \end{pmatrix}$$

a_1 through a_{17} =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.3.7) is achieved by the elements. This compatibility requirement is usually ignored in most flat shell/ plate elements.

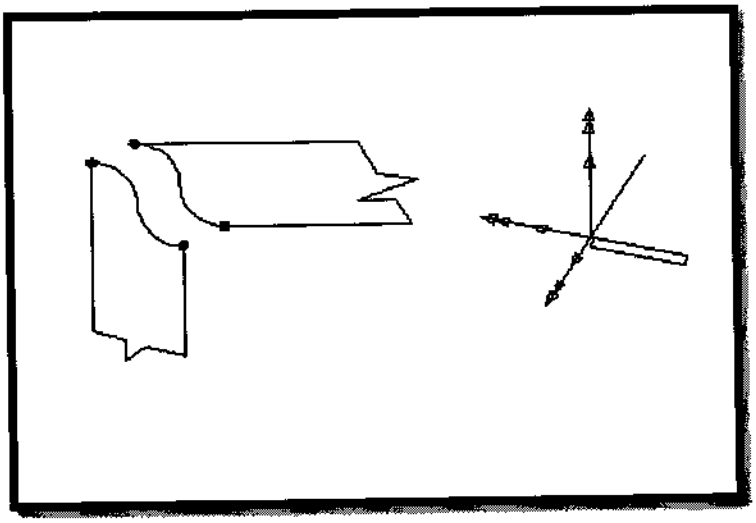


Fig.(3.7)Displacement compatibility

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.

*This chapter is adopted from Staad Pro Help

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 Kirchhoff 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.

*This chapter is adopted from Staad Pro Help

3-5 Element Local Coordinate System

The precise orientation of local coordinates is determined as follows:

1- Designate the midpoints of the four or three element edges IJ, JK, KL, LI by M, N, O, P respectively.

2- The vector pointing from P to N is defined to be the local x- axis. (In a triangle, this is always parallel to IJ).

3- The cross-product of vectors PN and MO (for a triangle, ON and MK) defines the local z-axis, i.e., z = PN x MO.

4- The cross-product of vectors z and x defines the local y- axis, i.e., y = z x x.

The sign convention of output force and moment resultants is illustrated in Fig.3.9.

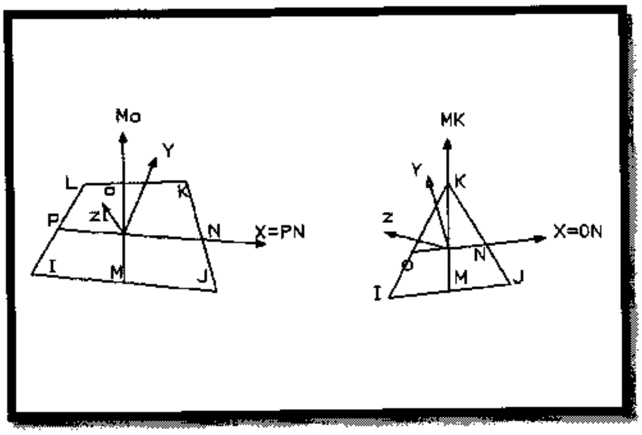


Fig.(3.8)Element designation

3-6 Output of Element Forces

Element force outputs are available at the following locations:

A. Center point of the element.

*This chapter is adopted from Staad Pro Help

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./unit thk.)
SX, SY, SXY	membrane stresses (force/ unit len./ unit thk.)
MX, MY, MXY	bending moments per unit width (moment/unit len.)
SMAX, SMIN	principal stresses (force/unit area)
TMAX	maxim. Shear stresses (force/unit area)
ANGLE	orientation of the principal plane (degrees)
VONT, VONB	Von Mises stress

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

Notes:

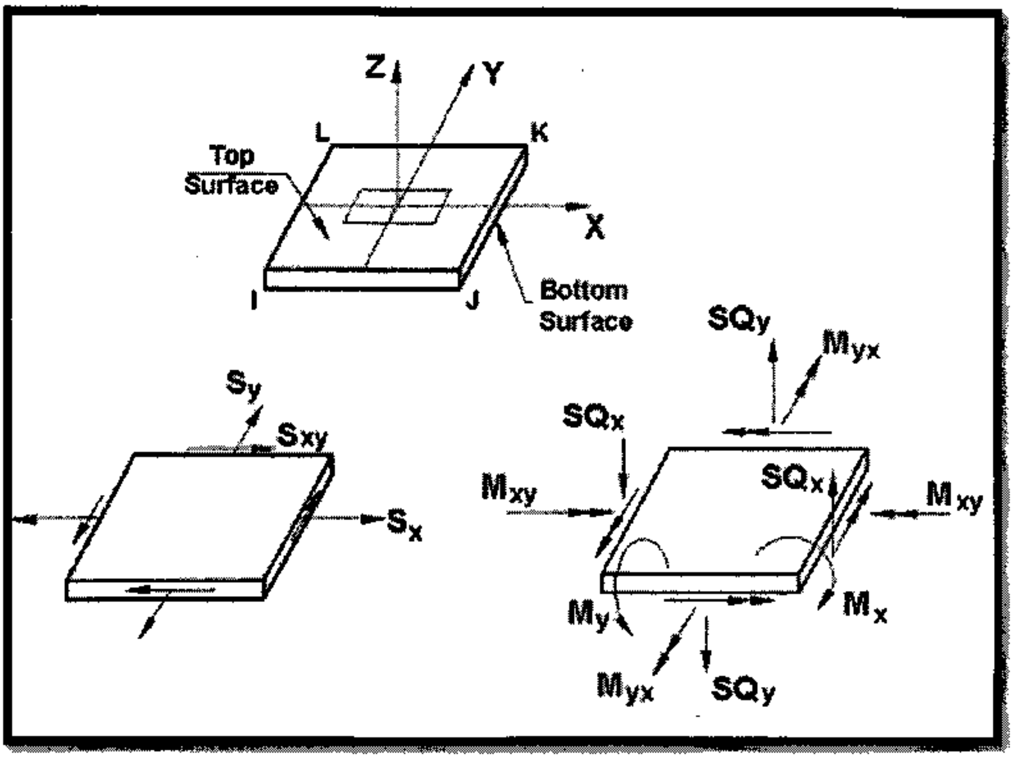
1. All element stress output is in the local coordinate system. The direction and sense of the element stresses are explained in Fig.3.9

2. To obtain element stresses at a specified point within the element, the user must provide the coordinate system for the element. Note that the origin of the local coordinate system coincides with the center node of the element.

3. Principal stresses (SMAX & SMIN), the maximum shear stress (TMAX), the orientation of the principal plane (ANGLE), and the Von Mis Stress (VONT & VONB) are also printed for the top and bottom

*This chapter is adopted from Staad Pro Help

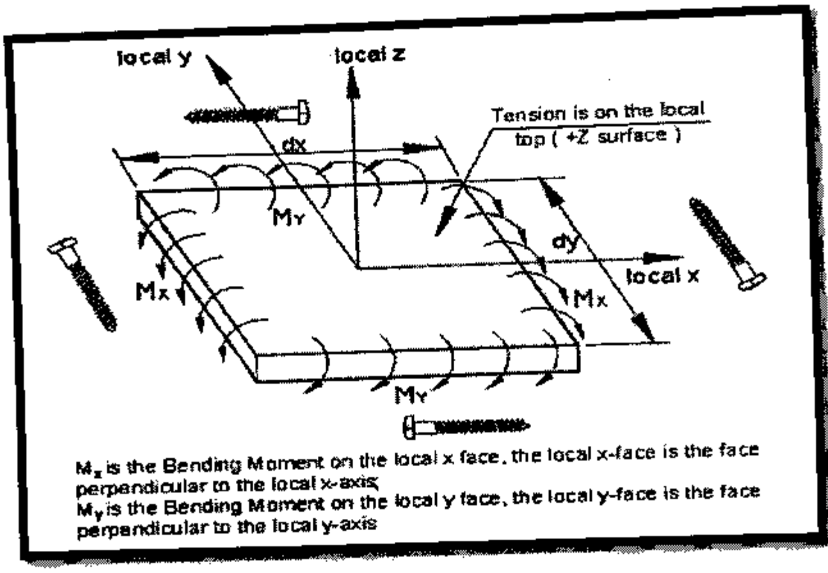
surfaces of the elements. The top and the bottom surfaces are determined on the basis of the direction of the local z-axis.



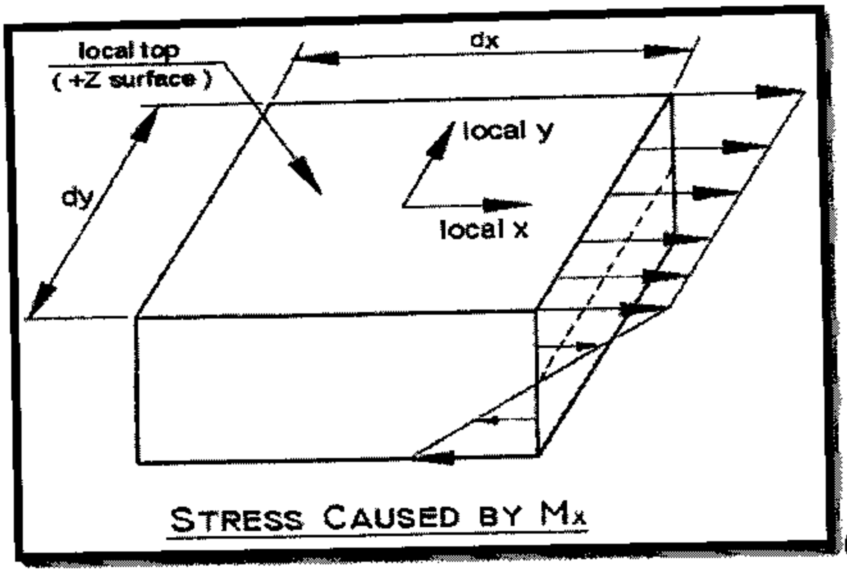
(A)

Fig.(3.9)Sign Convention of Element Forces

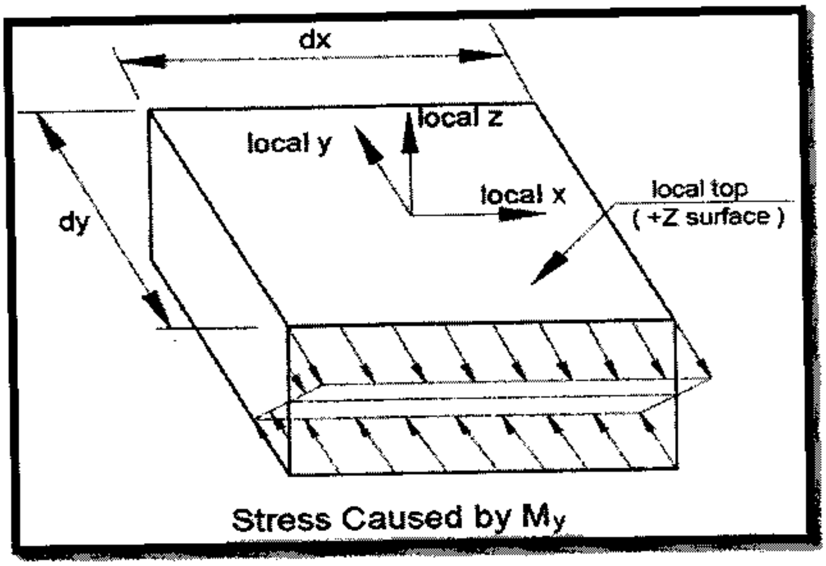
*This chapter is adopted from Staad Pro Help



(B)



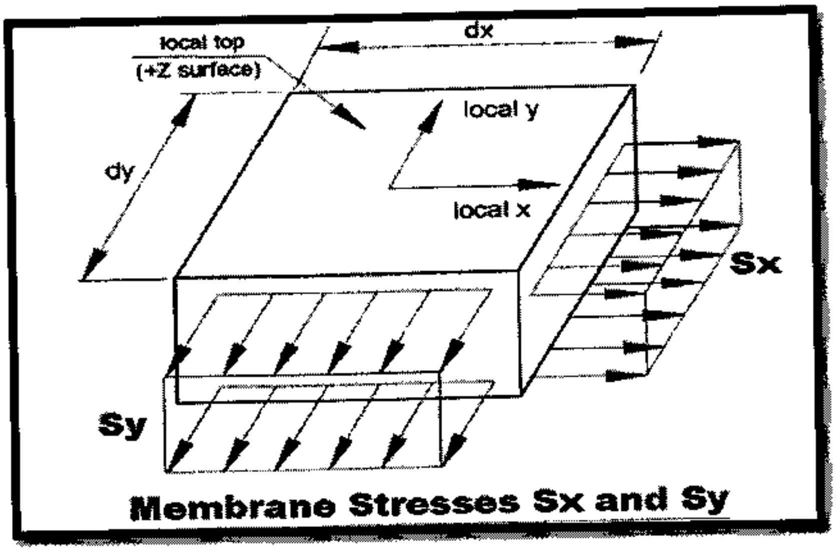
(C)



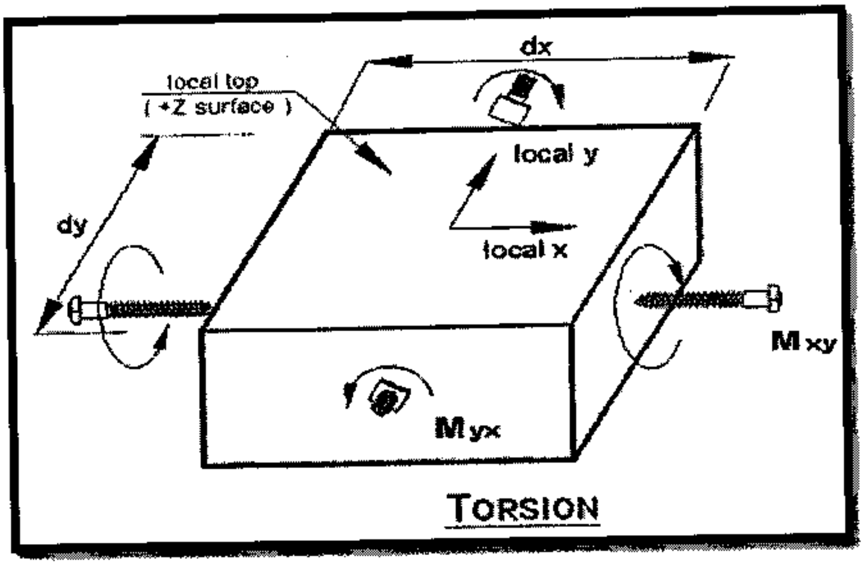
(D)

Fig(3.9) continued

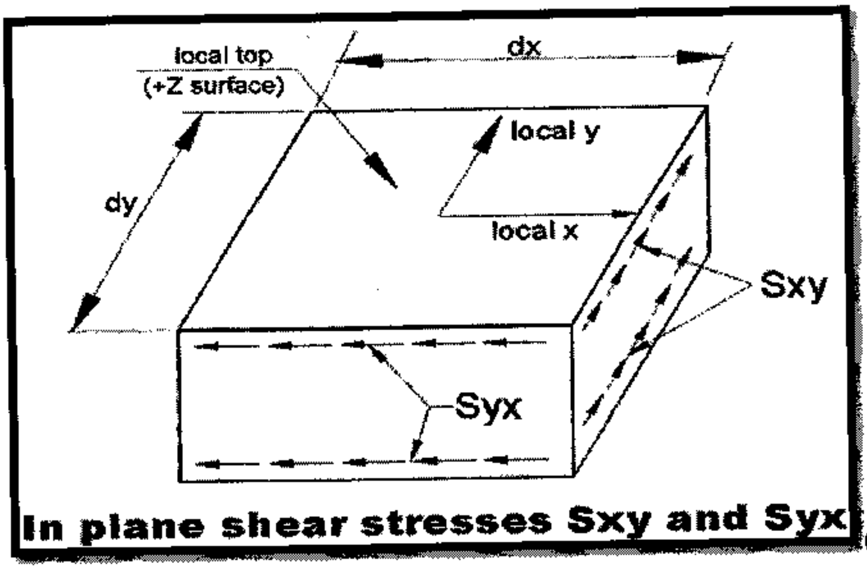
*This chapter is adopted from Staad Pro Help



(E)



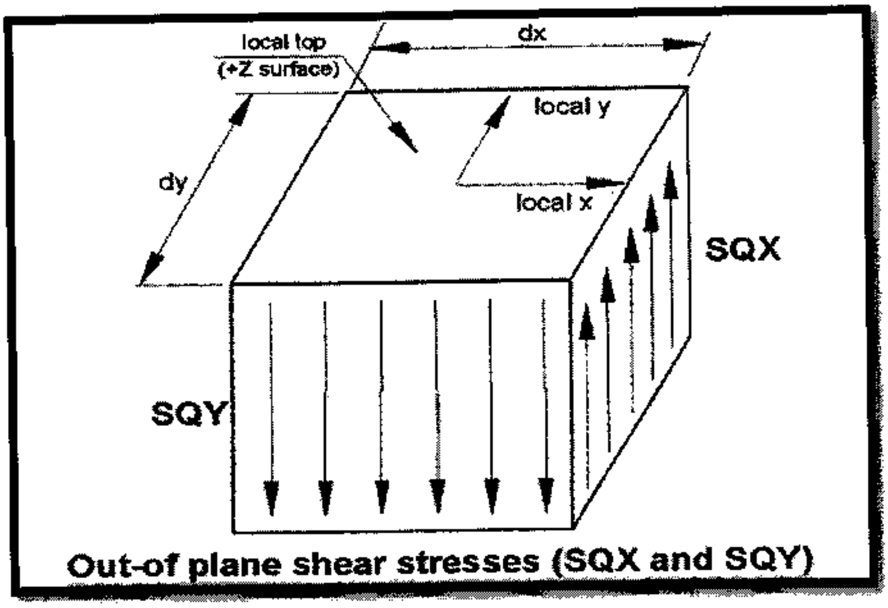
(F)



(G)

Fig(3.9) continued

*This chapter is adopted from Staad Pro Help



(H)

Fig(3.9) continued

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 self weight 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.3.9, the Von Mises stresses at the top and bottom surface of the element are also printed.

*This chapter is adopted from Staad Pro Help

3-7 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.3.10 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.

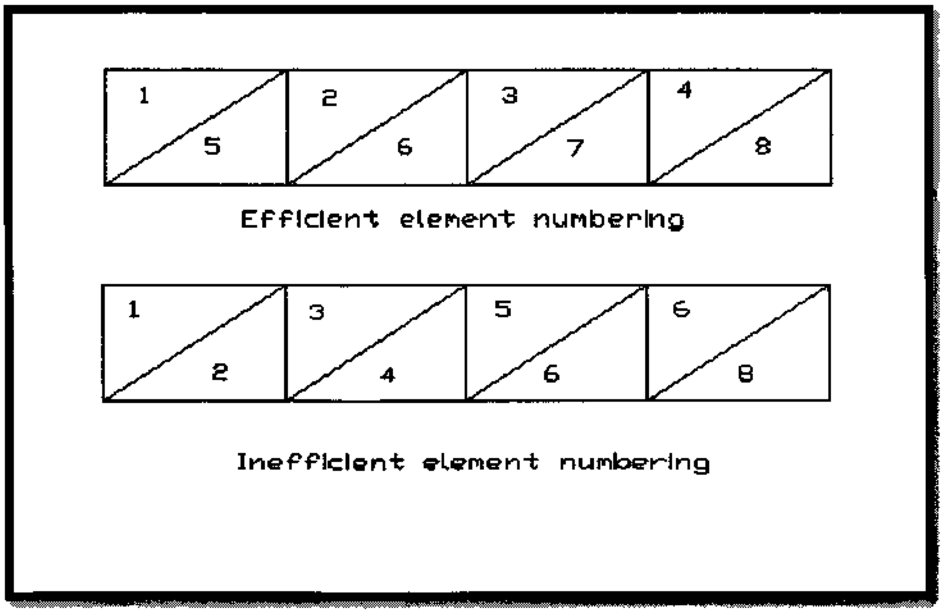
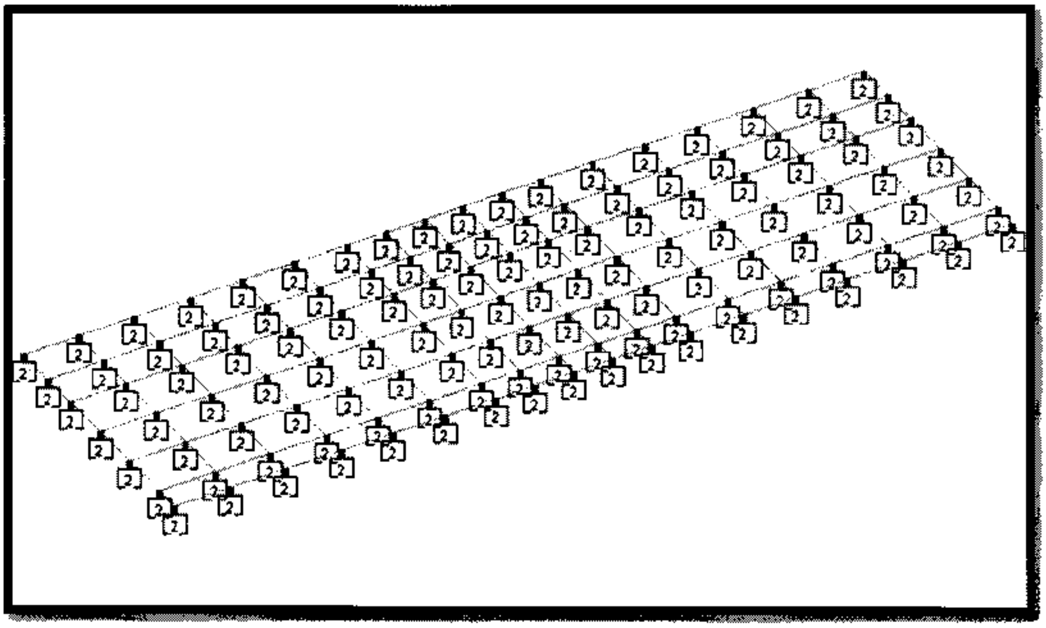


Fig.(3.10) Element numbering

*This chapter is adopted from Staad Pro Help

Example



This example illustrates the usage of commands necessary to utilize the built-in generation facility to generate spring supports for a slab on grade. The slab is subjected to various types of loading and analysis of the structure is performed. The numbers shown in the diagram below are the element numbers.

STAAD SPACE SLAB ON GRADE

Every STAAD input file has to begin with the word STAAD. The word SPACE signifies that the structure is a space frame and the geometry is defined through X, Y and Z axes. The remainder of the words form a title to identify this project.

UNIT FEET KIP

The units for the data that follows are specified above.

JOINT COORDINATES
1 0.0 0.0 40.0
2 0.0 0.0 36.0
3 0.0 0.0 28.167
4 0.0 0.0 20.333

*This chapter is adopted from Staad Pro Help

```
5 0.0 0.0 12.5
6 0.0 0.0  6.5
7 0.0 0.0  0.0
REPEAT ALL 3 8.5 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 5 6.0 0.0 0.0
REPEAT 3 8.0 0.0 0.0
REPEAT 3 8.5 0.0 0.0
```

For joints 1 through 7, the joint number followed by the X, Y and Z coordinates are specified above. The coordinates of these joints is used as a basis for generating 21 more joints by incrementing the X coordinate of each of these 7 joints by 8.5 feet, 3 times. REPEAT commands are used to generate the remaining joints of the structure. The results of the generation may be visually verified using the STAAD graphical viewing facilities.

```
ELEMENT INCIDENCES
1 1 8 9 2 TO 6
REPEAT 16 6 7
```

The incidences of element number 1 is defined and that data is used as a basis for generating the 2nd through the 6th element. The incidence pattern of the first 6 elements is then used to generate the incidences of 96 (= 16 x 6) more elements using the REPEAT command.

```
UNIT INCH
ELEMENT PROPERTIES
1 TO 102 TH 5.5
```

The thickness of elements 1 to 102 is specified as 5.5 inches following the command ELEMENT PROPERTIES.

```
UNIT FEET
CONSTANTS
E 420000. ALL
POISSON 0.12 ALL
```

*This chapter is adopted from Staad Pro Help

The modulus of elasticity (E) and Poisson's Ratio are specified following the command CONSTANTS.

```
SUPPORTS
1 TO 126  ELASTIC MAT DIRECTION Y SUB 10.0
```

The above command is used to instruct STAAD to generate supports with springs which are effective in the global Y direction. These springs are located at nodes 1 to 126. The subgrade modulus of the soil is specified as 10 kip/cu.ft. The program will determine the area under the influence of each joint and multiply the influence area by the sub grade modulus to arrive at the spring stiffness for the "FY" degree of freedom at the joint. Additional information on this feature may be found in the STAAD Technical Reference Manual.

```
PRINT SUPP INFO
```

This command will enable us to obtain the details of the support springs which were generated using the earlier commands.

```
LOAD 1 WEIGHT OF MAT & EARTH
ELEMENT LOAD
1 TO 102 PR GY -1.55
```

The above data describe a static load case. A pressure load of 1.55 kip/sq.ft acting in the negative global Y direction is applied on all the 102 elements.

```
LOAD 2 'COLUMN LOAD-DL+LL'
JOINT LOADS
1 2 FY -217.
8 9 FY -109.
5 FY -308.7
6 FY -617.4
22 23 FY -410.
29 30 FY -205.
26 FY -542.7
27 FY -1085.4
```

*This chapter is adopted from Staad Pro Help

```
43 44 50 51 71 72 78 79 FY -307.5
47 54 82 FY -264.2
48 55 76 83 FY -528.3
92 93 FY -205.0
99 100 FY -410.0
103 FY -487.0
104 FY -974.0
113 114 FY -109.0
120 121 FY -217.0
124 FY -273.3
125 FY -546.6
```

Load case 2 consists of several joint loads acting in the negative global Y direction.

```
LOADING COMBINATION 101 TOTAL LOAD
1 1. 2 1.
```

A load combination case, identified with load case number 101, is specified above. It instructs STAAD to factor loads 1 and 2 by a value of 1.0 and then algebraically add the results.

```
PERFORM ANALYSIS
```

The analysis is initiated using the above command.

```
LOAD LIST 101
PRINT JOINT DISPLACEMENTS LIST 33 56
PRINT ELEMENT STRESSES LIST 34 67
```

Joint displacements for joints 33 and 56, and element stresses for elements 34 and 67, for load case 101, is obtained with the help of the above commands.

```
FINISH
```

*This chapter is adopted from Staad Pro Help

Application

4-1 General

This chapter deals with comparative study for concrete ground tanks with constant dimensions and properties of materials. The tank is of 6x4x4m and of 0.2m thickness as shown in Fig.(4-1) the properties of concrete are: compressive strength (Fc'=30), Modulus of elasticity (E=21.7185) and Poisson's ratio (ν=0.17). The analysis is focused on the longest wall with 6x4 dimension. The connections between wall-wall and wall-slab are considered as either fixed or pinned. For both cases displacements, stresses and bending moments calculations are done.

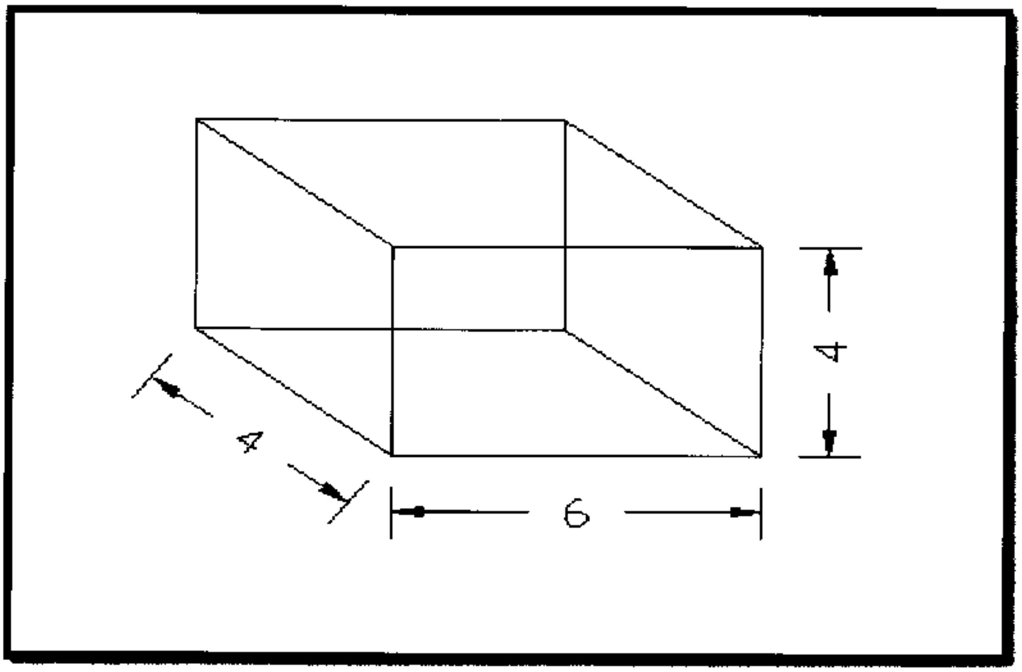


Fig.(4-1) Dimension **of tank**

4-2 Fixed Support :

Case1 : mesh (1x1)m

Displacement in z-direction for ground tank with 1x1 mesh

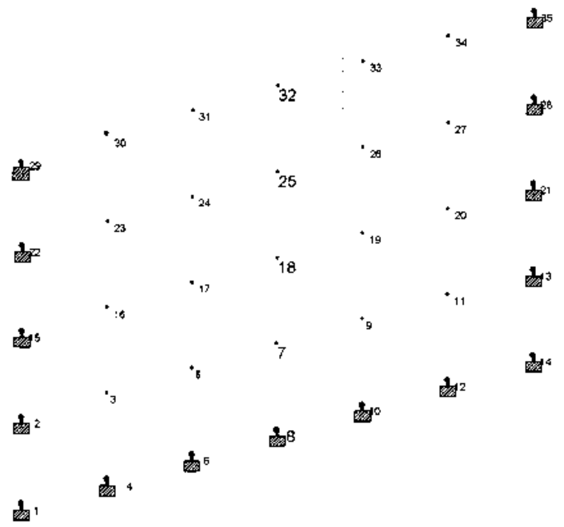


Table.(4.1) Maximum Displacement in z-direction

Node No.	Displacement (mm)
8	0.000
7	-0.645
18	-1.373
25	-1.602
32	-1.632

Case2 : mesh (0.5x0.5)

Displacement in z-direction for ground tank with 0.5x0.5 mesh

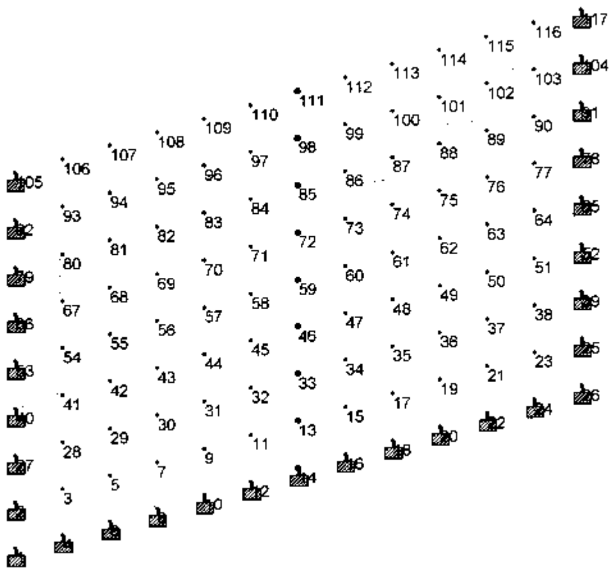


Table.(4.2) Maximum Displacement in z-direction

Node No.	Displacement (mm)
14	0.000
13	-0.245
33	-0.722
46	-1.190
59	-1.541
72	-1.755
85	-1.861
98	-1.910
111	-1.959

Case3 : mesh (0.25x0.25)

Displacement in z-direction for ground tank with 0.5x0.5 mesh

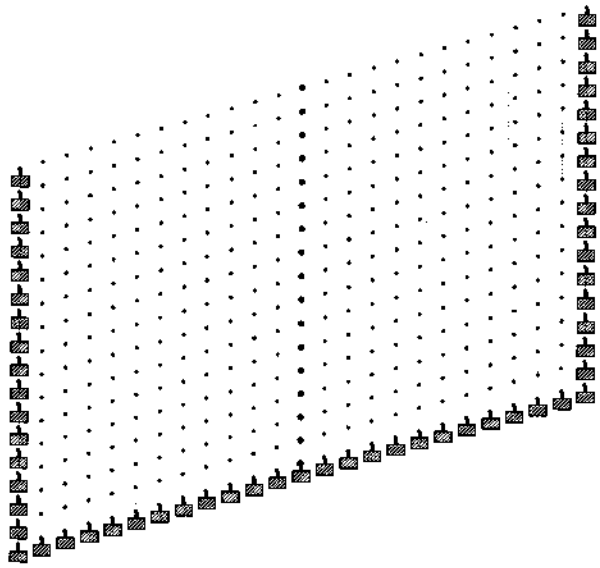


Table.(4.3) Maximum Displacement in z-direction

Node No.	Displacement (mm)	Node No.	Displacement (mm)
26	0.000	238	-1.713
25	-0.077	263	-1.811
63	-0.255	288	-1.883
88	-0.488	313	-1.933
113	-0.741	338	-1.970
138	-0.990	363	-2.000
163	-1.219	388	-2.029
188	-1.418	413	-2.065
213	-1.583		

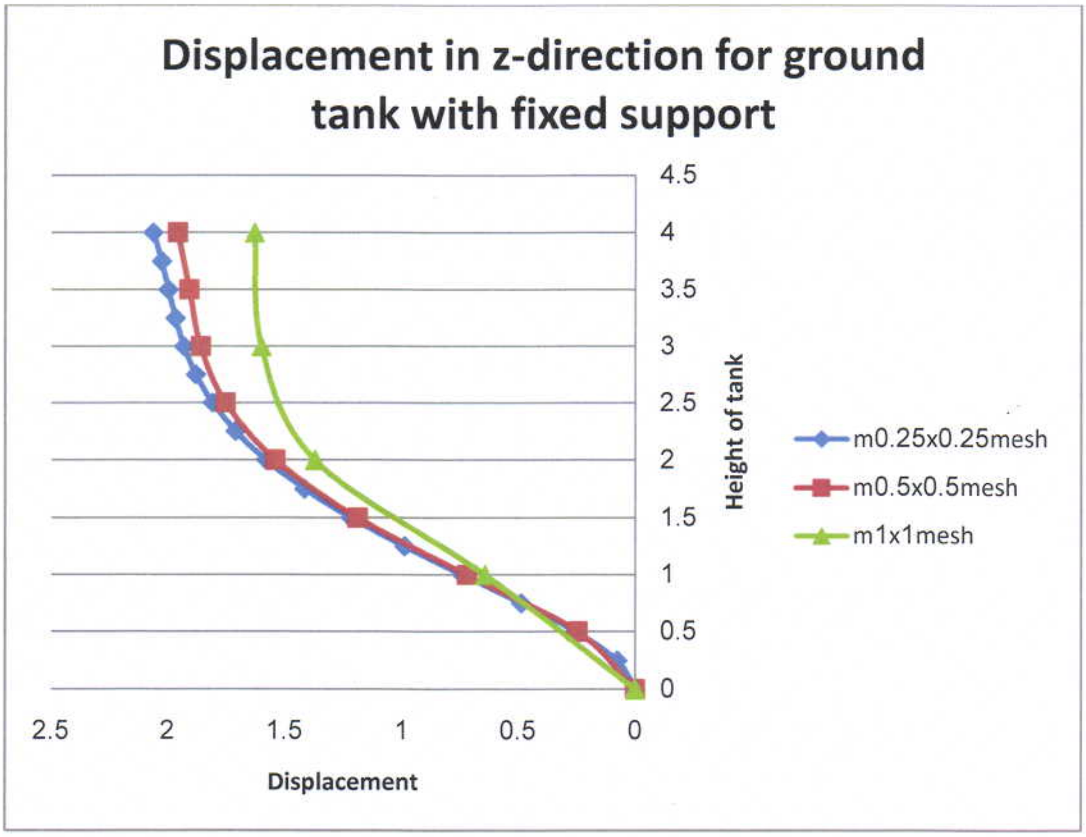
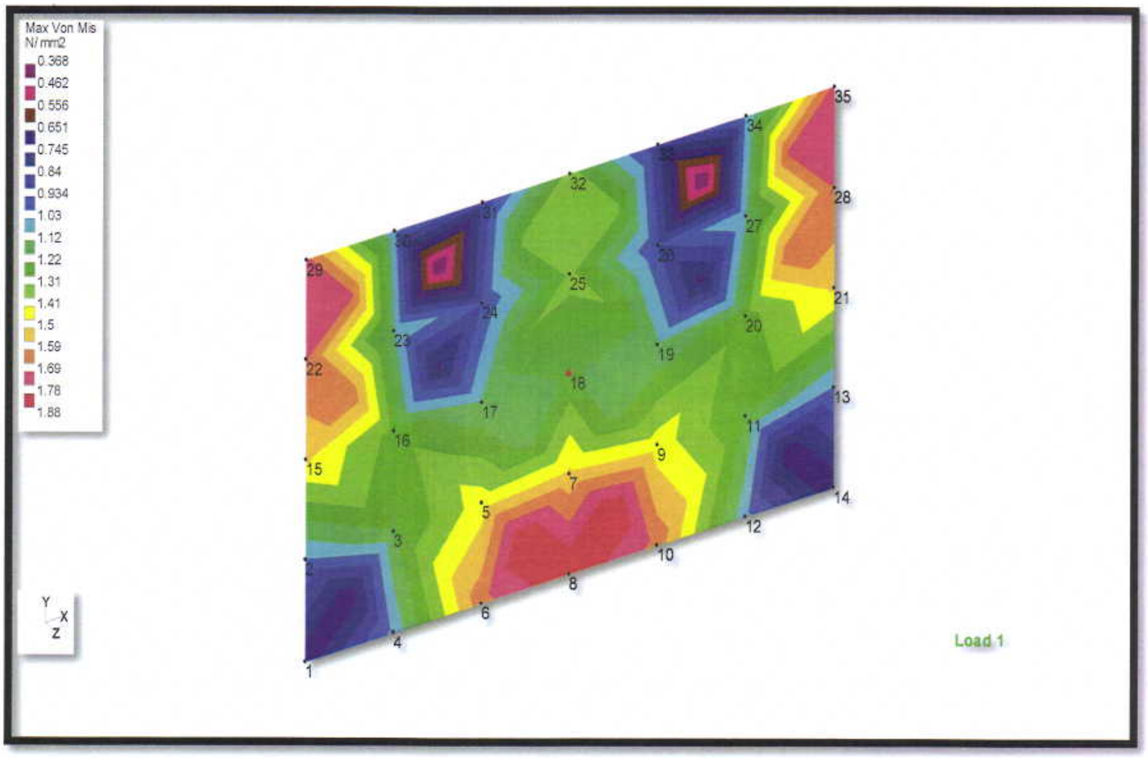
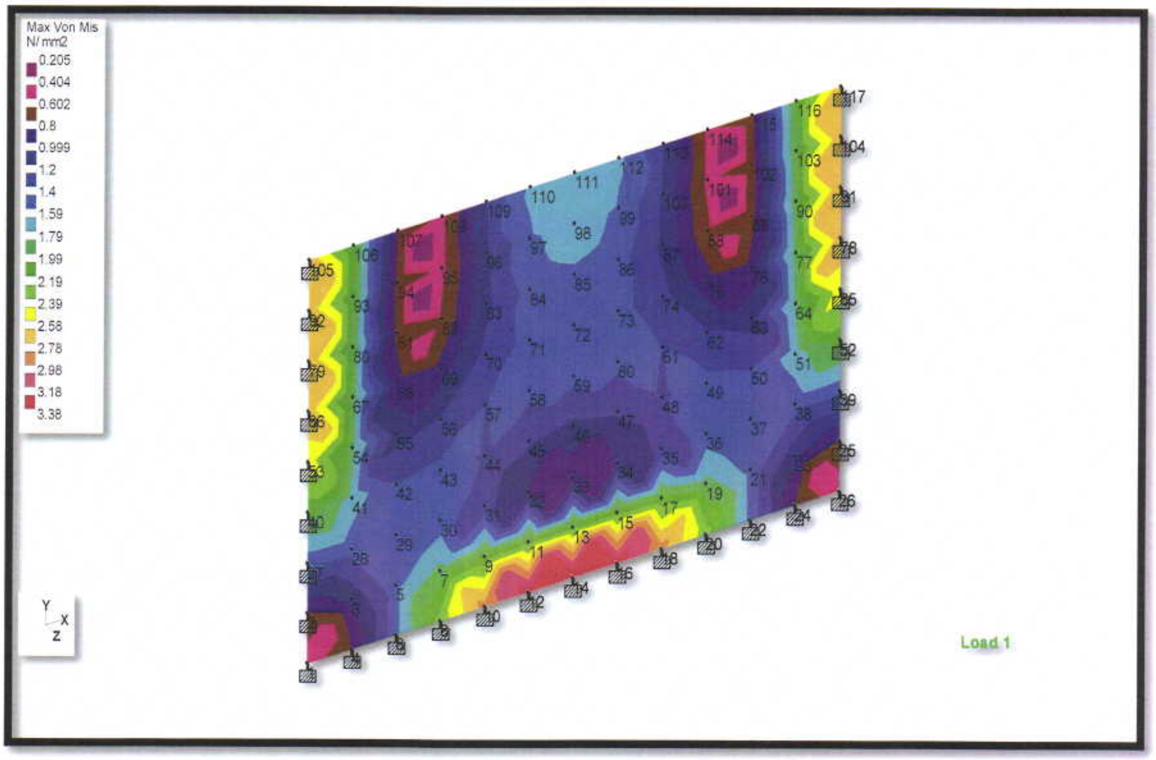


Fig.(4-2) Displacement in z-direction for ground tank with fixed support

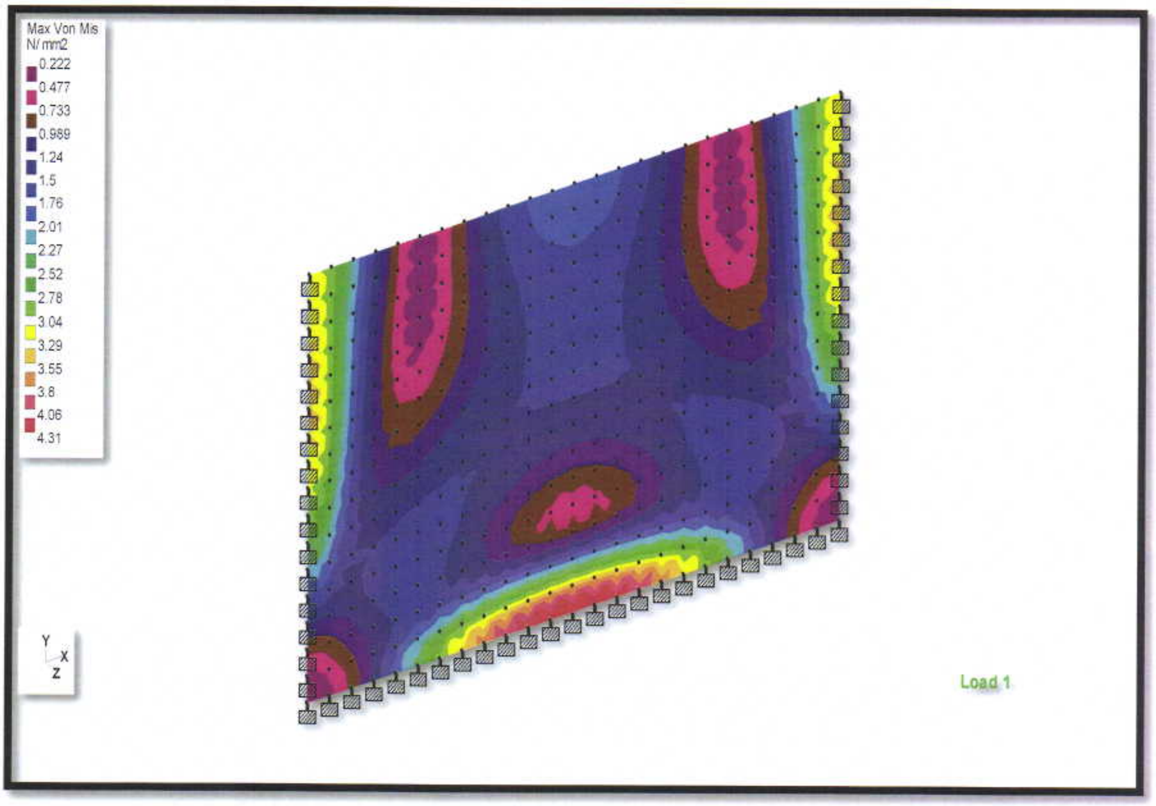


(A)



(B)

Fig.(4-3)Continued



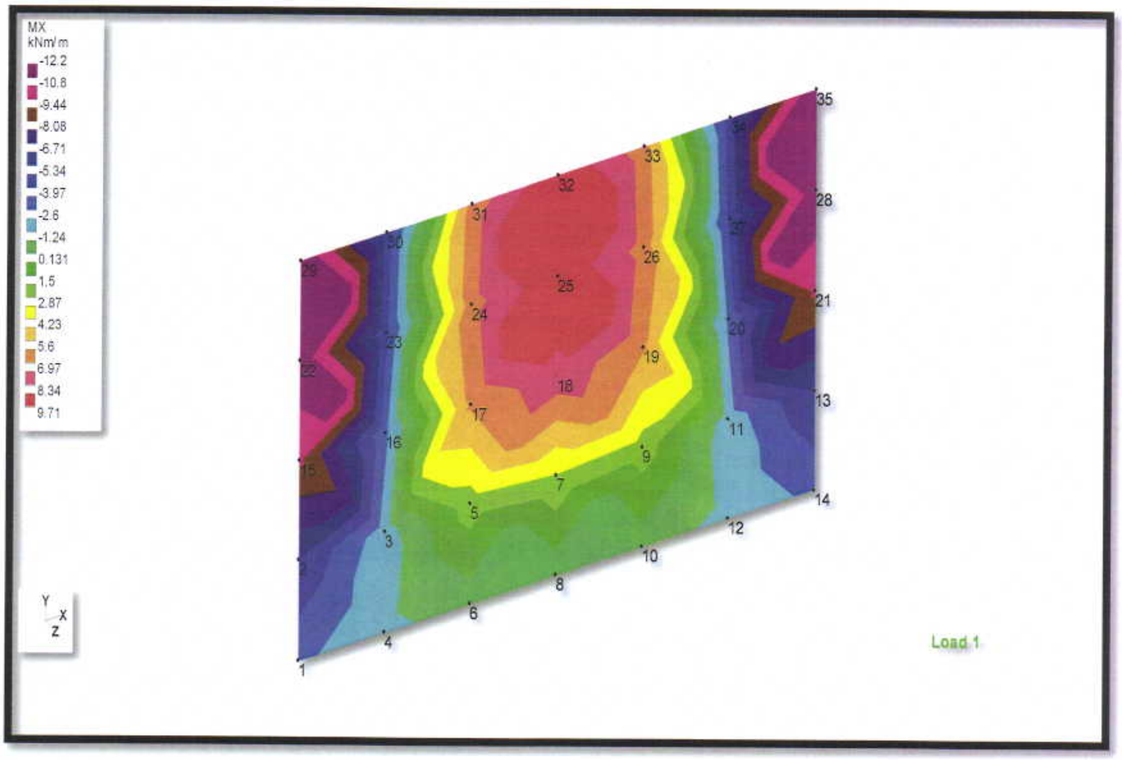
(C)

Fig.(4-3) Max Von Mis stress for ground tank with fixed support

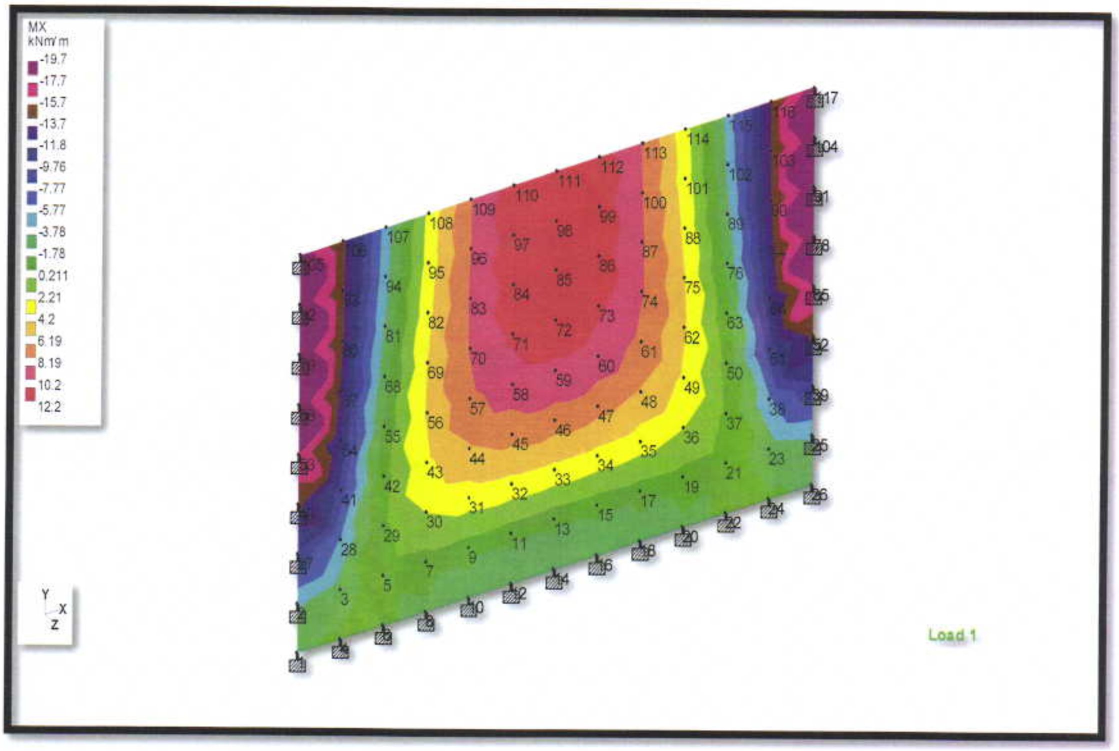
(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C) mesh 0.25x0.25m

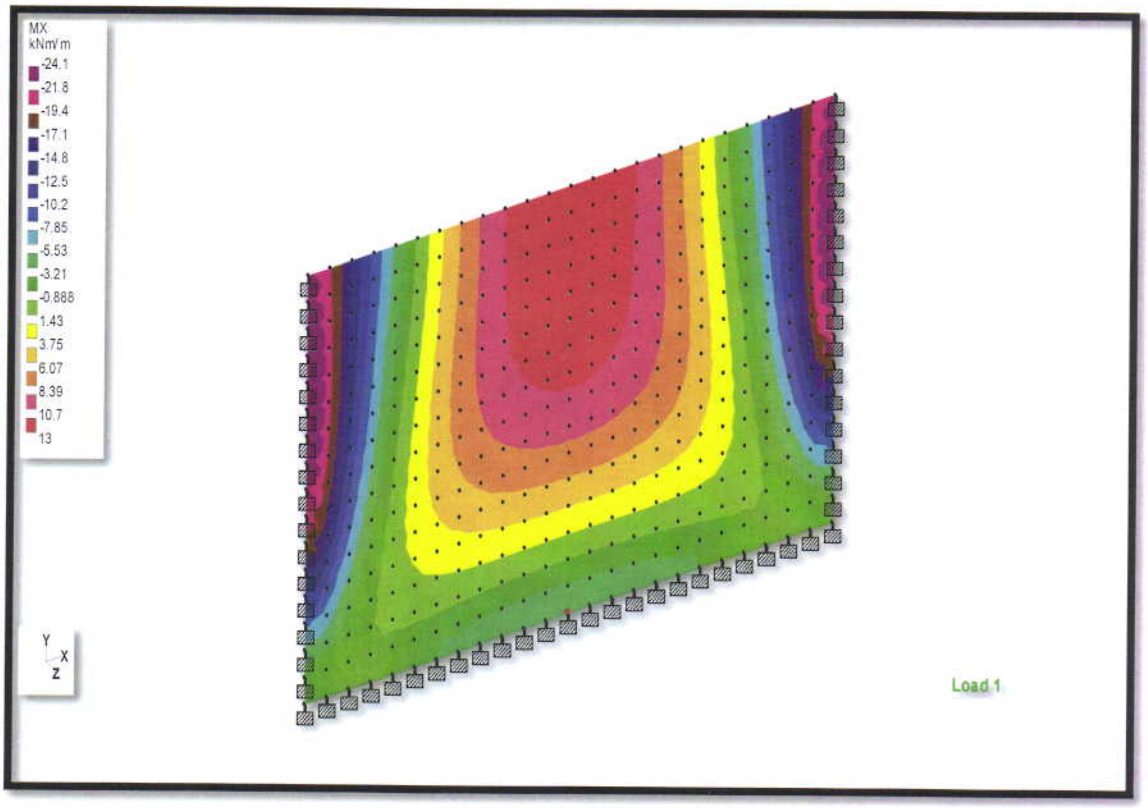


(A)



(B)

Fig.(4-4)Continued



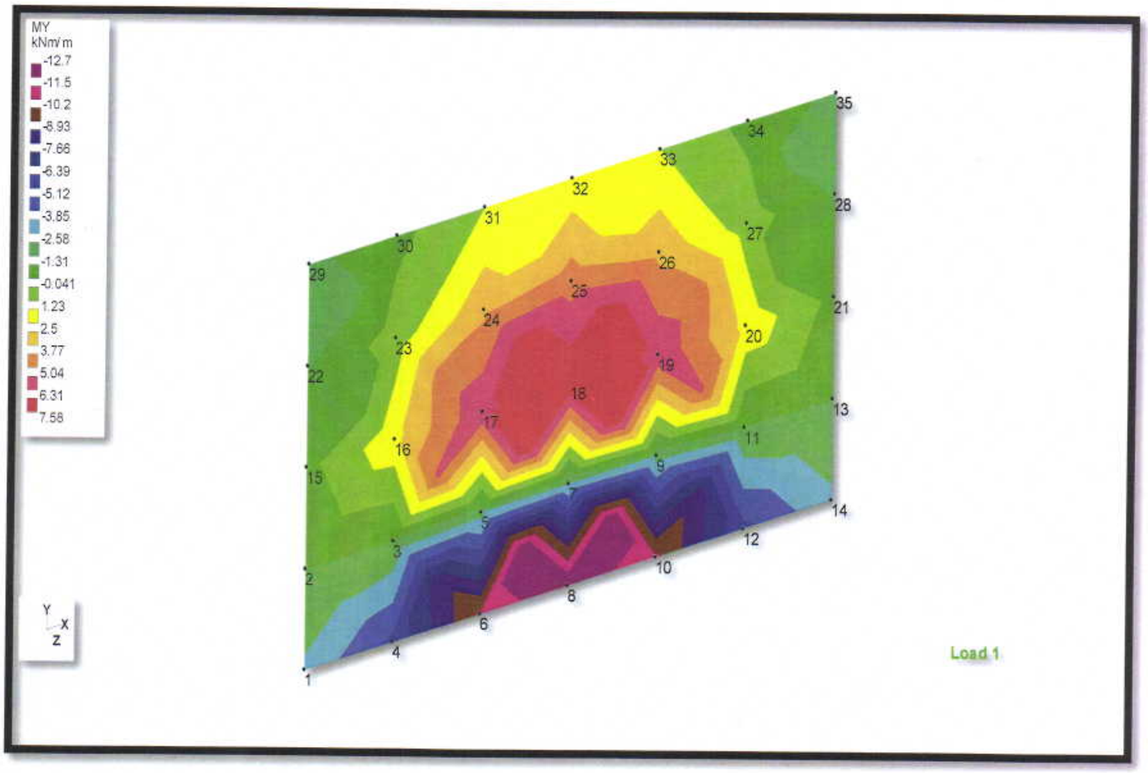
(C)

Fig.(4-4) Mx for ground tank with fixed support

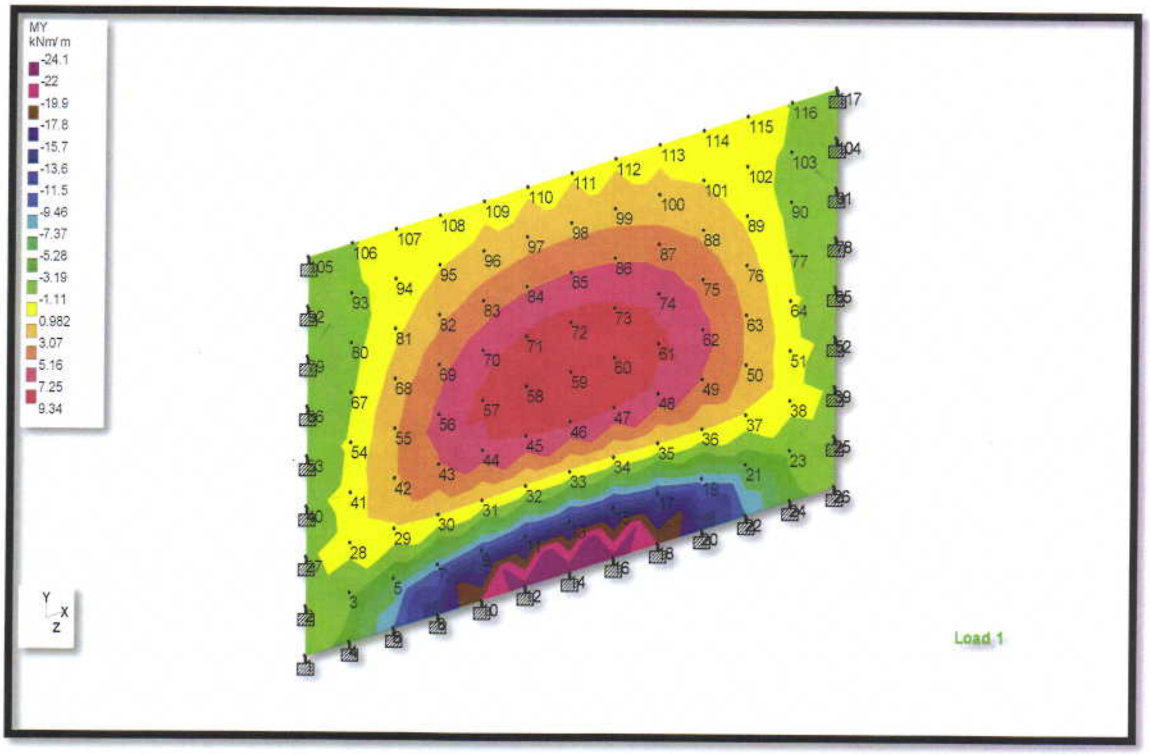
(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C) mesh 0.25x0.25m

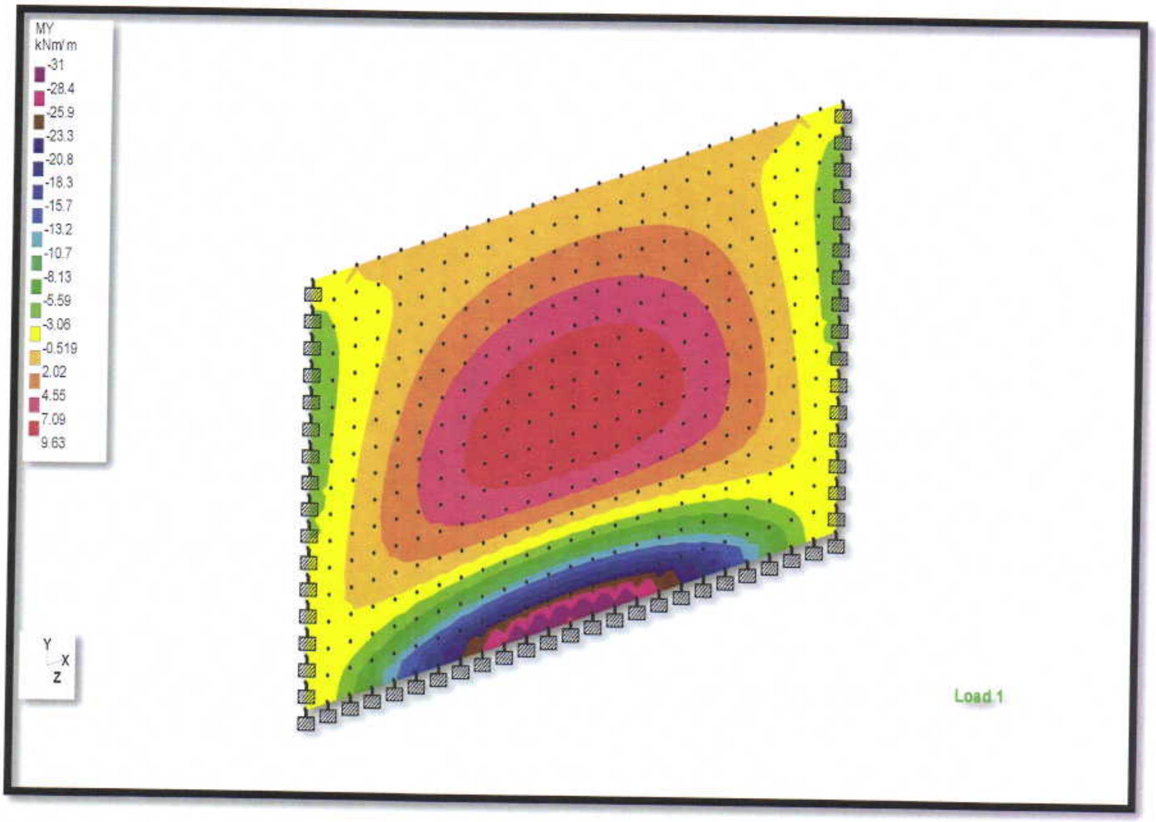


(A)



(B)

Fig.(4-5)Continued



(C)

Fig.(4-5) My for ground tank with fixed support

(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C) mesh 0.25x0.25m

4-3 Pinned Support

Case1 : mesh (1x1)m

Table.(4.4)Displacement in z-direction for ground tank with 1x1 mesh

Node No.	Displacement (mm)
8	0.000
7	-3.598
18	-6.002
25	-7.354
32	-8.396

Case2 : mesh (0.5x0.5)

Table.(4.5)Displacement in z-direction for ground tank with 0.5x0.5 mesh

Node No.	Displacement (mm)
14	0.000
13	-2.084
33	-3.933
46	-5.455
59	-6.636
72	-7.530
85	-8.224
98	-8.826
111	-9.450

Case3 : mesh (0.25x0.25)

Table.(4.6)Displacement in z-direction for ground tank with 0.5x0.5 mesh

Node No.	Displacement (mm)	Node No.	Displacement (mm)
26	0.000	238	-7.299
25	-1.084	263	-7.733
63	-2.129	288	-8.117
88	-3.119	313	-8.463
113	-4.023	338	-8.787
138	-4.848	363	-9.101
163	-5.584	388	-9.420
188	-6.234	413	-9.757
213	-6.803		

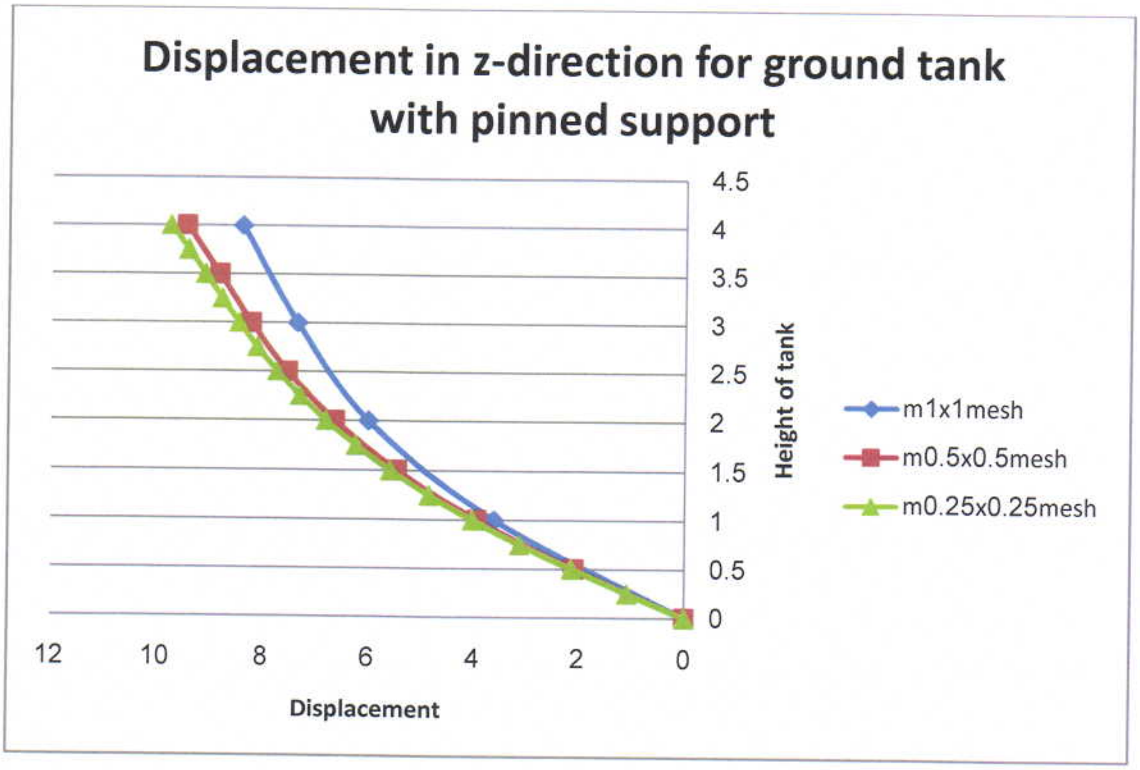
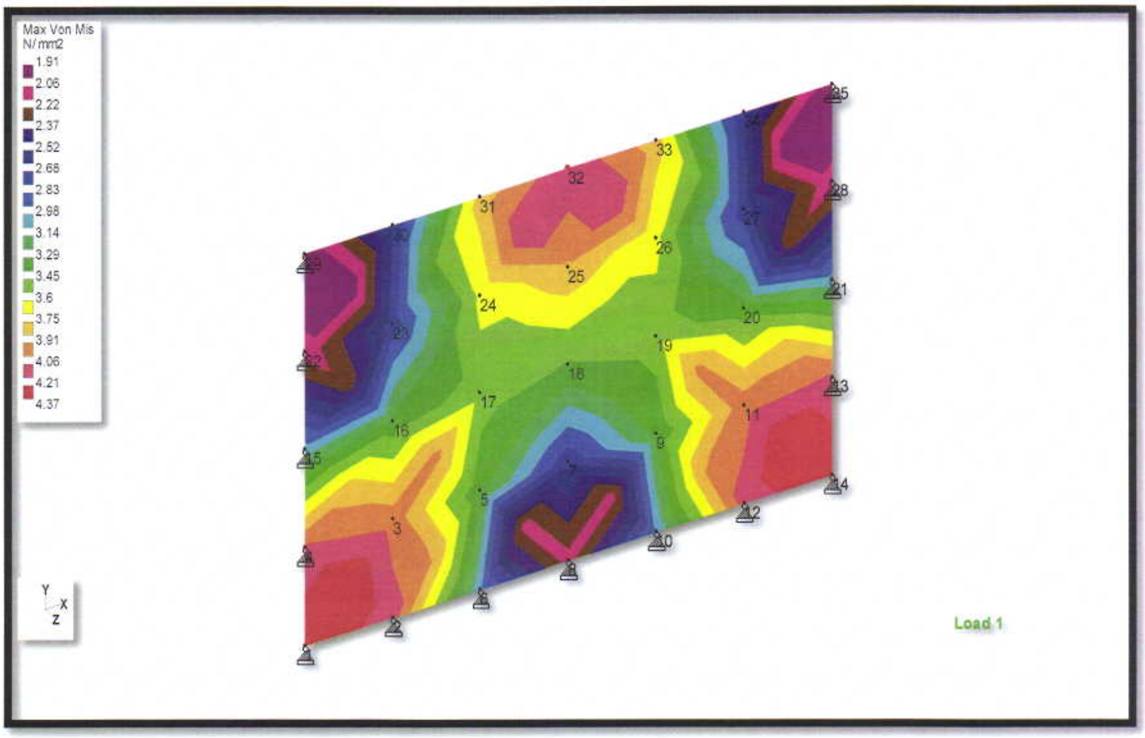
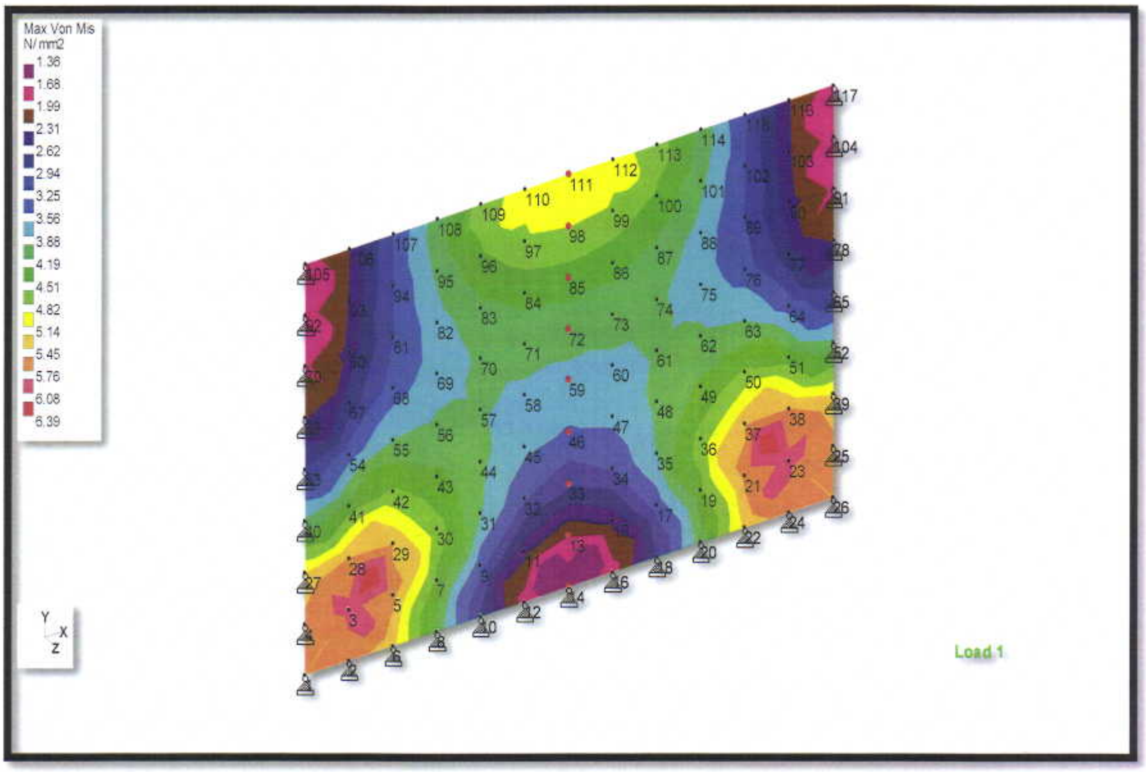


Fig.(4-6) displacement in z-direction for ground tank with pinned support

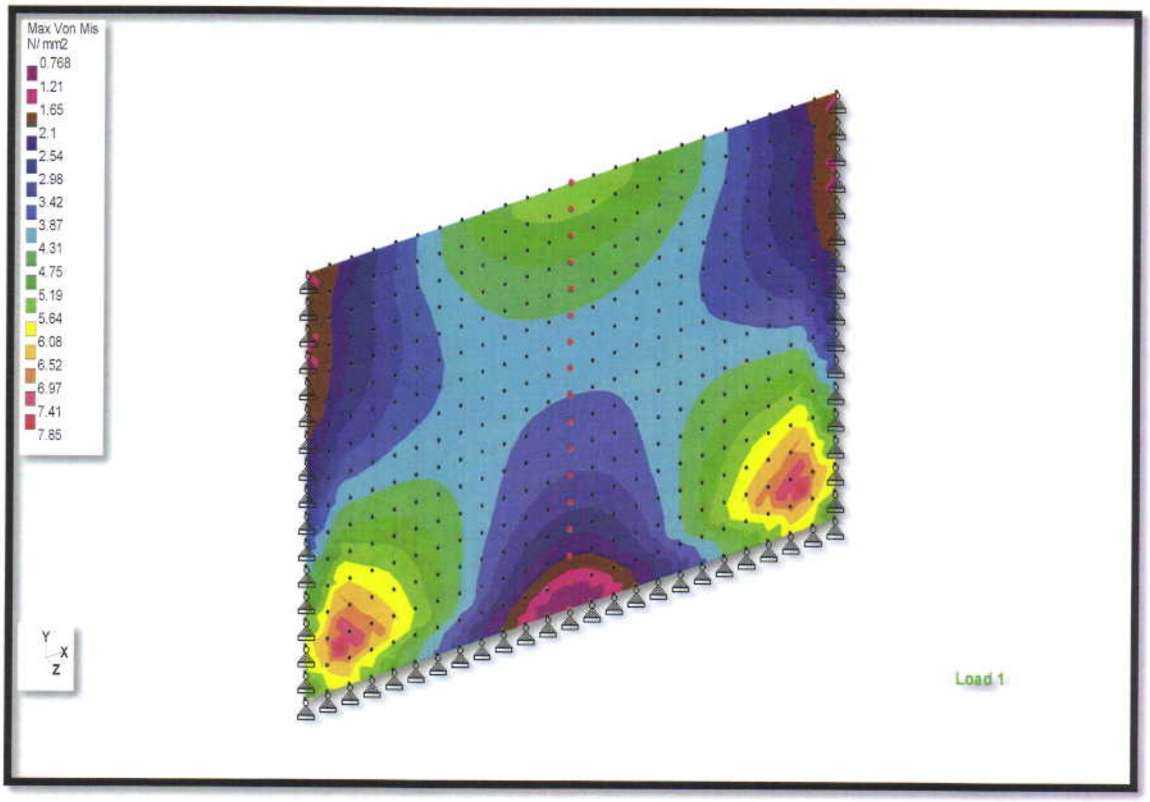


(A)



(B)

Fig.(4-7) Continued



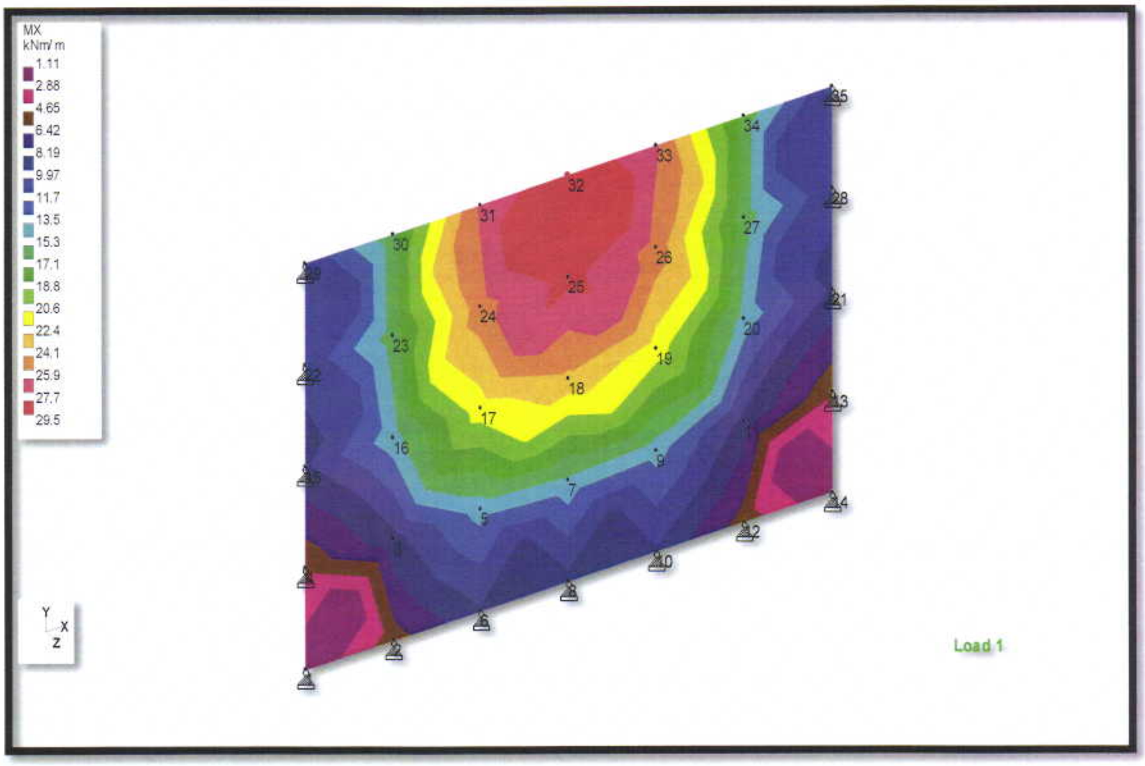
(C)

Fig.(4-7) Max Von Mis stress for ground tank with pined support

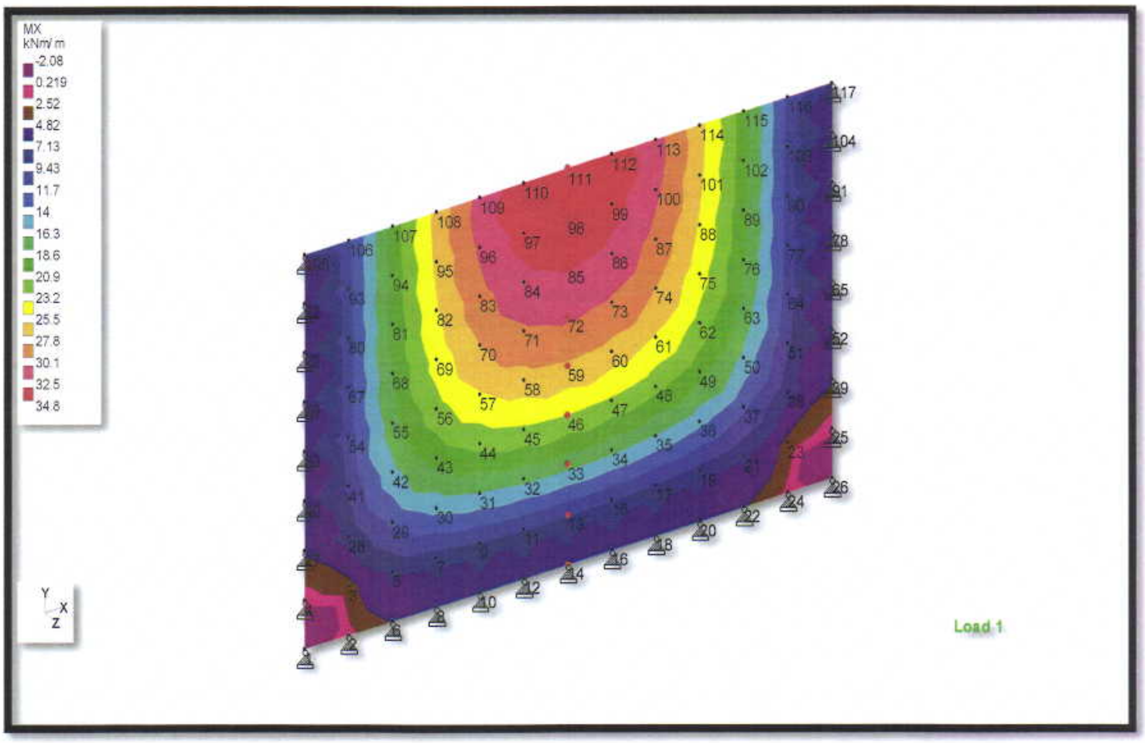
(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C) mesh 0.25x0.25m

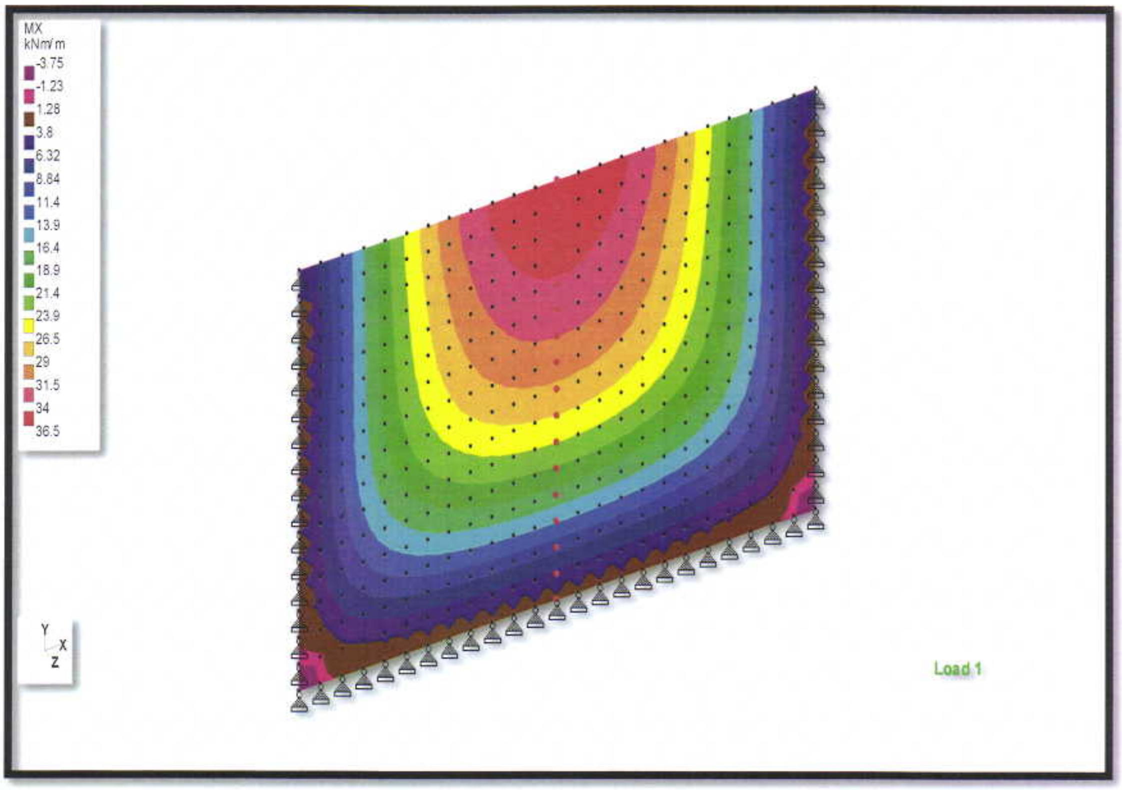


(A)



(B)

Fig.(4-8) Continued



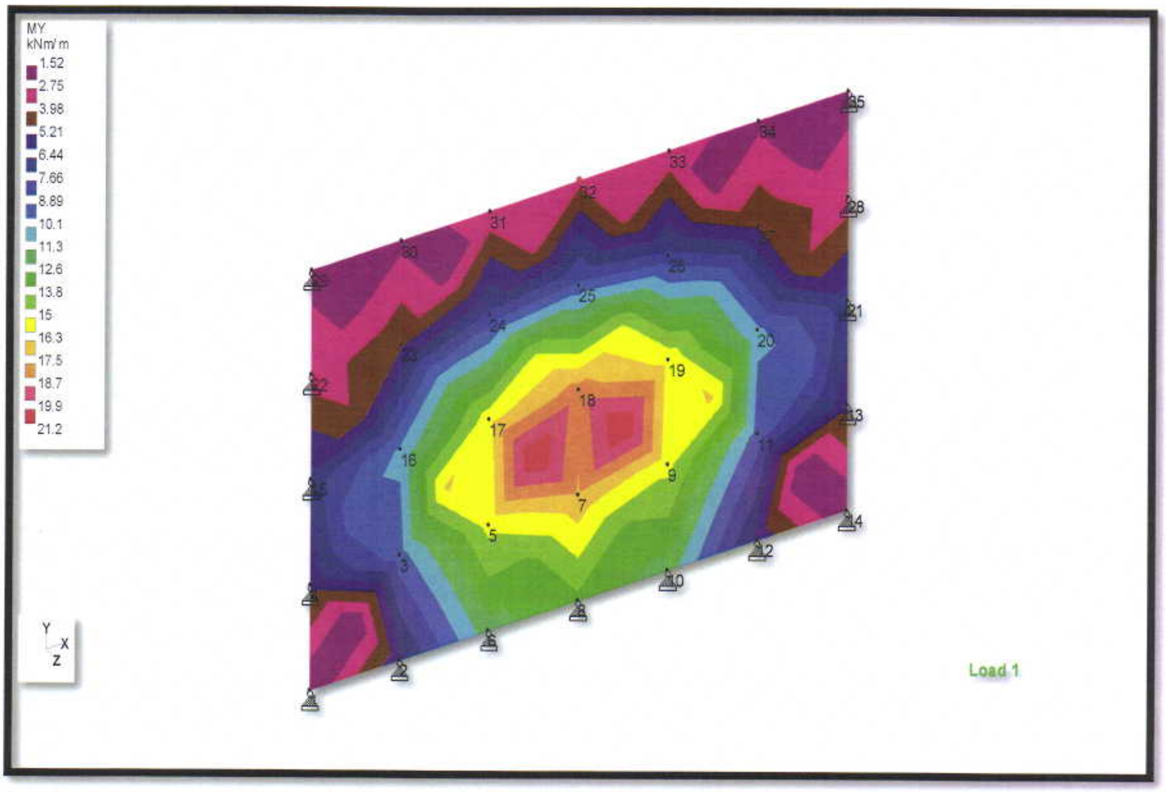
(C)

Fig.(4-8) MX for ground tank with pined support

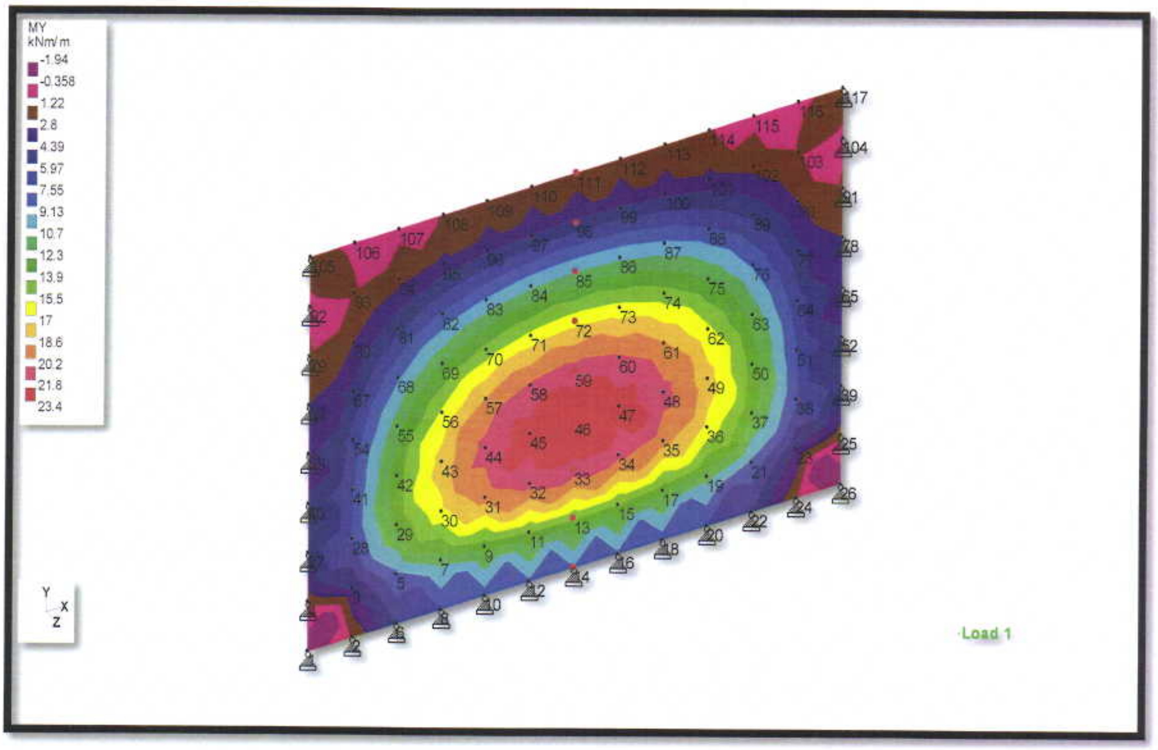
(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C) mesh 0.25x0.25m

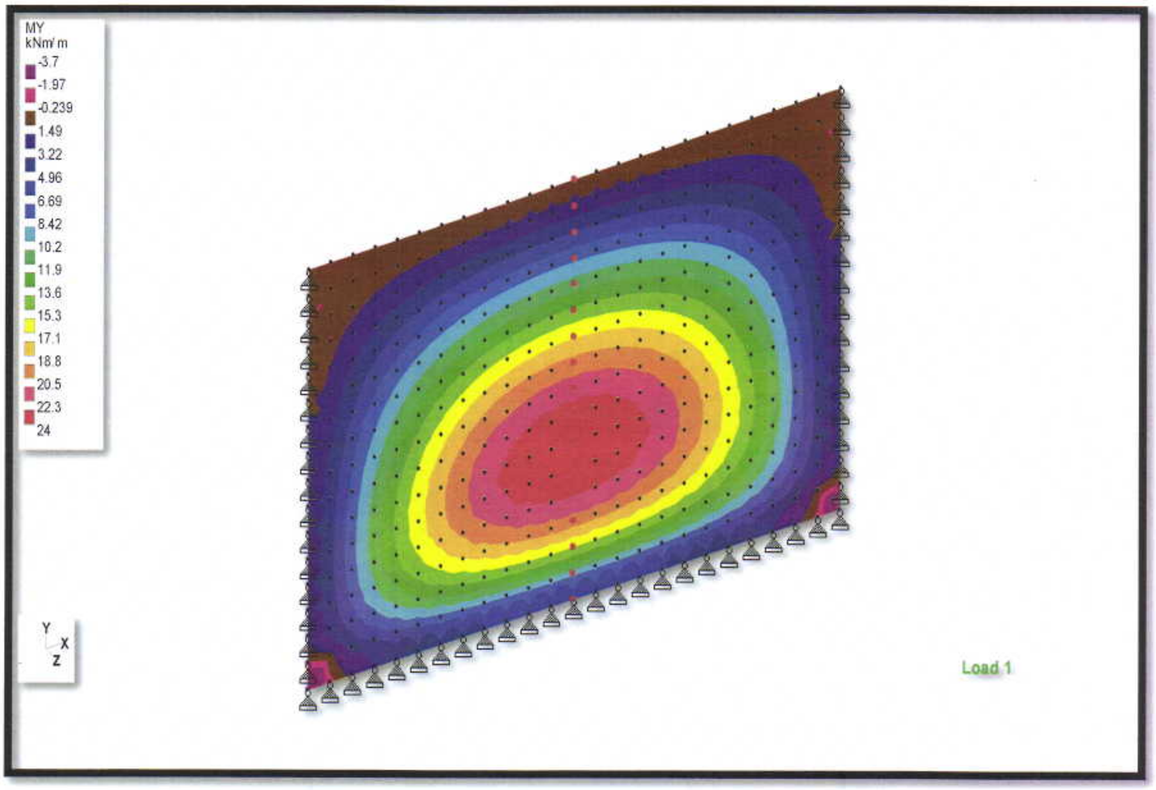


(A)



(B)

Fig.(4-9) Continued



(C)

Fig.(4-9) My for ground tank with pined support

(A) mesh 1x1m

(B) mesh 0.5x0.5m

(C)mesh 0.25x0.25m

Conclusions and Recommendations

5-1 Conclusions

Many methods are available to analyze tanks like finite element method, grillage method,....etc . This research has used a commercial package program, Staad Pro 2004 . This program utilize finite element method in solving for different types of structures. In this study a plate/shell element is used to model the side walls of tanks. Due to the present study a comparison is made to determine the behavior of the tank walls, specifically : displacements, stresses and bending moments.

1- For finer mesh sizes is seen that increase in displacements for fixed-supported walls of (21%) for 0.25m than that of 1m mesh size, respectively.

For pin-supported walls the ratio will become (14%) for 0.25m than that of 1m mesh size, respectively.

This proves that the design of the tanks using finite element method needs finer meshes to satisfy the requirement of the tank functionality .

2- The estimation of stresses is highly affected by type of boundary conditions. Also the bending moments in both directions (x,y) induced in the walls will give much different results.

The maximum von mises stresses for fixed supported walls are located at the mid-bottom edge of the wall, while the stresses are located in the lower corners of the wall, for pinned supported walls.

3- For both fixed supported and pin supported walls the maximum moments (Mx) located in mid-top edge, while the maximum moments (My) located

at the center of the wall.

4- The maximum moment (Mx) for pin supported is as larger as 2.8 times that of fixed supported walls.

5- The maximum moment (My) for pin support is as larger as 2.5 times that of fixed supported walls.

5-2 Recommendations

1- to analyze underground and/or elevated tanks using the Staad Pro program.

2- Taking in consideration the effect of flexibility of the connection lines between walls and slab.

3- Studying the behavior of the base slab on the ground.

4- Using another method like grillage method in the analysis.

ABSTRACT

Water storage tanks are very popular structure and frequently used in the civil engineering. The most important issue is to satisfy the structural and functional adequacy. In the design process, many methods are available to determine the material and geometric requirement.

This study has used staad pro. 2004 program which use finite element method. The plate/shell element is used to model the walls of the tanks.

The finite element method solution depends largely on the mesh size. It is noted that using finer mashes will lead to more accurate results. Fixed-supported walls idealization will give a 21% increase of (0.25 m) mesh size more than (1m) mesh size.

For pin-supported walls idealization, the maximum moment (M_x) is as large as 2.9 times that of fixed supported.

References

1- Reinforced concrete structure by I.C. syal and A.K. Goal, 1988 second edition, Page(1-6)
2- http://www.hacengineers.co.uk/pdfs/tanks/TANKSFINAL.pdf.com, Page(7-11)
3- Bairagi, N.K., Plate Analysis, Khanna Publishers, New Delhi, First Edition, 1986, Page(12-19)
4- Concrete liquid Retaining structures (1980) by J.Keith Green and Philip H. Perkins, Page(19-30)