Summer 2016

Global Sensitivity Analysis of Mat Foundation Behavior by Using Finite Element Modeling

Yang Zhao
Old Dominion University, yzhao012@odu.edu

GLOBAL SENSITIVITY ANALYSIS OF MAT FOUNDATION BEHAVIOR BY USING FINITE ELEMENT MODELING

by

Yang Zhao

B.S. July 2013, Xi’an University of Science and Technology, China

A Thesis Submitted to Faculty of Old Dominion University in Partial Fulfillment of the Requirement for Degree of

MASTER OF SCIENCE

CIVIL ENGINEERING

OLD DOMINION UNIVERSITY

August 2016

Approved by:

Reza Moradi (Director)

Isao Ishibashi (Member)

Duc Nguyen (Member)

ABSTRACT

GLOBAL SENSITIVITY ANALYSIS OF MAT FOUNDATION BEHAVIOR

BY USING FINITE ELEMENT MODELING

Yang Zhao
Old Dominion University, 2016
Director: Dr. Reza Moradi

Mat foundation is used mostly for high-rise buildings and, since the demand for high-rise buildings is growing, having extensive understanding of mat foundation behavior leads us into more efficient structural design. Mat foundation behaviors are affected mostly by soil's materials properties, foundation size, and thickness and loading conditions. In this study, the sensitivity of mat foundation structural responses to some of the design parameters are evaluated. Sobol decomposition, which is a variance-based technique, is used to perform the global sensitivity analysis. In this study, Patran, a finite element-based software, is used for modeling and simulations.

Approximate flexible method is a basic method which is used to calculate the structural response of a mat foundation. In this study, the responses of the mat foundation are verified by regenerating the ACIC 336 figure, which is used in approximate flexible method. After verification of Patran model responses, the global sensitivity analyses are performed.

"Modulus of elasticity of soil", "Modulus of elasticity of foundation material", "Column load", "Foundation aspect ratio", and "Foundation thickness" are used as the input variables and "Maximum deflection of the mat foundation" is used as the output response. The results show how the foundation deflection is affected by the design variables.

This thesis is dedicated to my family.

ACKNOWLEDGMENTS

The author would like to thank the thesis director, Dr. Reza Moradi. He helped the author throughout the thesis. When the author had any problem regarding the thesis, he was always there to help the author and answer the author's questions.

The author would like to thank the committee members, Dr. Isao Ishibashi and Dr. Duc Nguyen, for their time in serving on the committee.

Also, the author would like to thank his parents and friends. Without their encouragement and support, the author could not have finished his thesis.

Thanks again to everyone who has helped the author.

NOMENCLATURE

A_1, A_2, A_3, and A_4	Functions of r/L
cv	Coefficient of variation
E_F	Modulus of elasticity of foundation material
I_F	Moment of inertia of the cross section of the beam
k	Modulus of subgrade reaction
M	Moment at any section
Q	Column load
q	Stresses
r	Radial distance from the column load
μ	Mean value
μ_F	Poisson's ratio of foundation material
w	Deflection of mat
z	Deflection of soil

TABLE OF CONTENTS

Page

Page

LIST OF TABLES

LIST OF FIGURES

Figure Page

Figure Page

Figure Page

CHAPTER 1

INTRODUCTION

1. Mat foundation

Mat foundation, also called raft foundation, is a thick reinforced concrete slab covering the entire area of the bottom of a structure, like a floor. The slab is reinforced with bars running perpendicular to each other, both near the bottom and the top face of the slab.

Mat foundations provide an economical solution under certain difficult site conditions and are commonly used in the following cases:

(a) When soils have low bearing capacity and spread footings are impractical and not sufficient to transfer structural loads to the ground.

(b) When a pile foundation cannot be used advantageously.

(c) When the structure is vulnerable to subsidence because it is located in the mining area, or because of uncertain behavior of its sub-soil water conditions.

Mats can bridge across weak pockets in a non-uniform foundation. Therefore, mat foundations have various advantages that are summarized as following:

(a) The raft can be used as a basement floor;

(b) The flexural stiffness can be used to reduce differential settlements, due to shrinking and swelling of active soils;

(c) The flexural stiffness can be used to reduce contact pressures in regions of higher soil compressibility.

The total settlement can be reduced by using a mat foundation combined with piles. For deep basements, mat foundations are also widely applied to spread the column loads to a more uniform pressure distribution, as well as to provide the floor slab for the basement. If the basement is located at or below the ground water table, it can provide a water barrier. Because of all of these advantages, in recent years, mat foundations are becoming increasingly popular.

There are five main types of mat foundation:

1) Flat plate
2) Flat plate thickened under column
3) Beams and slab
4) Flat plate with pedestal
5) Slab with basement walls as a part of the mat.

The most common type of mat foundation is the flat plate. This type of foundation tends to be heavily over-designed due to uncertainly in analysis (Bowels, J.E, 1997).

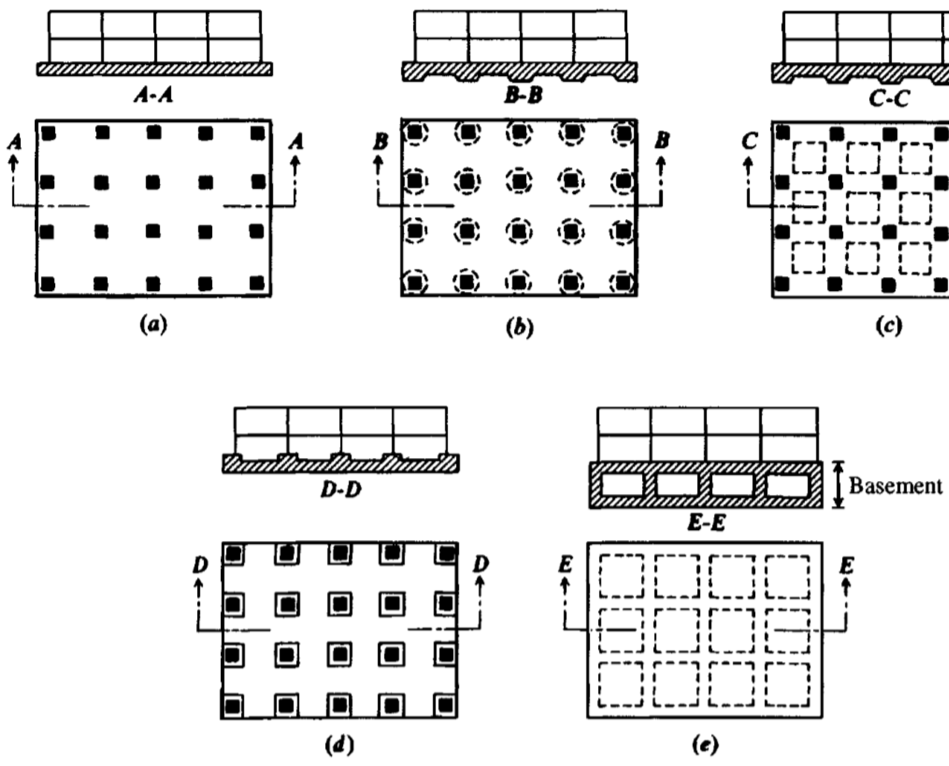


Fig 1.1 Five types of mat foundations used for buildings construction. (a) Flat plate, (b) Flat plate thickened under column, (c) Beams and slab, (d) Flat plate with pedestal, (e) Slab with basement walls as a part of the mat (Bowels, J.E, 1997).

1.2 Finite Element Analysis Method

The finite element method (FEM) is a numerical technique used for finding approximate solutions to boundary value problems for partial differential equations. It is also referred to as finite element analysis (FEA). FEM subdivides a large problem into smaller, simpler parts, called finite elements. The simple equations that model these finite elements are then assembled into a larger system of equations that models the entire problem. FEM then uses variational methods from the calculus of variations to approximate a solution by minimizing an associated error function (Zienkiewicz, O. C., 1968).

There are various theories for the design of a mat foundation. Finite element method is one of the effective and economical numerical methods for analyzing these foundations. FEM includes the use of mesh generation techniques for dividing a complex problem into smaller elements. In applying FEA, the complex problem is usually a physical system with the underlying physics, such as the Euler-Bernoulli beam equation, the heat equation, or the Navier-Stokes equations expressed as either Partial Differential Equations (PDE) or integral equations, while the divided small elements of the complex problem represent different areas in the physical system.

For mat foundation, advanced numerical modeling techniques are utilized by dividing the Mat into grid elements and predicting the behavior of the structure under given loading. The Finite Element Software analysis has been extensively used in most of the previous studies for modeling different structural elements, such as foundations.

The typical steps for Finite Element Modeling include:

1) Dividing the domain of the problem into a collection of subdomains, with each subdomain represented by a set of element equations to the original problem

2) Systematically recombining all sets of element equations into a global system of equations for the final calculation. The global system of equations has known solution techniques and can be calculated by the initial values of the original problem to get a numerical answer (Sutradhar, A., 1999).

CHAPTER 2

LITERATURE REVIEW

2.1 Previous Study on Mat Foundation Modeling

The mechanical response of naturally occurring soils is influenced by a variety of factors such as the shape, the size, and the mechanical properties of the individual soil particles, the configuration of the soil structure, the intergranular stress history and the presence of soil moisture, the degree of saturation and the soil permeability. These factors mostly impact the stress-strain relationship and create non-linear, irreversible and time dependent characteristics. In order to obtain useful and reliable information for practical problems, it is necessary to model soils for idealizing the behavior of soil (Srilakshmi, G., 2011).

There are several methods to analyze mat foundation:

a) Conventional method (Tang, Z.S., 2012)

b) ACI approximate flexible method

c) Baker's method (Hussin, J., 2006)

d) Finite difference method

e) Finite grid method (Bowels, J.E, 1997)

Among the five methods, only conventional method treats mat as a rigid slab; all of the other methods treat mat as a flexible slab using the Winkler Model. The ACI method is based on Schleicher's infinite flexible slab on continuous spring support solution, and Baker's method was designed to simulate that solution. Finite difference and finite grid methods are numerical analysis methods and require extensive computer support, although though they do not offer any substantial improvement in results over Schleicher's solution (Yamshita, K., 1998).

Some of the main previous studies which include the modeling of mat foundation are presented in the rest of this section.

2.1.1 Winkler Model

The idealized model of soil-structure interaction was proposed by Winkler (1867). In this method, reactions from foundation soil are modeled by a composition of independent vertical springs. When a concentrated load is applied on the surface, the model does not only deflect under the load, but also deflects with displacements diminishing with distance in the adjacent area. The model assumes that deflection w, of the soil medium at any point on the surface has a proportion with the stress, q applied at that point and independent of stresses applied at other location, where k is the modulus of sub grade reaction with units of stress per length. The Winkler Model considers that the soil medium consists of a system of multiple independent spring elements with constant k.

The Winkler Model has an important feature, though: the displacement occurs only under a loaded area; outside this region, the displacement is zero. The displacements of a loaded region for this model are constant when the region is subjected to either an infinitely rigid load or a uniform flexible load (Ahmad, S., 1969).

2.1.2 Vlasov and Leont'ev Model

The Vlasov and Leont'ev Model (1992) is an improved subgrade model and has the same basic differential equation as other several subgrade models. But it has the lowest order in calculation, which has the simplest approximation of the series solution compared to others. So the Vlasov and Leont'ev Model is easier to incorporate into a soil-structure interaction computer code and gives a more accurate solution than other subgrade models (Vlasov, V.Z. and Leonitiev, N.N, 1960).

2.1.3 AMPS 3D Finite Element Method Model

In the AMPS 3D Finite Element Method (FEM) (2004), the structural element of the mat and piles is modeled and the interface between the structural components and soil is simulated. Linear or nonlinear soil properties are employed to achieve a satisfactory solution.

There are two main reasons to use the 3D FEM model:

(1) If the problem is complex and simplified methods cannot be used

(2) If codes for the FEM are available and capable of being run on the PC.

When the resistance of pile to axial loading depends on the lateral stress in the soil, the prediction of the influence of the vertical stress from the mat on the lateral stress at every point along the length of each of the piles for all ranges of loading is beyond the capability of any simplified method. However, the 3D FEM can model not only that, but also the piles as a reinforcement of soil, which influences the behavior of the raft (Ukarande, G.S., 2008).

2.1.4 YS-MAT Model

The YS-MAT Model (2015), as an improved analytical method of the mat foundation design, uses Pasternak's shear layer model in order to overcome the restrictions of a conventional Winkler model. The model is intermediate in complexity and its theoretical accuracy is between the Winkler and the continuum models. The continuum model uses the theory of elasticity to analyze semi-infinite, homogeneous, and isotropic problems. A linear elastic solid subjected to concentrated force acts normal to the plane boundary. This model provides much more information on the stresses and deformations within a soil mass than Winkler model does. Also the time factor, both in modelling and computation, can be exhausting (Rana, S. C., 2002).

YS-MAT was proposed on the basis of modeling mat flexibility and coupled soil springs. It can calculate the displacements, bending moments, and shear forces on a mat foundation for a given condition, such as the geometry, the loads, the properties of the mat foundation, and stiffness of the soil springs (Fig. 2.2).

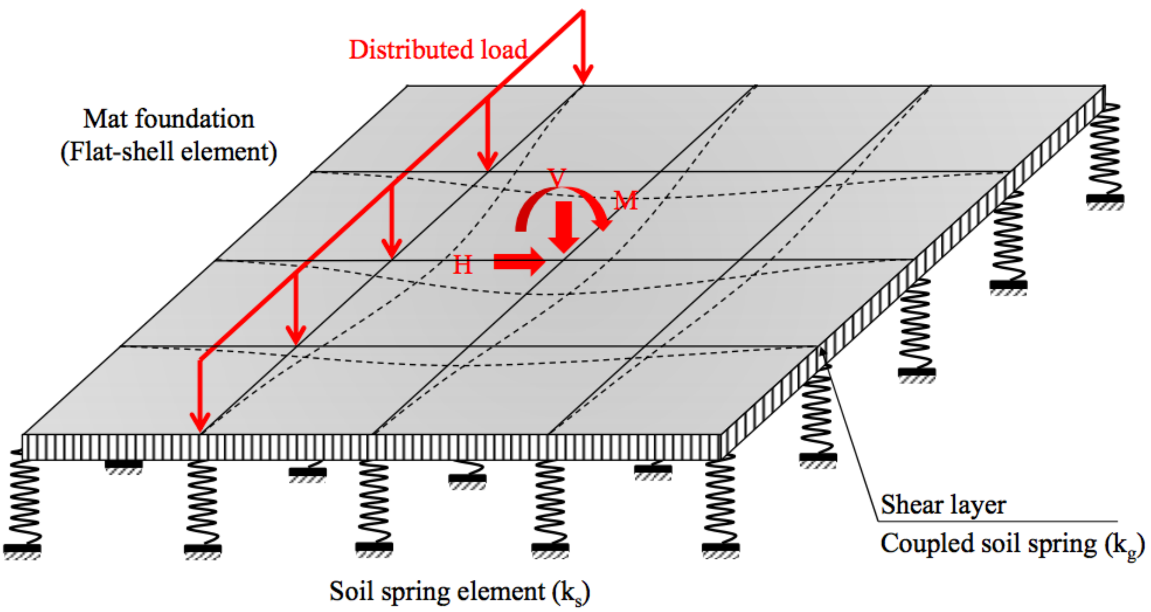


Fig.2.2 Idealized 3D model for mat foundation used in YS-MAT (Lee, J., 2015)

The YS-MAT is capable of predicting the settlement of a large mat foundation. The membrane action of the flat shell elements and the soil coupling effect can overcome the limitations of the conventional method. The YS-MAT can be used in the preliminary design of a large mat foundation (Lee, J., 2015).

2.2 Sensitivity Analysis

The main objective of this study is to evaluate the sensitivity of mat foundation behavior to soil and foundation parameters. "The study of how uncertainty in the output of a model can be apportioned to different source of uncertainty in the model input" is called sensitivity analysis (Saltelli et al., 2004). In other words, one should first define the system and input and output variables and then find the sensitivity of outputs to the inputs. There are two methods of sensitivity analysis: the local method and the global method (Salteli, A., 2004).

2.2.1 Local Method

In the local method, the variables are varied one at a time by a small amount around some fixed point. Then, the sensitivity of the outputs is measured by taking a partial derivative of the output variables to the input variables around that fixed point.

Here are the disadvantages of the local method:

- It cannot evaluate the sensitivity of output to inputs over the whole input domain. It can find the sensitivity around only one specific point.
- If there is nonlinearity between output and input, this method cannot capture that, because this method is based on finding the slope of output variables versus input variables. So it assumes a linear relationship between output and inputs.
- Since these methods are based on the variation of inputs one at a time, it cannot measure the effect of interaction between parameters on output variables.

2.2.2 Global Method

In the global method, all variables are varied simultaneously through their entire feasible space or through the user-selected range. In this method, the effect of individual variables and also the interaction between them on output are assessed. This method can also capture the nonlinearity between input and output. Therefore, global sensitivity analysis methods are more powerful than local methods, but they are more expensive because of the higher computational time compared to local methods. In this study, Sobol decomposition is used as a technique to perform the global sensitivity analysis (Salteli, A., 2008).

2.2.2.1 Sobol Decomposition

Sobol Decomposition is one of the variance-based decomposition technique in which the variance of the output function can be written as a summation of the variance of all of the parts of the output function after decomposition. Assume F(X) as an output function in which X K^n is an input vector and contains “n” uniformly distributed variables between 0 to 1.

$$K^n = \{X : 0 \leq X_i \leq 1, i = 1, \ldots, n\} \qquad (\boldsymbol{Eq. 2.2})$$

The Sobol decomposition of F(X) is given by

$$F(X) = F_0 + \sum_{1 \leq i \leq n} F_i(X_i) + \sum_{1 \leq i < j \leq n} F_{ij}(X_i, X_j) + \cdots + F_{12\ldots n}(X_1, \ldots, X_n) \qquad (\boldsymbol{Eq. 2.3})$$

This decomposition splits the main function into mean value, functions with one variable, functions with two variables, and so on. These functions are call Sobol functions. In order to have the variance of the output function as a summation of the variance of all the Sobol functions after decomposition, these functions should be orthogonal, which means:

$$\int_{K^n} F_{i_1,i_2,\ldots i_s}(x_{i_1}, x_{i_2}, \ldots x_{i_s}) F_{j_1,j_2,\ldots j_t}(x_{j_1}, x_{j_2}, \ldots x_{j_t}) \mathrm{d}x = 0$$

$$(\boldsymbol{Eq.\,2.4})$$

When at least one of the subscripts on the two functions are composing the integrand differently.

By considering this condition, Sobol functions are calculated as:

$$(\boldsymbol{Eq.\,2.5})$$

$$F_0 = \int_{K^n} F(x)\mathrm{d}x$$

$$F_i(X_i) = \int_{K^{n-1}} F(x_{\sim i}, X_i)\mathrm{d}x_{\sim i} - F_0 \qquad (\boldsymbol{Eq.\,2.6})$$

$$F_{ij}(X_i, X_j) = \int_{K^{n-2}} F(x_{\sim ij}, X_i, X_j)\mathrm{d}x_{\sim ij} - F_i(X_i) - F_j(X_j) - F_0$$

$$(\boldsymbol{Eq.\,2.7})$$

Where F_0 is the mean value of the output function, $x_{\sim}i$ is the vector of dummy variables corresponding to all but the component Xi of the input vector X, $dx_{\sim}i$ means integration with respect to all variables except Xi. Similarly, higher order Sobol functions can be calculated.

The variance of F(X) can be decomposed according to

$$D = \sum_{1 \le i \le n} D_i + \sum_{1 \le i < j \le n} D_{ij} + \cdots + D_{12\ldots n}$$

$$(\boldsymbol{Eq.\,2.8})$$

In which

$$D = \mathrm{var}[F(X)] = \int_{K^n} F^2(X)\mathrm{d}X - F_0^2 \qquad (\boldsymbol{Eq.\,2.9})$$

$$D_i = \int_{K^1} F_i^2(x_i)\mathrm{d}x_i \qquad (\boldsymbol{Eq.\,2.10})$$

$$D_{ij} = \int_{K^2} F_{ij}^2(x_i, x_j)\mathrm{d}x_i\mathrm{d}x_j$$

$$(\boldsymbol{Eq.\,2.11})$$

The Sobol indices are defined as

$$S_i = \frac{D_i}{D} \qquad (Eq. 2.12)$$

$$S_{ij} = \frac{D_{ij}}{D} \qquad (Eq. 2.13)$$

and the summation of all the Sobol function is 1.

$$\sum_{1 \le i \le n} S_i + \sum_{1 \le i < j \le n} S_{ij} + \cdots + S_{12\ldots n} = 1 \qquad (\mathbf{Eq. 2.14})$$

The higher the Sobol indices of given input variables, the higher the sensitivity of the output function to that input variable (Salteli, A., 2004).

CHAPTER 3

MODELING METHOD

Two modeling methods, “Patran Modeling” and the “Approximate Flexible Method”, are used in this study. The details of the two methods are discussed in this chapter.

3.1 Patran Modeling

Virtual Product Development (VPD) is an approach that takes a design at its earliest concept stage and fully evaluates design specifications and usage scenarios, and then uses this information to guide the development process.

The main point of the VPD process is its capability to represent physical environments and events in a product design by using simulation software. Simulation is begun by building a model of the product structure. After that, specialized finite element analysis (FEA) codes are used to analysis simulation model. The simulation model is used to analyze how it responds to the assumed environment.

Patran is the software that can manage and carry out several phases of the VPD process. It can be used to build a model of a product, to simulate an environment, to manage finite element analysis, and to interpret numerical results. Patran can complete all of the simulation tasks by itself, and it can also be used in conjunction with other CAD software, modeling packages, and analysis codes.

3.1.1 Steps for Simulation project

The main steps to construct a design model in Patran are as follows:

1) Turning product designs into geometry models: Patran provides a set of tools for geometry creation and editing. There are more than hundred options of the geometry tools which can create parts with two- and three- dimensional wireframe, surface, and solid geometry. Patran's CAD interface can also import and edit CAD data from many leading CAD programs.

2) Meshing and creating elements: After importing or creating the geometry, Patran can create and verify the finite element mesh using various meshing tools such as the industry leading automeshers for curves, surfaces, and solids. Patran also allows interactive editing of the models.

3) Modeling materials: This step can define the materials for the analysis model in Patran. The Material model describes what the model is made of (such as steel or a composite) and how the attribute properties of that material are (stiffness, space density, and so on).

4) Simulating forces and loads: Patran analyzes the reaction of a particular model for particular loads and boundary conditions. It simulates the force, pressure, temperature, and voltage, the environmental factors which represent the loads. Boundary conditions can also be defined by Patran.

5) Analyzing the model with environmental loads: Once the product design model is completed, the analysis stage begins. Patran has several options for running a finite element analysis such as MSC's analysis codes, an outside commercial code, or an in-house proprietary code. In each case, several tasks are completed by Patran to format and set up the analysis.

6) Tailoring the model for a selected analysis code: Patran defines element types and element-related properties for regions of the model, and then it uses these for geometric or FEM entities. Selection of element type is affected by the finite element code, the dimensions of the model, and assumptions about the model's behavior.

7) Running a finite element analysis: The analysis application provides a link between the Patran environment and the analysis solvers. The analysis application includes:

a) Identifying a desired analysis type

b) Defining translation and solution parameters

c) Selecting a sequence of load cases

d) Selecting the desired output

e) Sending the model data to the analysis solver

f) Reading the result quantities from the results files.

8) Compiling the analysis results: Patran can use computer graphics, animation, and other results tools to make the results of a finite element analysis easy to understand.

9) Visualizing numerical results: Patran is state-of-the-art in its ability to display, sort, combine, scale, and query, in a general way, a single results database. After execution, analysis results are loaded directly into the Patran relational database and can be sorted by time step, frequency, temperature, or spatial location for visual displays (MSC. Software, 2012).

The procedure for modeling the Mat foundation with Patran is explained step by step in the Appendix.

3.2 Approximate Flexible Method

In the approximate flexible method of design, the soil is assumed to be equivalent to an infinite number of elastic springs, as shown in Fig. 3.1. This assumption is referred to as the Winkler foundation. The elastic constant of these assumed springs is called the coefficient of subgrade reaction, k.

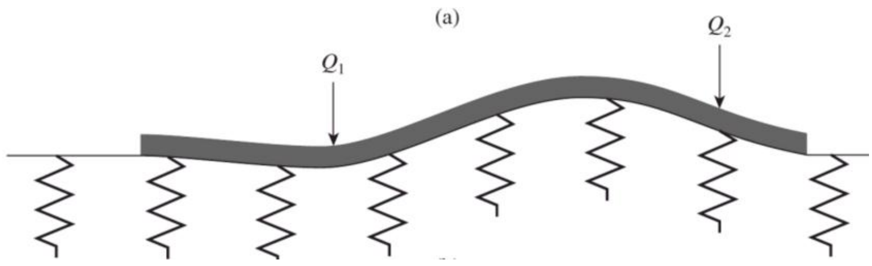


Fig. 3.1 Approximate flexible method configuration

In the approximate flexible method, the foundation is assumed as a beam of width B having infinite length and a concentrated load Q. It uses the mechanics of materials fundamental equations (Eq.3.1).

$$M = E_F I_F \frac{d^2z}{dx^2} \qquad (Eq. 3.1)$$

where

M = moment at any section

E_F = modulus of elasticity of foundation material

I_F = moment of inertia of the cross section of the beam $= \left(\frac{1}{12}\right) B_1 h^3$

$\frac{dM}{dx} = \text{shear force} = V$

$\frac{dV}{dx} = q = \text{soil reaction}$

Hence,

$$\frac{d^2M}{dx^2} = q \qquad (Eq. 3.2)$$

Combine Eqs (3.1) and (3.2)

$$E_F I_F \frac{d^4z}{dx^4} = q \qquad (Eq. 3.3)$$

Where the soil reaction is

$$q = -zk' \qquad (Eq. 3.4)$$

where

z = deflection

$k' = kB_1$

k = coefficient of subgrade reaction (kN/m^3 or lb/in^3)

$$E_F I_F \frac{d^4 z}{dx^4} = -zkB_1 \qquad (Eq. 3.5)$$

$$z = e^{-ax}(A' \cos \beta x + A'' \sin \beta x) \qquad (Eq. 3.6)$$

where A' and A'' are constants and

$$\beta = \sqrt[4]{\frac{B_1 k}{4E_F I_F}} \qquad (Eq. 3.7)$$

β is an important parameter that determines the method by which mat foundation should be designed. According to the American Concrete Institute Committee 336 (1988), mats should be designed by the approximate flexible method if the space of the columns is larger than 1.75/β.

The following are the steps for analyzing mat foundation by the approximate flexible method introduced by ACI Committee 336:

1) Assume a thickness h for the mat

2) Determine the flexural rigidity R of the mat

$$R = \frac{E_F h^3}{12(1-\mu_F^2)} \qquad (Eq. 3.8)$$

where

E_F = modulus of elasticity of foundation material

μ_F = Poisson's ratio of foundation material

3) Use the equation to determine the radius of effective stiffness L'

$$L' = \sqrt[4]{\frac{R}{k}} \qquad (Eq. 3.9)$$

where k = coefficient of subgrade reaction.

4) Determine the moment using (Eq.3.9) and (Eq.3.10)

$$M_r = radial\ moment = -\frac{Q}{4}[\,A_1 - \frac{(1-\mu_F)A_2}{\frac{r}{L'}}] \qquad \text{(Eq.3.10)}$$

$$M_t = tangential\ moment = -\frac{Q}{4}[\,A_1 + \frac{(1-\mu_F)A_2}{\frac{r}{L'}}] \qquad \text{(Eq.3.11)}$$

where

r = radial distance from the column load

Q = column load

A_1, A_2 = functions of r/L'

$$M_x = M_t \sin^2 \alpha + M_r \cos^2 \alpha \qquad \text{(Eq.3.12)}$$

$$M_y = M_r \sin^2 \alpha + M_t \cos^2 \alpha \qquad \text{(Eq.3.13)}$$

5) Determine the shear force for the unit width of the mat.

$$V = \frac{Q}{4L'} A_3 \qquad \text{(Eq.3.14)}$$

6) Determine the moment and shear force along the edge, if the edge of mat foundation is still in column influence zone.

7) Find out the deflection at any point.

$$\delta = \frac{QL'^2}{4R} A_4 \qquad \text{(Eq.3.15)}$$

Get all A_1, A_2, A_3, and A_4 from the Fig.3.2.

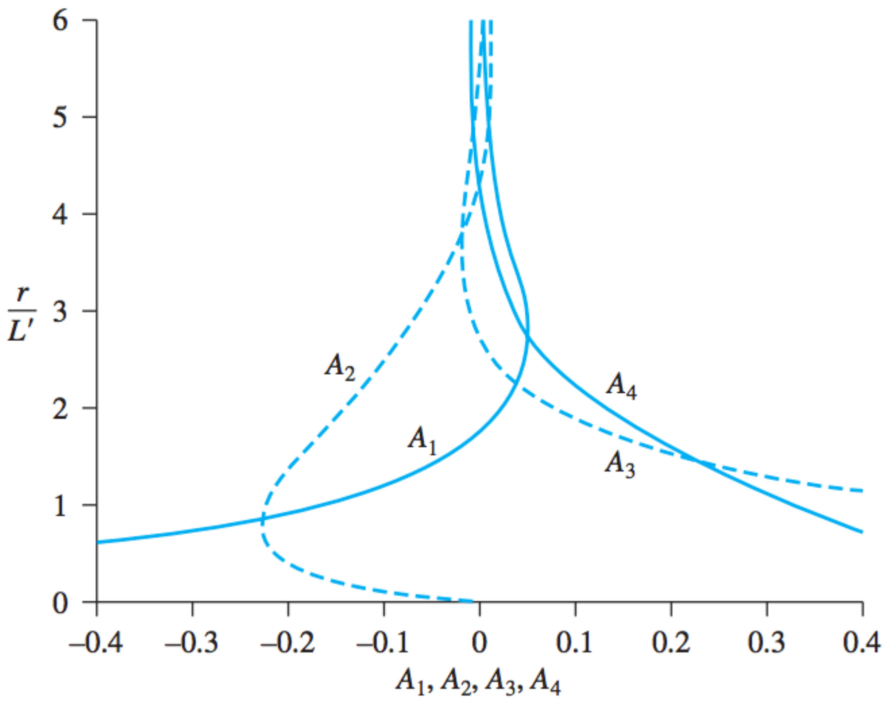


Fig.3.2 A_1, A_2, A_3 and A_4 values in ACIC 336 (Das, M., 2007)

3.3 Calculation of Spring Constant for Patran Modeling

The approximate flexible method treats soil foundation as springs. To analyze the flexible mat, the coefficient of a subgrade reaction needs to be calculated. Vesic (1961) proposed an equation to estimate a subgrade reaction.

$$k = 0.65\sqrt[12]{\frac{E_S B^4}{E_F I_F}}\,\frac{E_S}{B(1-\mu_S^2)} \qquad (Eq. 3.16)$$

where

E_s = modulus of elasticity of soil

B = foundation width

E_F = modulus of elasticity of foundation material

I_F = moment of inertia of the cross section of the foundation

μ_s = Poisson's ratio of soil

After obtaining the subgrade reaction k for a flexible mat, use this reaction to multiply the area of different springs to get the spring constant for each spring in (Eq.3.17-3.19) (Das, M, 2007).

$$k_i = (L \times B)k \qquad \text{(Eq.3.17)}$$

$$k_e = \frac{(L \times B)}{2} k \qquad \text{(Eq.3.18)}$$

$$k_c = \frac{(L \times B)}{4} k \qquad \text{(Eq.3.19)}$$

where

k_i = constant of inside springs

k_e = constant of springs at edge

k_c = constant of springs at corner

k = subgrade reaction

L = length of each spring's area

B = width of each spring's area

CHAPTER 4

VERIFICATION OF FINITE ELEMENT MODELING METHOD BY APPROXIMATE FLEXIBLE METHOD

The main objective of this study is to perform a sensitivity analysis of mat foundation by finite element modeling. In order to use finite element modeling, the result should be verified. In this chapter, the result of the approximate flexible method is used to verify the result of Finite Element Modeling.

4.1 Patran Modeling

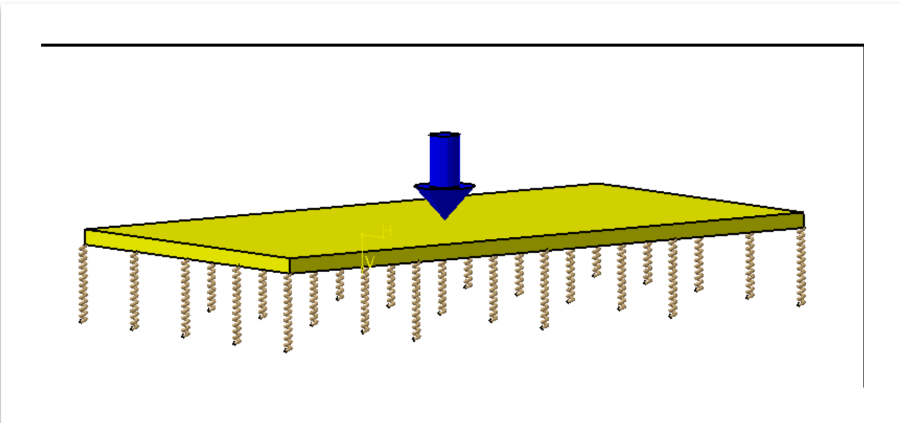


Fig.4.1 Model of Mat Foundation for Verification

In this section, a mat foundation with 20 meters length, 15 meters width, and 0.1 meter thickness subjected to 1-point load is modeled. 25 springs (Fig.4.1) which are found from convergence study, are used for modeling soil. The result of this convergence study is presented in the next section of this chapter.

4.2 Optimum Number of Springs for Soil Modeling

Soil is modeled by springs, and the optimum number of springs needs to be found in order to have an economical simulation with convincing accuracy. In this study, three different numbers of springs: 9 springs, 16 springs, and 25 springs are considered for modeling soil. The result for the maximum foundation deflection with 9, 16, and 25 springs are 34.1mm, 43.1mm, and 45.9mm respectively. Figures 4.2, 4.3, and 4.4 show the deflection for models with 9 springs, 16 springs, and 25 springs, respectively. The deflection does not change significantly from 16 springs to 25 springs. Thus, 25 springs is chosen as the optimum number of springs for soil modeling.

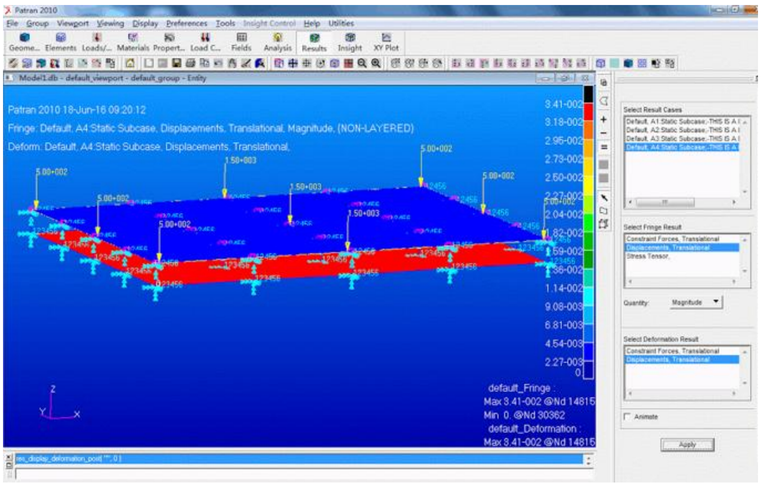


Fig.4.2 Mat foundation deflection with 9 springs

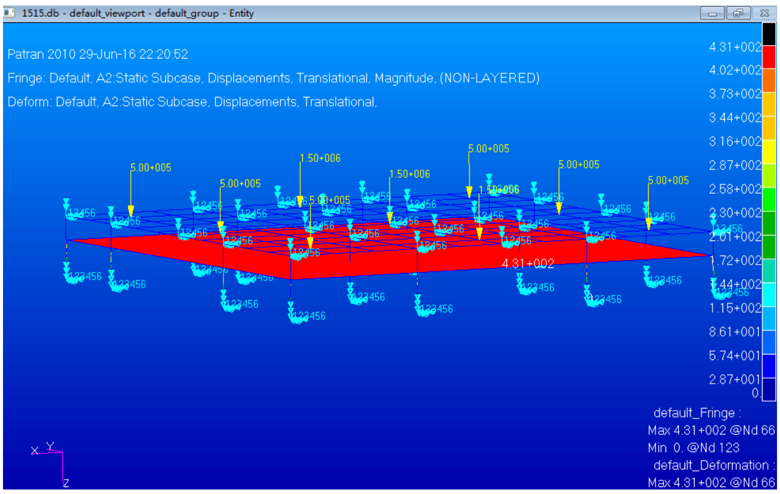


Fig.4.3 Mat foundation deflection with 16 springs

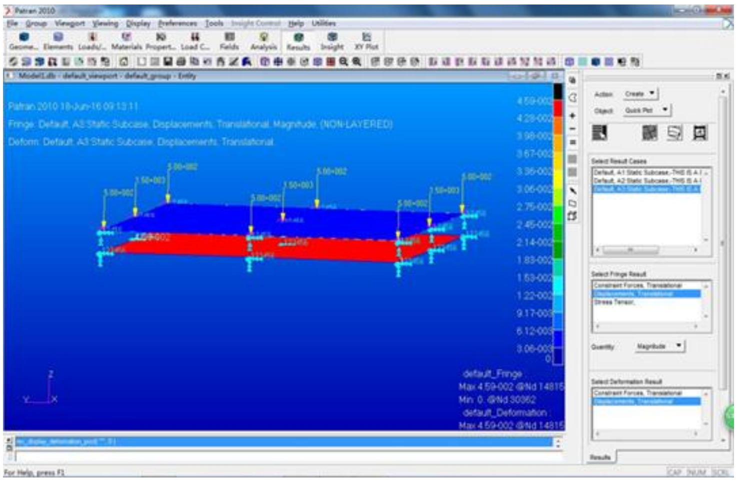


Fig.4.4 Mat foundation deflection with 25 springs

4.3 Comparing the results of Patran Modeling and Approximate Flexible Method

The ACIC 336 figure (Fig. 3.2), which is used in Approximate Flexible Method, has been developed based on experimental studies (Das, M., 2007). As explained in the approximate flexible method section, the radial moment, the tangential moment, and the shear force and deflection at any point of mat foundation can be found by using an ACIC 336 figure (Fig.3.3). These outputs can be found from the Patran Modeling directly. In order to verify the validation of these outputs, the figures (Fig. 4.5- 4.8) were developed from the results of Patran.

This model is 20 meters in length, 15 meters in width, and has 0.1 meter thickness. So the largest distance for the center is 12.5 meters (Table 4.1).

Table 4.1 Material Properties in Patran Modeling

Material:	Es (MPa)	Ec (MPa)	Q(kN)	Poisson's Ratio
	20000000	25000000000	1500000	0.2

Take 14 points as the distance to center from 0 to 12.5 meters and each two points has a 1 meter spacing, except that the last one has only a 0.5 meter spacing. The result of the X moment, Y moment, and Shear force and Deflection for points with different distance from loading point on the foundation are presented in Table 4.2.

Table 4.2. Result of Patran Modeling for Verification

r(m)	Deflection(mm)	V(N)	MomentX(Pa)	MomentY(Pa)
0	25.34	1028040	483021	545019
1	24.34	392530	198522	194008
2	22.33	193745	103763	89694
3	21.33	91279	52679	38600
4	19.78	39553	23922	15631
5	18.79	38279	5947	32332
6	17.34	410069	176356	233533
7	16.46	87951	28610	59341
8	14.56	29402	12534	16868
9	12.98	7453	3967	3486
10	10.36	-4260	1115	-5375
11	8.76	21982	5549	16433
12	6.56	104947	47895	57052
12.5	6.55	0	0	0

By using the Eqs. 3.7 to 3.14 from the approximate flexible method, the value of A_1, A_2, A_3, and A_4 at every point are recalculated (Table 4.2). Figs. 4.5 – 4.8 show the answers for A1, A2, A3, and A4.

Table 4.2. A1, A2, A3, A4 in each point on the Mat foundation

r	A1	A2	A3	A4	r/L
0	-0.81	0	1.42	0.63	0
1	-0.67	-0.14	1.13	0.56	0.28
2	-0.4	-0.22	0.72	0.45	0.58
3	-0.23	-0.2	0.54	0.38	0.86
4	-0.14	-0.185	0.44	0.32	1.16
5	-0.02	-0.165	0.37	0.27	1.45
6	0.03	-0.14	0.25	0.25	1.73
7	0.05	-0.11	0.13	0.17	2.03
8	0.06	-0.076	0.07	0.12	2.32
9	0.1	-0.05	0.01	0.08	2.61
10	0.11	-0.025	-0.01	0.06	2.89
11	0.1	-0.01	-0.005	0.03	3.18
12	0.09	0	0	0.02	3.48
12.5	0.08	0	0	0.017	3.48

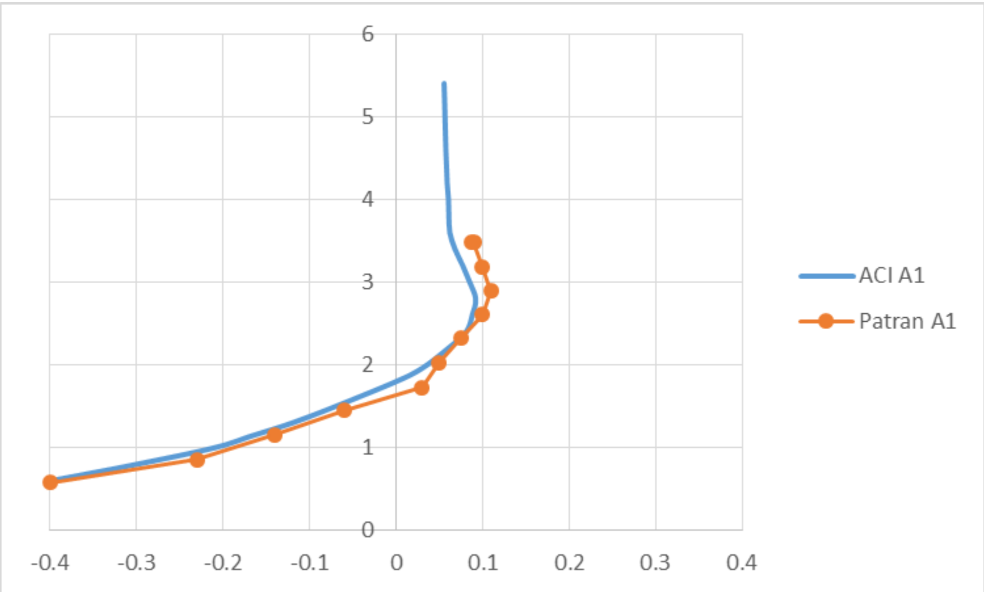


Fig.4.5 A1 graph from ACI and developed by Patran

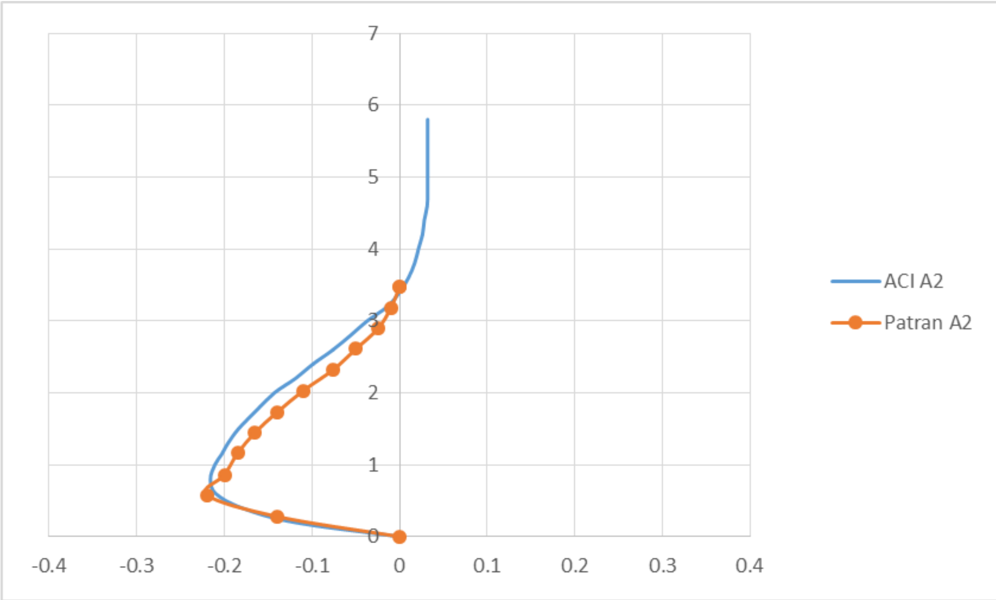


Fig.4.6 A2 graph from ACI and developed by Patran

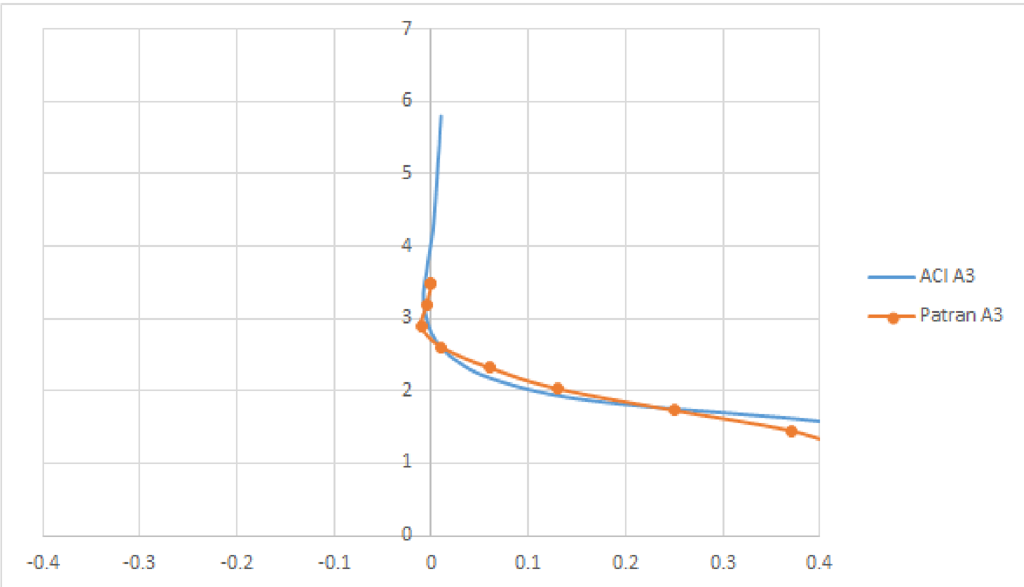


Fig.4.7 A3 graph from ACI and developed by Patran

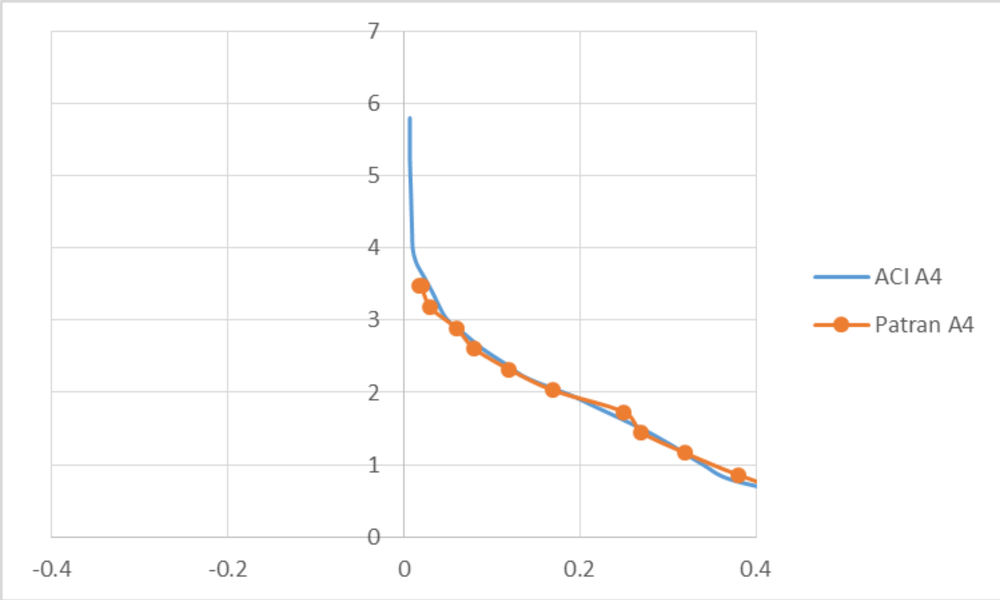


Fig.4.8 A4 graph from ACI and developed by Patran

The figures (Fig. 4.5- 4.8) show that the graphs developed by Patran are almost identical to the ACIC 336 graphs. This result verifies the validity of using Patran for modeling mat foundation.

CHAPTER 5

GLOBAL SENSITIVITY ANALYSIS OF MAT FOUNDATIONS

There are several factors that influence the behavior of mat foundations such as the modulus of elasticity of soil and foundation, the foundation width, the length and thickness, and the loading patter. The sensitivity of mat foundation behavior to each of these factors is different. In order to perform sensitivity analysis, input variables and output variables for this system should be defined.

Since considering the variation of all of these variables at the same time would be costly due to the large number of simulations, the global sensitivity analysis is performed on two sets of input variables. The input variables for the first analysis are "Modulus of elasticity of soil", "Modulus of elasticity of foundation (which is concrete)", and "Column loads". The input variables for the second analysis are "Modulus of elasticity of soil", "Foundation thickness", and "Foundation aspect ratio". The output variable is the maximum deflection of the foundation for both sensitivity analyses.

After choosing the input variables, the next step is to define the domain for input variables. Different approaches are chosen for developing the input domain for the two sensitivity analyses. For the first sensitivity analysis, the input domain is developed by choosing a mean value for each variables and a coefficient of variation of 0.1. For the second sensitivity analysis, the input domain is considered as the whole range that each input variable can vary.

5.1 Input variables (1): Soil elastic modulus, Concrete elastic modulus, Column loads

A mat foundation with 20 meters length, 15 meters width, and 1 meter thickness subjected to 9-point column load is modeled for this section. 25 springs, which is found from convergence study, are used for modeling soil (Fig.5.1).

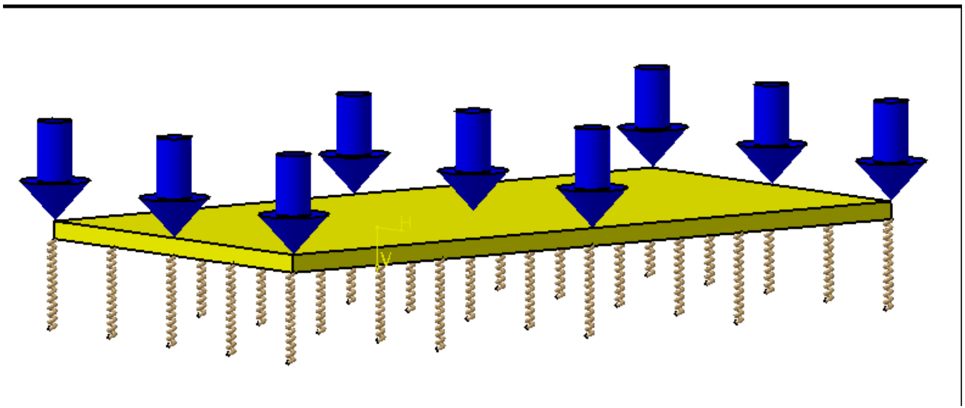


Fig.5.1 Model of Mat Foundation for Sensitivity analysis

5.1.1 Domain Development for Input Variables

There is an uncertainty about the input variables in any system or model. In this section, the uncertainty for input variables is considered by accounting for the coefficient of variation, c_v, equal to 0.1, for all the input variables. We also assume that the input variables are uniformly distributed in the domain. We define the range of variation for input variable X from a to b (a < x < b). Thus, we should find "a" and "b" in terms of x mean value "μ" and a coefficient of variation "c_v":

The probability density function of the continuous uniform distribution is:

$$f(x) = \begin{cases} \frac{1}{b-a}, & a \le x \le b \\ 0 & \text{otherwise} \end{cases} \qquad \text{(Eq.5.1)}$$

Here is how the mean of uniform distribution is calculated:

$$\mu = E(X) = \int_{-\infty}^{\infty} x f(x) \mathrm{d}x = \int_a^b x \frac{1}{b-a} \mathrm{d}x = \frac{1}{2(b-a)} \left[x^2\right]_a^b$$

$$= \frac{b^2 - a^2}{2(b-a)}$$

$$= \frac{b+a}{2} \quad \text{(Eq.5.2)}$$

Here is how the variance of uniform distribution ($\sigma^2 = V(X)$) is calculated:

$$V(X) = E(X^2) - [E(X)]^2$$

$$= \int_a^b x^2 . \frac{1}{b-a} \mathrm{d}x - \left(\frac{b+a}{2}\right)^2 = \frac{1}{3(b-a)} \left[x^3\right]_a^b - \left(\frac{b+a}{2}\right)^2$$

$$= \frac{b^3 - a^3}{3(b-a)} - \left(\frac{b+a}{2}\right)^2 = \frac{b^2 + ab + a^2}{3} - \frac{b^2 + 2ab + a^2}{4} = \frac{(b-a)^2}{12} \quad \text{(Eq.5.3)}$$

The standard deviation (σ), which is the square root of variance, is

$$\sigma = \sqrt{V(X)} = \frac{b-a}{2\sqrt{3}} \quad \text{(Eq.5.4)}$$

Combining Eqs. (5.2) and (5.4), the value of “a” and “b” are calculated as follows:

$$\begin{cases} a = \mu - \sqrt{3}\sigma \\ b = \mu + \sqrt{3}\sigma \end{cases} \quad \text{(Eq.5.5)}$$

Considering coefficient of variation is the ratio of standard deviation (σ) over mean value (μ)

$$c_v = \frac{\sigma}{\mu} \quad \text{(Eq.5.6)}$$

the value of “a” and “b” in equation 5.5 can be written as

$$\begin{cases} a = \mu(1 - \sqrt{3}c_v) \\ b = \mu(1 + \sqrt{3}c_v) \end{cases} \quad \text{(Eq.5.7)}$$

The next step in sensitivity analysis is to choose a mean value for the input variables. As was mentioned before, the three input variables in this section are “Modulus of elasticity of soil”, “Modulus of elasticity of concrete”, and “Column loads”.

Since the soil under a mat foundation usually has a low bearing capacity, the mean value for the modulus of elasticity of soil is considered 20 MPa. This is the average value of the modulus of elasticity for clay (Das, M., 2007).

The foundation material is considered to be concrete; therefore, the mean value for the modulus of elasticity of the foundation material is considered 25000 MPa. This is the average value of modulus of elasticity for concrete.

For a four story building, the mean values of the column load in the edge and in the center of the foundation are assumed to be 500 kN and 1500 kN, respectively (Bowels, J.E., 1997).

Table 5.1 shows the mean value and the range of variations for all of the input variables. The minimum and maximum values in this table are the "a" and "b" values in Equation 5.7. The coefficient of variation (c_v) for all the input variables is considered as "0.1"

Table 5.1 Mean value and range of variations for all the input variables

Input Variables	Mean Value	Minimum Value	Maximum Value
Es (MPa)	20	17	23
Ec (MPa)	25000	20670	29330
P(kN)	500	413	587

5.1.2 Result of Sobol functions and Sobol indices

The maximum displacement of the foundation is chosen as the output variable for the sensitivity analysis. In order to calculate the Sobol function and Sobol indices, we need to have a sufficient amount of simulations. Five points equally spaced in the domain are chosen for each input variable and the model is run for all of the possible conditions. Since there are three input variables, the number of simulations is **5x5x5= 125**. Table 5.2 shows five points along the domain for input variables which are used in simulations.

Table 5.2 Five points along the domain of input variables (1) used in simulations

Input Variables	Point 1	Point 2	Point 3	Point 4	Point 5
Es (MPa)	17.2	18.6	20	21.4	22.8
Ec (MPa)	21535.9	23267.9	25000	26732.1	28464.1
P(kN)	430.7	465.4	500	534.6	569.3

Figures 5.6, 5.7 and 5.8 show the Sobol functions and Sobol indices calculated from 125 simulations.

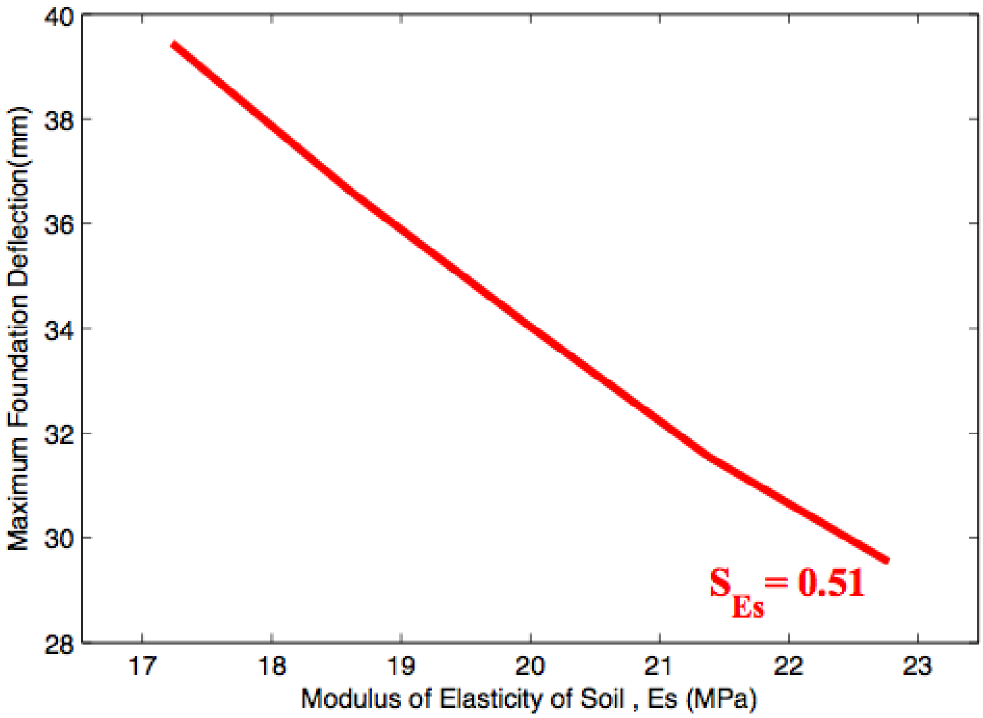


Fig.5.6 Maximum foundation deflection versus modules of elasticity of soil for input variables (1)

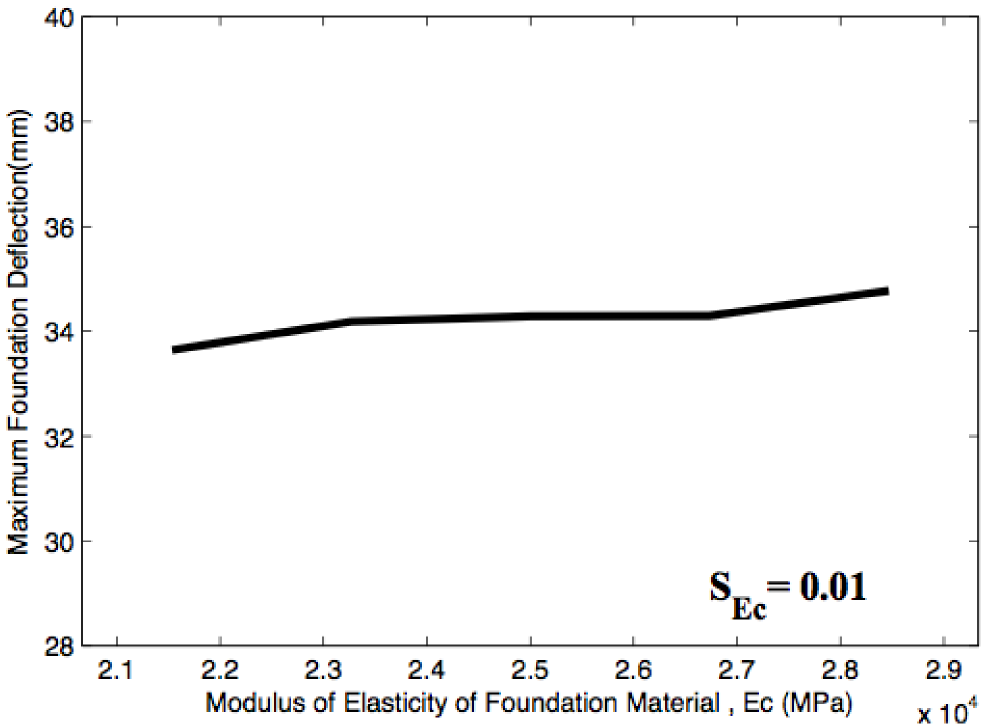


Fig.5.7 Maximum foundation deflection versus modules of elasticity of concrete for input variables (1)

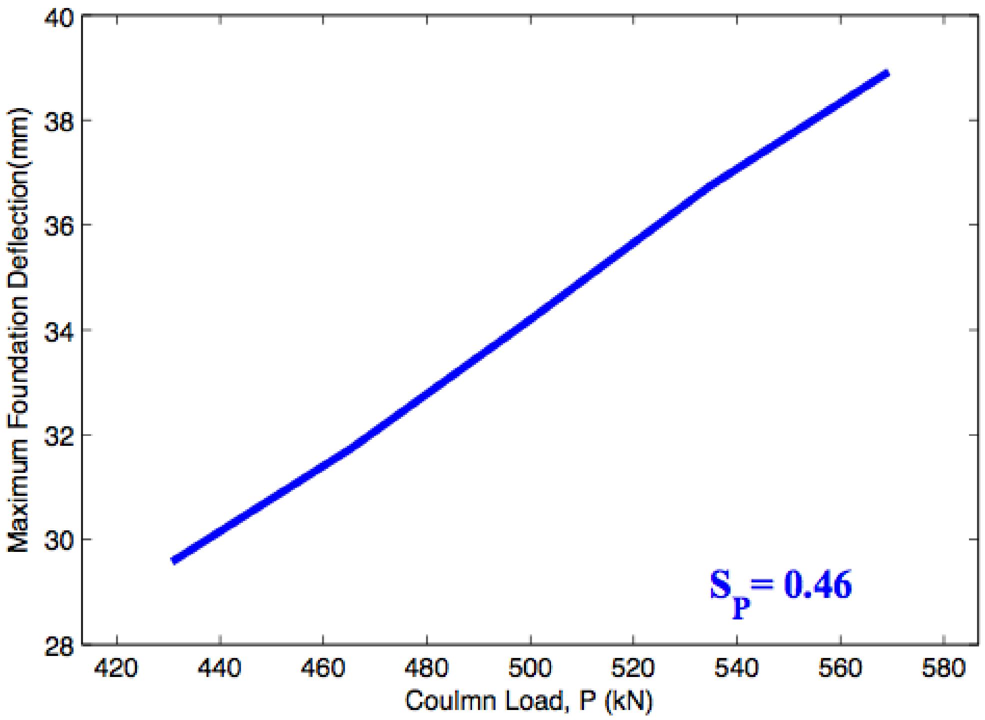


Fig.5.8 Maximum foundation deflection versus column load for input variables(1)

It should be mentioned that the higher the Sobol indices of given input variables, the higher the sensitivity of the output function to that input variable. Therefore, it can be seen from Sobol functions and Sobol indices that the maximum deflection of the foundation are sensitive to the following components from the most to the least:

1. Modulus of Elasticity of Soil
2. Column Load
3. Modulus of Elasticity of Concrete

The summation of Sobol indices is also equal to

$$S_{Es} + S_{Ec} + S_P = 0.978 \approx 1$$

This shows that the second order indices are negligible and that the interaction of these inputs does not have significant effect on the output.

5.2 Input variables (2): Soil Elastic Modulus, Foundation Thickness, Foundation Aspect Ratio

In this section, the sensitivity of "Foundation Deflection" to "Soil Elastic Modulus", "Foundation Thickness", and "Foundation Aspect Ratio" is evaluated. The loading pattern in this section is similar to the loading pattern in section 5.1 (Fig 5.1) with column loads equal to 500 kN and 1500 kN in the edge and in the center of the foundation, respectively. The elastic modulus of concrete is 25000 MPa in the analysis of this section.

5.2.1 Domain Development for Input Variables

The domains of input variables for this section are developed based on the possible value that these variables can have. The elastic modulus for soft and medium clay is in the range of 4 MPa to 40 MPa (Das, M., 2007). Foundation thickness is assumed to be in the range of 0.1m and 0.7m. The thickness can be more than 0.7m, but the results show foundation with thickness more than 0.7m will have no relative displacement under given loading conditions. Thus, 0.7 is chosen as the highest value for the thickness range. The foundation aspect ratio is assumed to be in the range of 1 to 3 (Das, M., 2007).

5.2.2 Result of Sobol functions and Sobol indices

Similar to section 5.1, for calculation of the Sobol function and Sobol indices, we need to have a sufficient amount of simulations. Five points equally spaced in the domain are chosen for each input variable and the models are run for all the possible conditions. Since there are three input variables, the number of simulations is **5x5x5= 125**. Table 5.3 shows five points along the domain for input variables which are used in simulations.

Table 5.3 Five points along the domain of input variables (2) used in simulations

Input Variable	Point 1	Point 2	Point 3	Point 4	Point 5
Es (MPa)	4	13	22	31	40
Thickness (m)	0.10	0.25	0.40	0.55	0.70
L/B	1.0	1.5	2.0	2.5	3.0

Figures 5.9, 5.10 and 5.10 show the Sobol functions and Sobol indices calculated from 125 simulations.

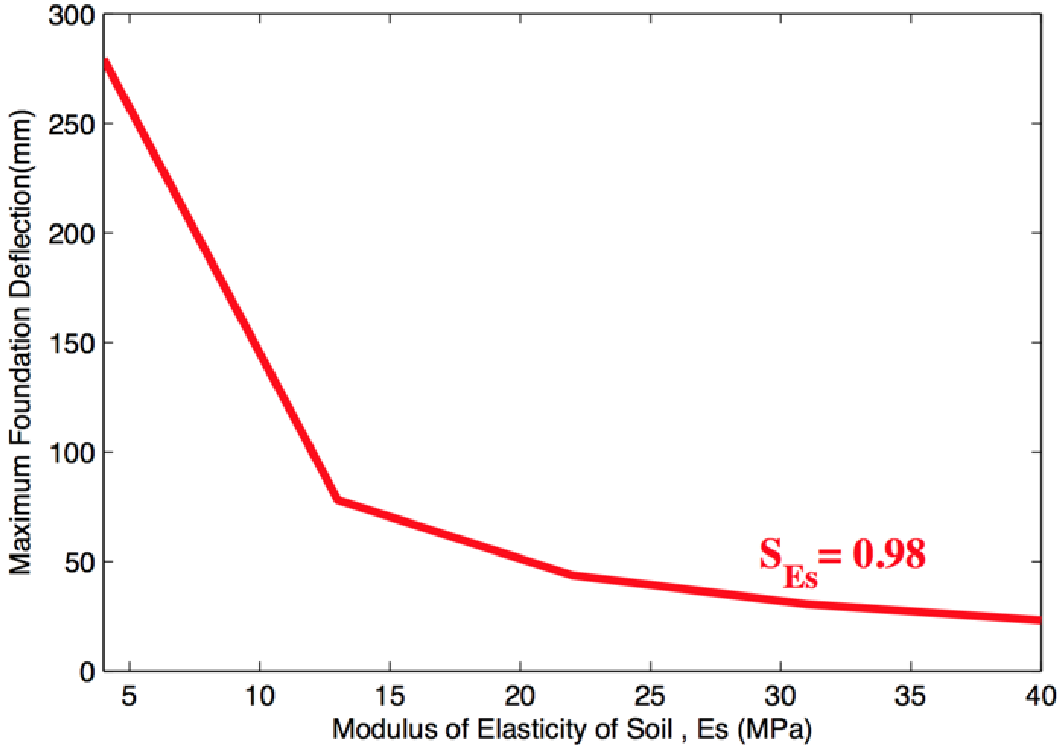


Fig.5.9 Maximum foundation deflection versus modules of elasticity of soil for input variables 2)

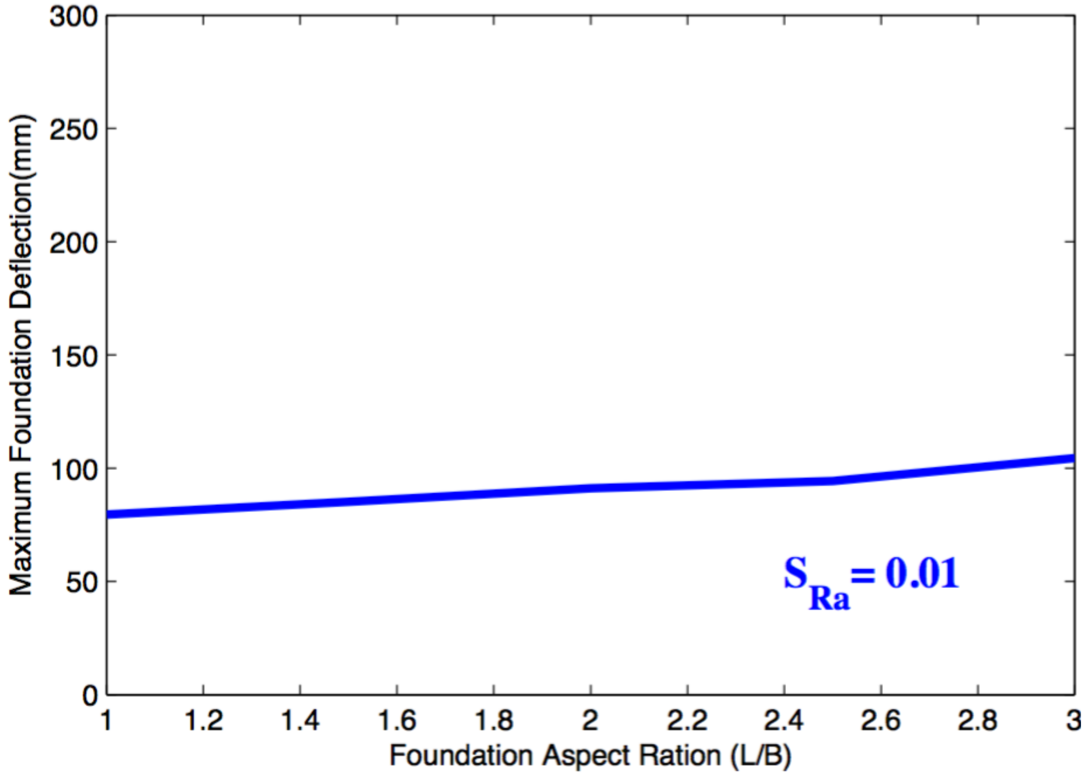


Fig.5.10 Maximum foundation deflection versus foundation aspect ratio for input variables (2)

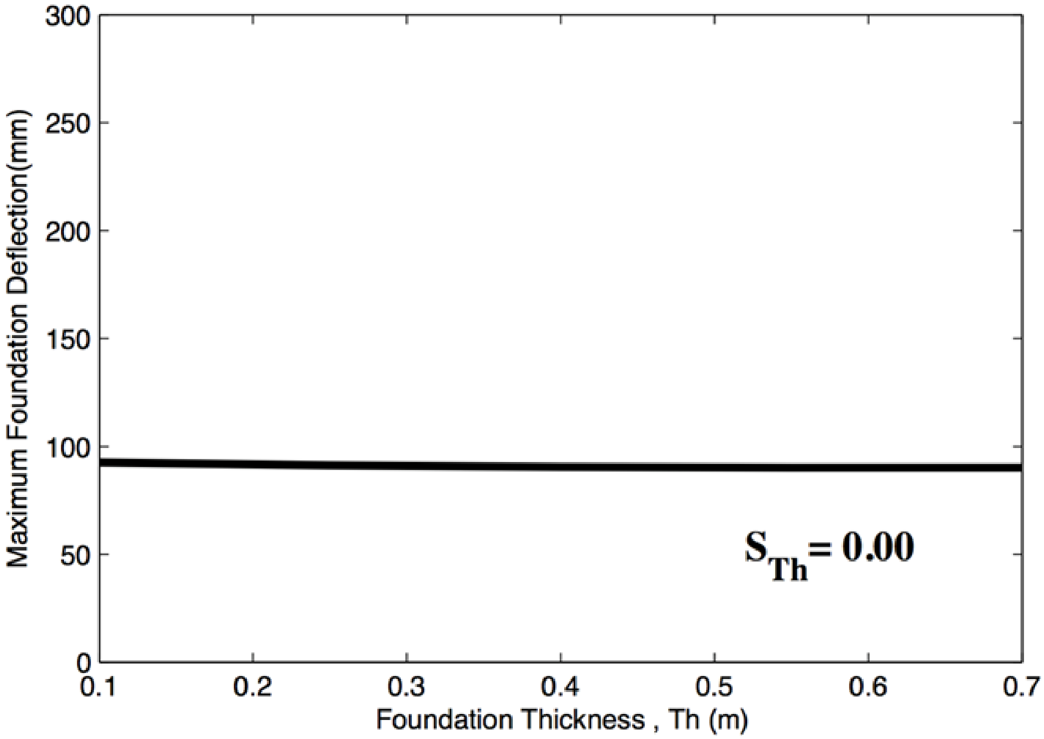


Fig.5.11 Maximum foundation deflection versus foundation thickness for input variables(2)

It should be mentioned that the higher the Sobol indices of given input variables, the higher the sensitivity of the output function to that input variable. For these input variables, it can be seen from Sobol functions and Sobol indices that maximum deflection of the foundation are sensitive to the following components from the most to the least:

1. Soil Elastic Modulus
2. Foundation Aspect Ratio
3. Foundation Thickness

The summation of Sobol indices is also equal to

$S_{Es} + S_{Ra} + S_{T_h} = 0.99 \approx 1$

This also shows that the second order indices are negligible and that the interaction of these inputs does not have significant effect on the output. Foundation thickness has no effect at all from 0.1m to 0.7m.

Two sensitivity analyses with different sets of input variables are performed. In the first analysis, "Modulus of elasticity of soil", "Modules of elasticity of foundation material", and "Column load" is chosen as the input. In the second analysis, "Modulus of elasticity of soil", "Foundation aspect ratio", and "Foundation thickness" is chosen as the input. "Maximum deflection of the Mat foundation" is considered as the output response in both analyses.

Modulus of elasticity of soil is the most sensitive input variables in those five variables. Modulus of elasticity of foundation material and Foundation aspect ratio have a slightly effect on the deflection of mat foundation. Foundation thickness almost has no effect at all from 0.1m to 0.7m.

CHAPTER 6

SUMMARY AND CONCLUSIONS

In this study, the sensitivity of mat foundation behavior to some of the design parameters is investigated by using Patran, which is a Finite Element Based software. Working with finite element-based software such as Patran for modeling structural components is often less expensive than using analytical methods. But the validity of finite element modeling needs to be verified before using finite element analysis results.

The results of the Patran modeling in this study are verified by approximate flexible method by regenerating the ACIC 336 Figure in approximate flexible method. The ACIC 336 Figure has been developed experimentally and it has four graphs: A1, A2, A3, and A4. These graphs are used to find “Radial Moment”, “Tangential Moment”, “Shear Force”, and “Deflection” in the mat foundation, respectively. Patran is used to redraw these graphs.

Results show that the ACIC 336 graphs are almost similar to the graphs developed by Patran. This result verifies the validity of using Patran for modeling mat foundation. Therefore, Patran can be used for modeling mat foundation under different kinds of loadings and conditions.

After verification of the finite element modeling, the next section of this study is to use FEM in a global sensitivity analysis. Sobol decomposition, which is used in this study, is one of the variance based techniques which can be used for global sensitivity analysis. It is shown that Sobol decomposition can be used as a convenient method for performing global sensitivity analysis for engineering problems.

Two sensitivity analyses with different sets of input variables are performed. In the first analysis, “Modulus of elasticity of soil”, “Modules of elasticity of foundation material”, and “Column load” is chosen as the input. In the second analysis, “Modulus of elasticity of soil”, “Foundation aspect ratio”, and “Foundation thickness” is chosen as the input. “Maximum deflection of the mat foundation” is considered as the output response in both analyses. 125 simulations are performed for each analysis, and the results for Sobol function and Sobol indices are calculated. Sobol function can be used to find how the variation of foundation deflection is, with respect to input variables. The values of Sobol indices can be used as a parameter for comparing the sensitivity of the response function to different input variables.

For the first sensitivity analysis, the following results are found by evaluating Sobol function and Sobol indices:

- The foundation deflection reduces by increasing the modulus of elasticity of soil and concrete and increases by increasing the column loads.
- The Sobol indices’ value in order of magnitude are “Modulus of elasticity of soil”, “Column load”, and “Modules of elasticity of concrete”. This shows that foundation deflection is very sensitive to modulus of elasticity of soil, and then foundation loading, and then modulus of elasticity of concrete.
- The Sobol indices are almost zero for modulus of elasticity of concrete. This shows that Modulus of elasticity of concrete does not affect the foundation deflection, compared to Modulus of elasticity of Soil and Column load.

For the second sensitivity analysis, the following results are found by evaluating Sobol function and Sobol indices:

- Foundation deflection reduces by increasing the modulus of the elasticity of soil and increases slightly by increasing the foundation aspect ratio.
- Foundation deflection almost has zero change by increasing foundation thickness.

- The Sobol indices’ value in the order of magnitude are “Modulus of elasticity of soil”, “Foundation aspect ratio”, and “Foundation thickness”. This shows that foundation deflection is very sensitive to modulus of elasticity of soil, and then foundation aspect ratio, and then foundation thickness.

- The Sobol indices are zero for foundation thickness. This shows that for this mat foundation with the given loading condition, the foundation thickness does not affect the deflection compared to foundation aspect ratio and modulus of elasticity of soil.

REFERENCES

1) Ahmad, S., Curved Finite Elements in the Analysis of solid, shell and plate structures, Ph.D. Thesis, University College of Swansea, 1969

2) Arwade, S., Moradi, M., and Louhghalam, A., Variance decomposition and global sensitivity for structural systems, Engineering Structure, Vol.32, 2010. pp 1-10.

3) Bowels, J.E., Foundation analysis and Design, 5th Edition. McGraw Hill Book Co., New York, 1997.

4) Das, M., Principles of Foundation Engineering, 7th Edition. Cengage Learning, 2007.

5) Das, M., Principles of Foundation Engineering 6th ed. Cengage Learning, 2006.

6) Hussin, J., Method of Soft Ground Improvement. Taylor & Francis Group, 2006

7) Lee, J., 3D analytical method for mat foundations considering coupled soil springs, Geomechanics and Engineering, Vol. 8, 2015

8) Patran 2012 User's Guide, MSC. Software Corporation, 2012

9) Rana, S. C., A critical review on idealization and modeling for interaction among soil-foundation-structure system," elsevier, nr. Elsevier Science Ltd., 2002.

10) Salteli, A., Tarantola, S., Campolongo, F., and Ratto, M., Sensitivity Analysisi in Practice: A Guide to Assessing Scientific Models, John Wily & Sons Ltd, England, 2004

11) Salteli, A., Tarantola, S., Campolongo, F., and Cariboni, J., Gatelli, D., Saiana, M., and Tarantola, S., Global Sensitivity Analysis: The Primer, John Wily & Sons Ltd, England, 2008

12) Srilakshmi, G., Analysis of MAT Foundation using Finite Element Method, International Journal of Earth Sciences and Engineering, Vol 04, 2011, pp. 113-115

13) Sutradhar, A., An improved design rationale for Mat Foundation based on Finite Element Analysis, Master. Thesis, Bangladesh University of Engineering and Technology, 1999

14) Z.S. Tang, N.S.V.K. Rao , Jodin Makinda & Nurmin Bolong, Converntional and finite element method analysis of foundation soil ineteraction, the 10th Seminor of Science & Technology, 2012

15) Ukarande, G.S., A Parametric Study on Raft Foundation, The 12th International Conference of International Association for Computer Methods and Advances in Geomechanics, India, 2008

16) Vlasov, V.Z. and Leonitiev, N.N, Beams, Plates and Shells on an Elastic Foundation, USSR: Fizmatgiz, Moscow, 1960

17) Yamshita, K. and Yamada, T., Simplified method for analyzing piled raft foundations, Deep Foundation on Bored and Auger Piles, 1998

18) Zienkiewicz, O. C., The Finite Element Method in Structural & Mechanics, McGraw Hill, London, 1968

APPENDIX A

STEP BY STEP PROCEDURE FOR MODELING MAT FOUNDATION WITH PATRAN

In this section, the step-by-step procedure for modeling Mat foundation with Patran is presented. A mat Foundation with 20 meters length, 15 meters width, and 0.1meter thickness, subjected to a 1-point load in the center. The mat foundation is modeled in the section. 25 springs is used for modeling soil. The points, nodes and elements in all the figures in this section are random and they are used as an example for explaining different sections.

The steps are as follow:

A.1 Geometry modeling

There are two parts of Mat foundation modeling: the Mat Plate and the Springs.

1) Creating the Mat Plate

Create the Base of the Mat Plate

On the Patran Main Menu, click on the Geometry Application button. (Fig. A1)

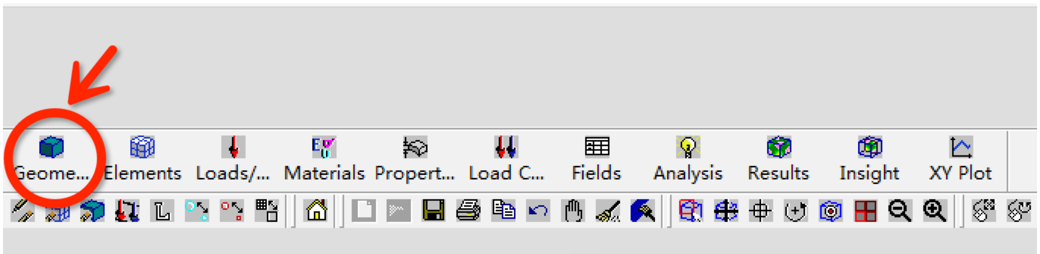


Fig. A1 Patran Main Menu for Geometry

On the top of the Geometry form, select Action>> Create, Object>> Point, Method>> XYZ, Point ID is 11, Refer. Coordinate Frame is Coord 0, Click Auto Execute Selection and Point Coordinates List is Node 1. (Fig. A2)

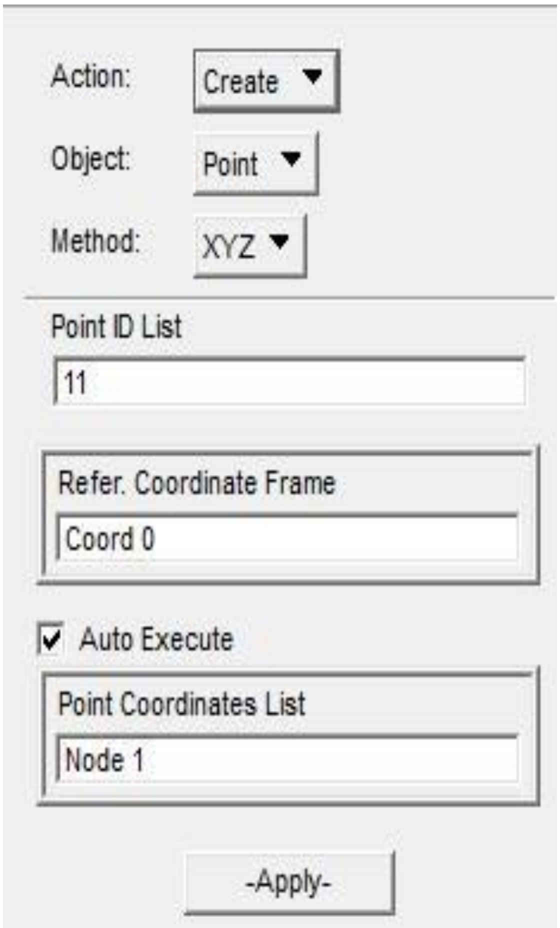


Fig. A2 Geometry form for Mat base

Create the Surface of the Mat Plate

On the Patran Main Menu, click on the Geometry Application button. (Fig.A1)

On the top of the Geometry form, select action>> Create, Object>> Surface, Method>> Curve, Surface ID >> 1, Option>> 4 Curve, Click>> Auto Execute and Starting Curve List are Curve 1, Curve 2, Curve 3, Curve 4. (Fig.A3)

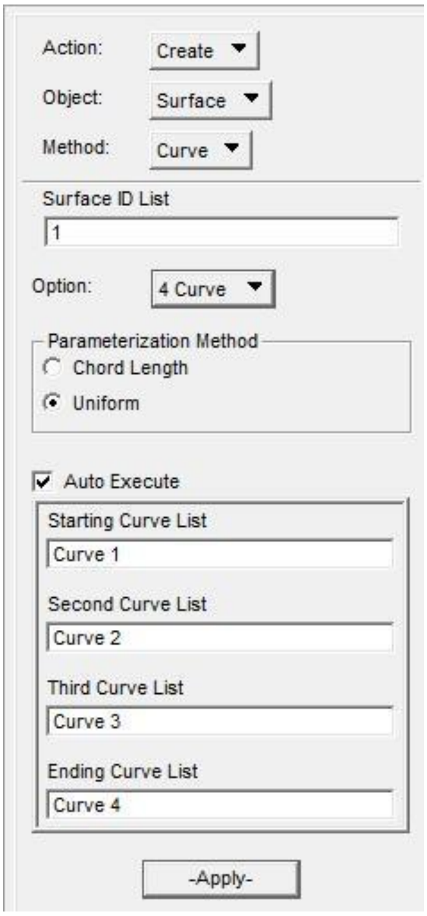


Fig.A3 Geometry form for Mat surface

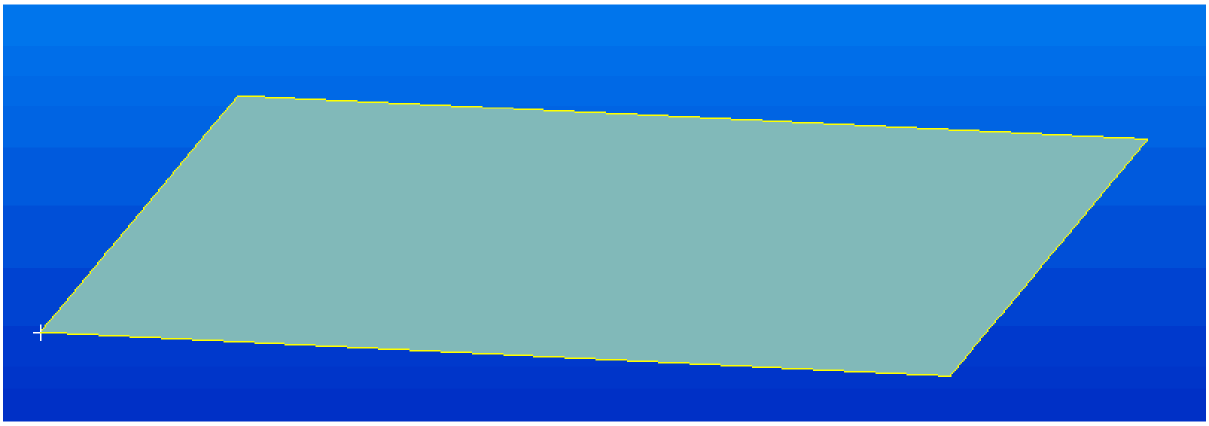


Fig.A4 Mat Plate

Click Apply button to see the Mat Plate (Fig.A4)

2) Creating the Spring

Create the Curve of Spring

On the Patran Main Menu, click on the Geometry Application button. (Fig. A1)

On the top of the Geometry form, select action>> Create, Object>>Curve, and Method>> Point. Curve ID >> 5. Option>> 2 Point, and Click Auto Execute Selection (Fig. A5)

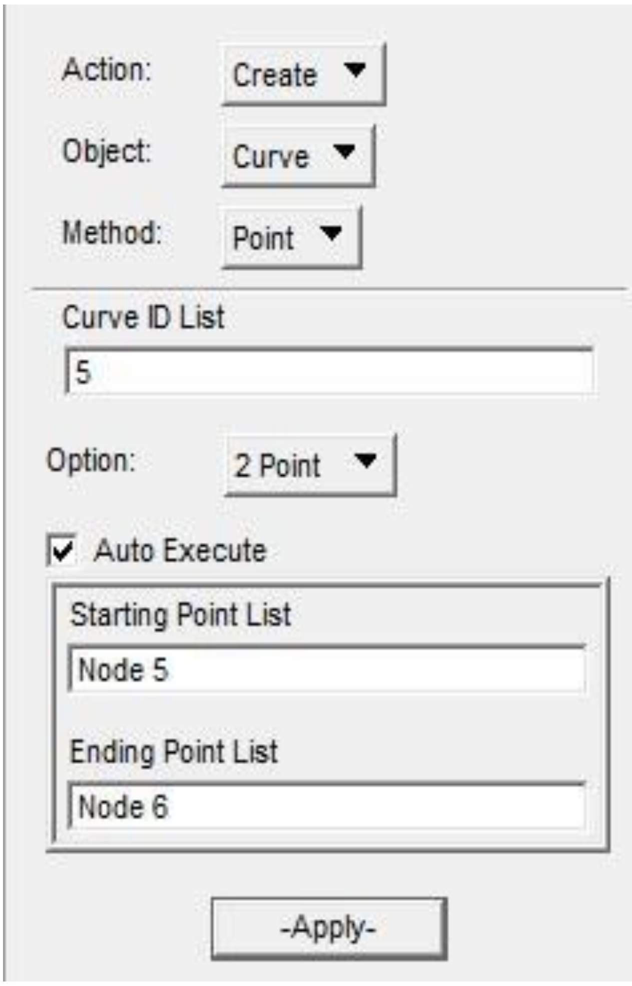


Fig. A5 Geometry form for Spring curve

Create the base of the Spring

On the Patran Main Menu, click on the Geometry Application button. (Fig.A1)

On the top of the Geometry form, select action>> Create, Object>> Point, Method>> XYZ, Point ID is 5, Refer. Coordinate Frame is Coord 0, Click Auto Execute Selection and Point Coordinates List Node 5. (Fig.A6)

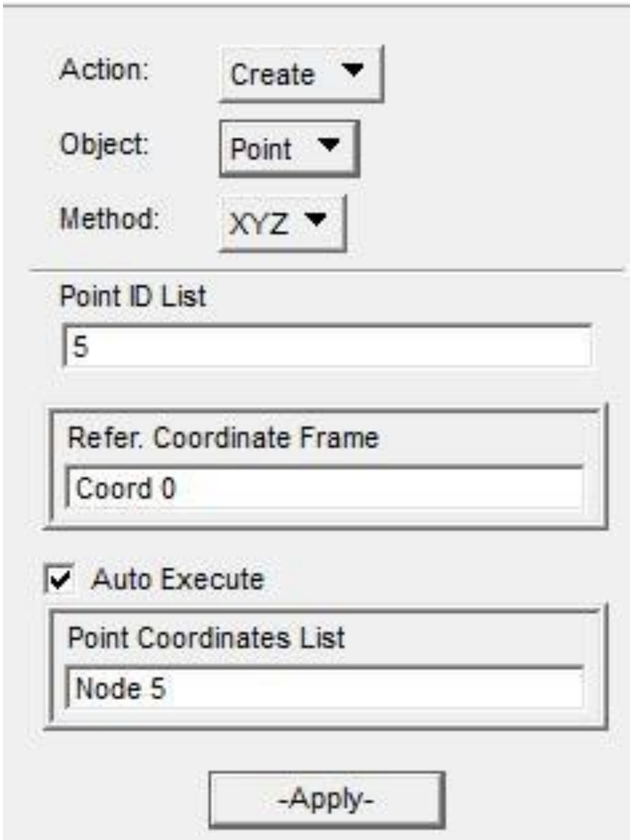


Fig.A6   Geometry form for spring base

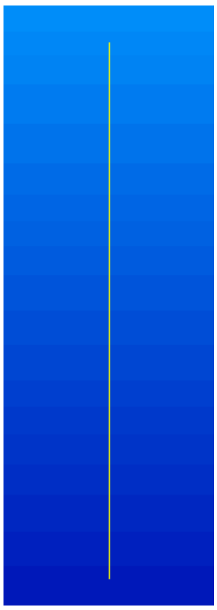


Fig.A7 Spring Model

Click Apply button to see the Spring (Fig.A7)

A.2 Elements creating

Similar to Geometry Modeling, Elements Modeling also has two parts: Mat Plate and Spring.

Creating Mat Plate Element

On the Patran Main Menu, click on the Elements Application button. (Fig.A8)

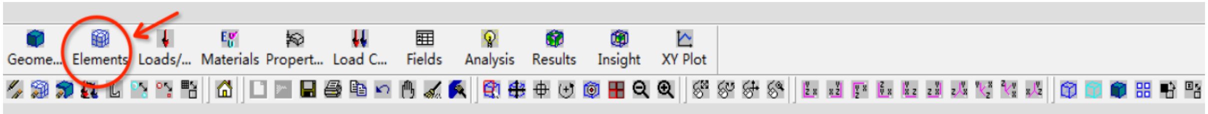


Fig.A8 Patran Main Menu for Element

On the top of Finite Element form, select Action>>Create, Object>>Mesh, Type>>Surface, Node ID List is 30413, Element ID List is 30036, Shape>>Quad, Mesher>>IsoMesh, Topology>>Quad4, Surface List Surface 1 And Click Automatic Calculation.(Fig.A9)

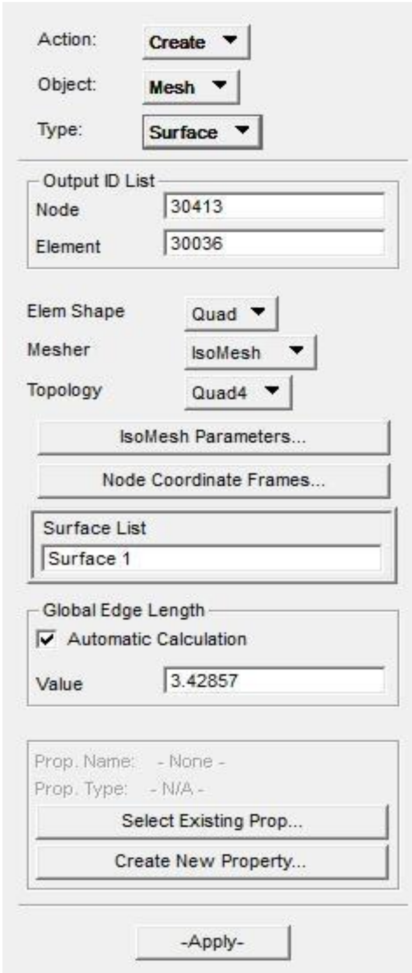


Fig.A9 Element form for Mat

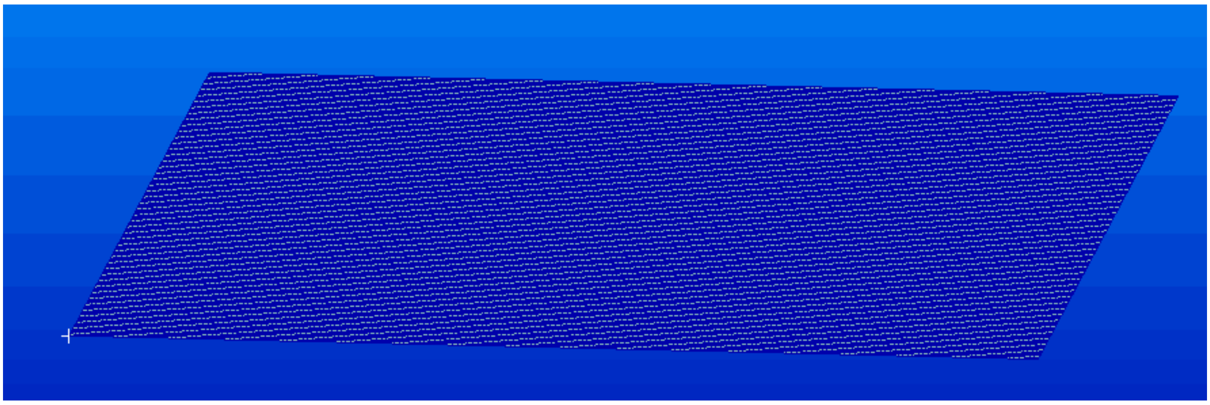


Fig.A10 Mat Plate Element

Click Apply button to see the Mat Plate Element. (Fig.A10)

Creating Spring Element

On the Patran Main Menu, click on the Elements Application button. (Fig.A8)

On the top of Finite Element form, select Action>>Create, Object>>Mesh, Method>>Curve, Element ID List is 30036, Node is 30413, Topology>>Bar2, Curve List is Curve 1, Click Automatic Calculation and Value is 2. (Fig.A11)

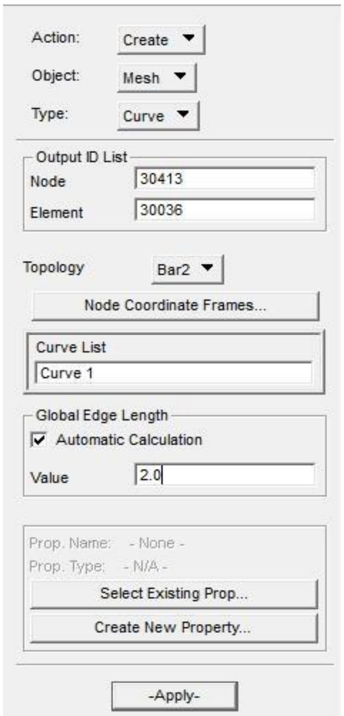


Fig.A11 Finite Element form for spring

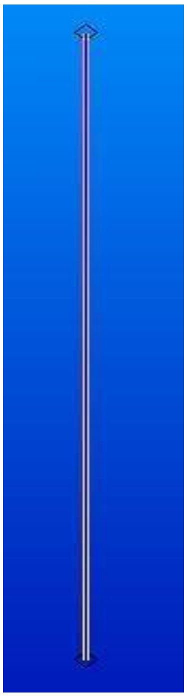


Fig.A12 Spring Element

Click Apply button, get the Spring Element. (Fig.A12)

A.3 Material modeling

Creating Mat Plate Materials

On the Patran Main Menu, click on the Materials Application button. (Fig.A13)

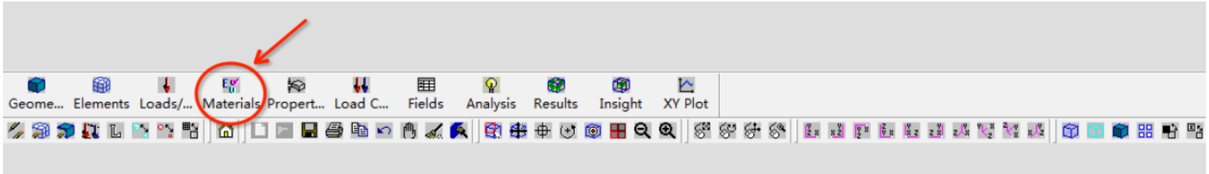


Fig.A13 Patran Main Menu for Materials

On the top of the Materials form, select Action>>Create, Object>>Isotropic, Method>>Manual Input, Existing Materials chose Plate and Material Name is Plate (Fig.A14)

Click on the Input Properties button. On the Input Options form, enter the Elastic Modulus data box, Poisson's Ratio data box and Density. (Fig.A15)

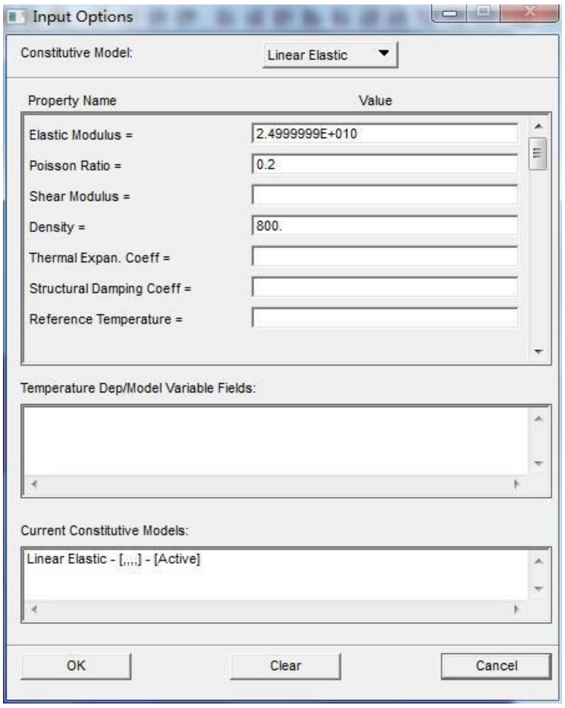


Fig.A14 Material form for mat

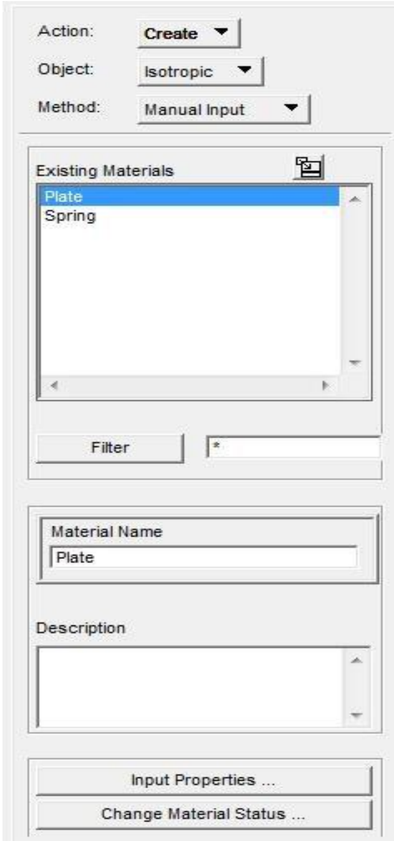


Fig.A15 Input form for mat

Click OK to close the Input Option form, and then click Apply on the Materials for data box.

Creating Spring Materials

On the Patran Main Menu, click on the Materials Application button. (Fig.A14)

On the top of the Materials form, select Action>>Create, Object>>Isotropic, Method>>Manual Input, Existing Materials chose Spring and Material Name is Spring. (Fig.A16)

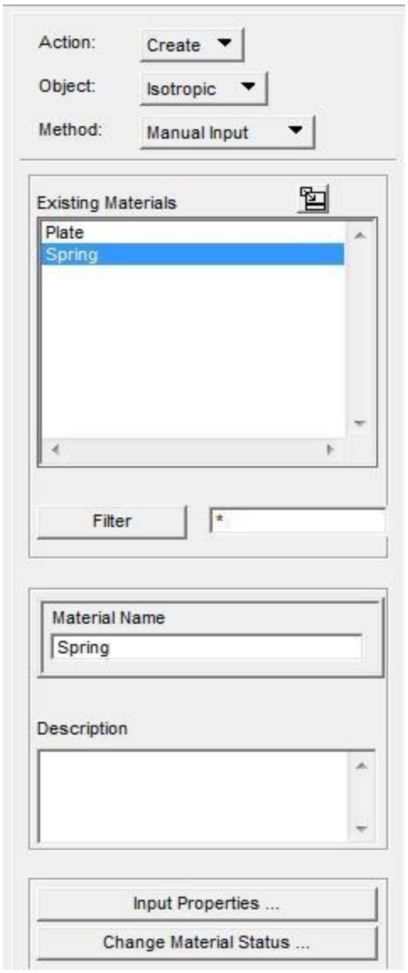


Fig.A16 Material form for spring

Click on the Input Properties button. On the Input Options form, enter the Elastic Modulus data box and Poisson's Ratio data box. (Fig.A17)

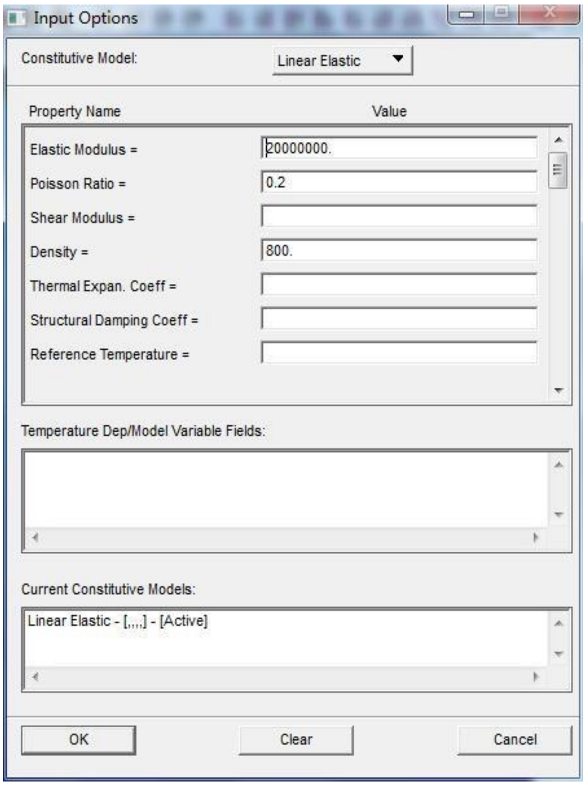


Fig.A17 Input form for springs

A.4 Define Element Properties

Creating the Element Properties of Mat Plate

On the Patran Main Menu, click on the Properties Application button. (Fig.A18)

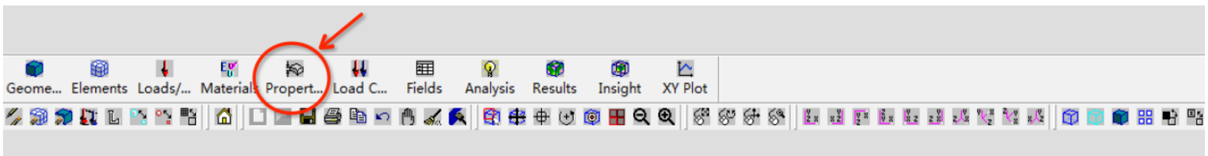


Fig.A18 Patran Main Menu for Properties

On the top of the Element Properties form, select Action >> Create, Object >>2D, Type >>Shell, Name is Plate, Option>> Thin, Homogeneous, and Standard

Foundation. (Fig.A19)

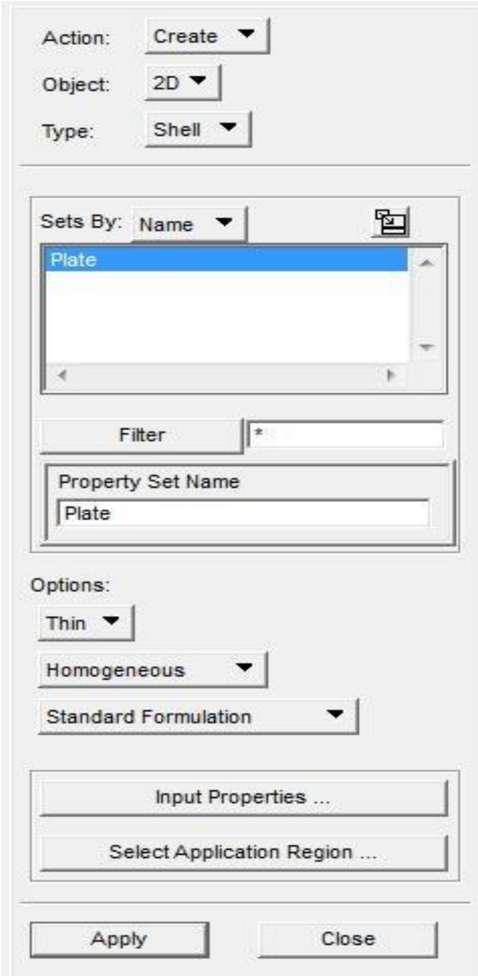


Fig.A19 Element Properties for mat

Click on the Input Properties. Click in the Material Name textbox and get the Thickness of the Mat. (Fig.A20)

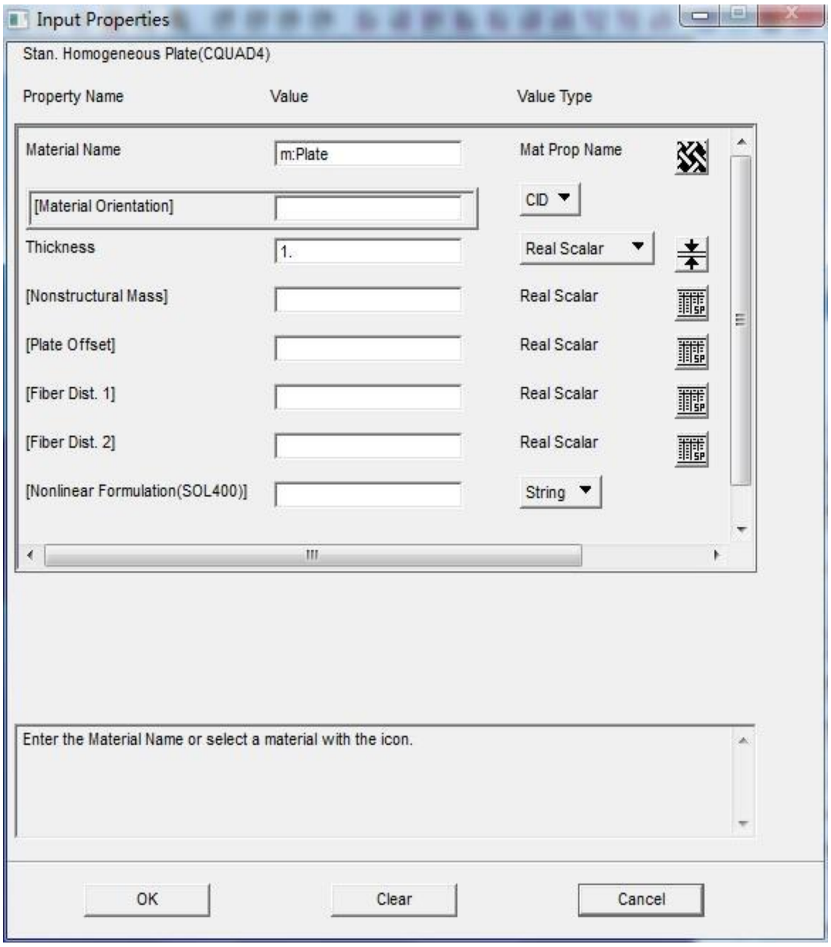


Fig.A20 Properties Input form for mat

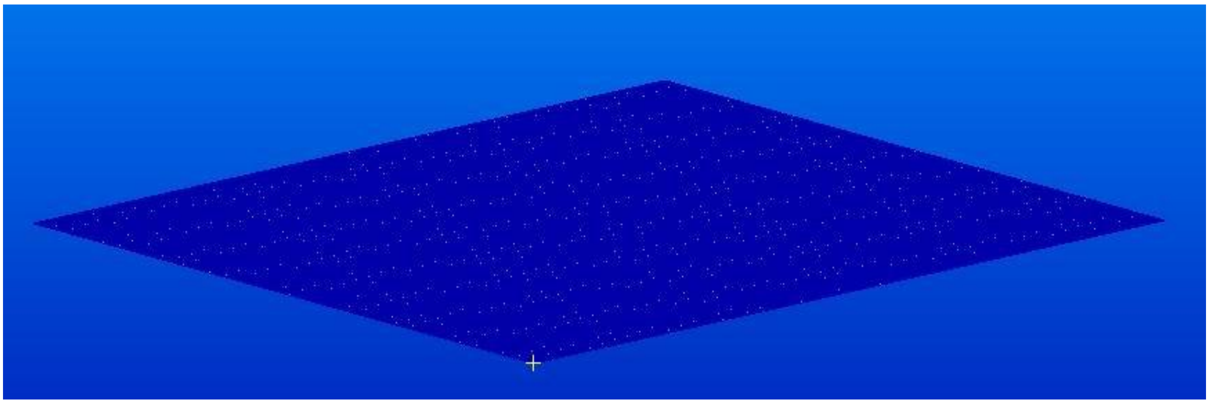


Fig.A21 Mat Model after Properties Step

Click OK and then Apply on the Properties form. (Fig.A21)

Creating the Element Properties of Mat Plate

On the Patran Main Menu, click on the Properties Application button. (Fig.A18)

On the top of the Element Properties form, select Action >> Create, Object >>1D, Type>> Beam, Name is Spring, Option>>General Section, and Standard Foundation. (Fig.A22)

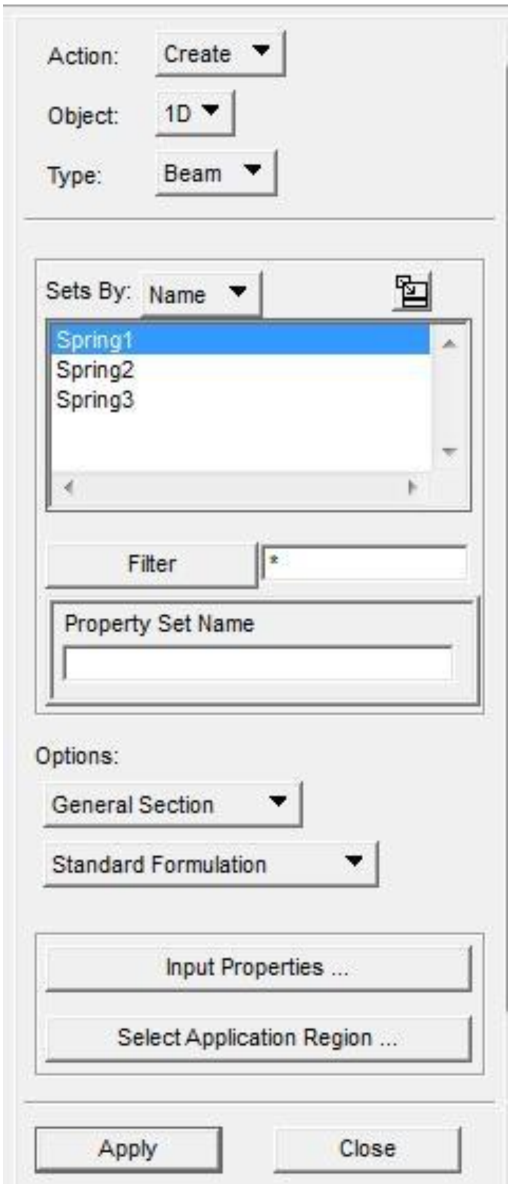


Fig.A22 Element Properties form for spring

Click on the Input Properties. Click in the Spring Constant textbox, Def at Node 1 and Node 2. (Fig.A23)

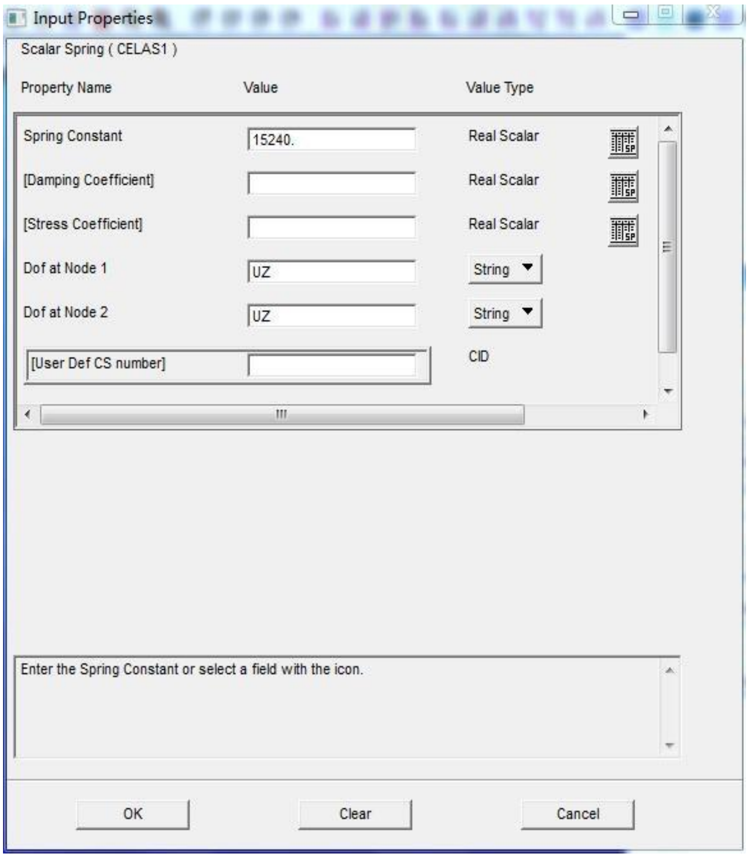


Fig.A23 Properties Input form for spring

Click OK and then Apply on the Properties form. (Fig.A24)

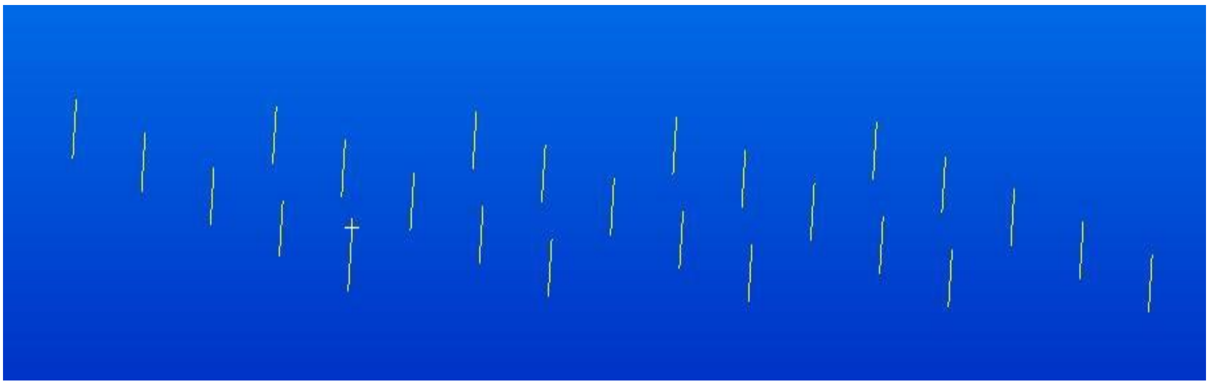


Fig.A24 Spring Models after Properties Step

Before simulate load and boundary condition, we need to connect spring models and mat plate together.

On the Patran Main Menu, click on the Elements Application button. (Fig.A8)

On the top of Finite Element form, select Action>>Create, Object>>MPC, Type>>RBE2, and Click in MPC ID enter 1. (Fig.A25)

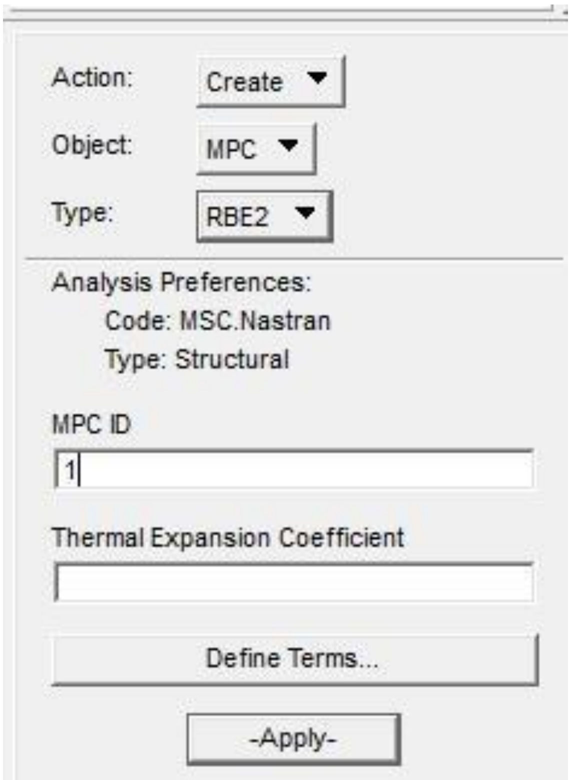


Fig.A25 Finite Element form connection

Click Apply button, get the Mat Foundation Model with springs. (Fig.A26)

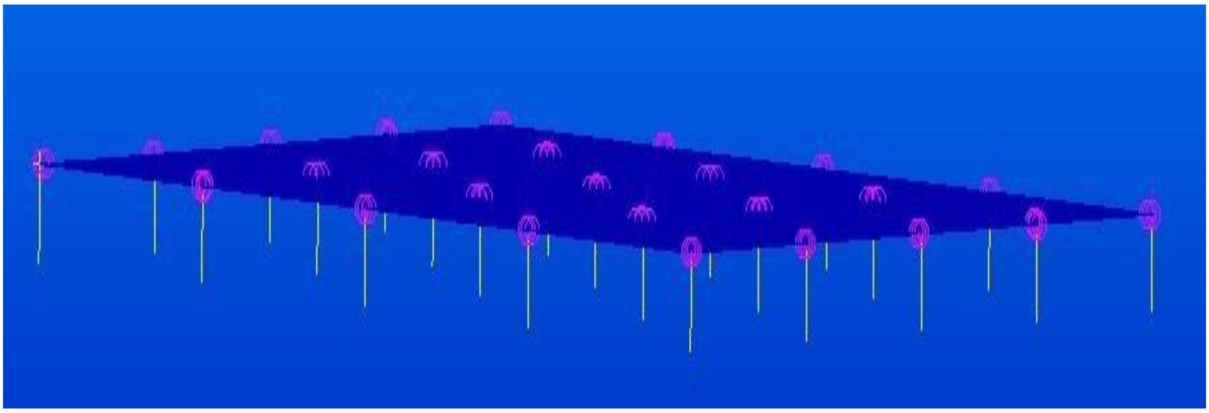


Fig.A26 Mat Foundation Model with springs

A.5 Simulating load and Boundary Condition

Create a Distributed Load

On the Patran Main Menu, click on the Loads/BCs Application button. (Fig.A27)

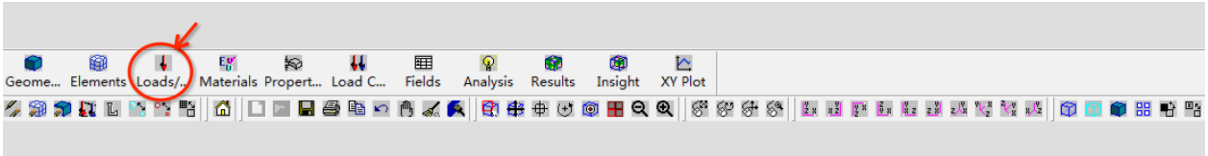


Fig.A27 Patran Main Menu for Load/BCs

On the top of the Loads/BCs form, select Action >> Create, Object >> Force, Type >>Nodal. New Set Name is F2. (Fig.A28)

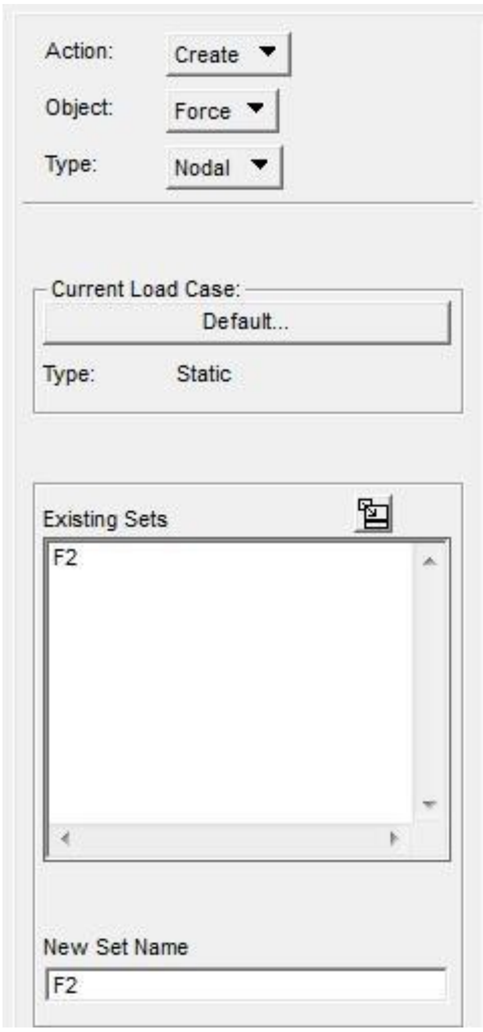


Fig.A28 Loads/BCs form for one load

Click on the Input Data button. On the Input Data form, enter Load/BC Set Scale Factor 1. Click in Force<F1 F2 F3> textbox <0,0,-1500000> (Fig.A29)

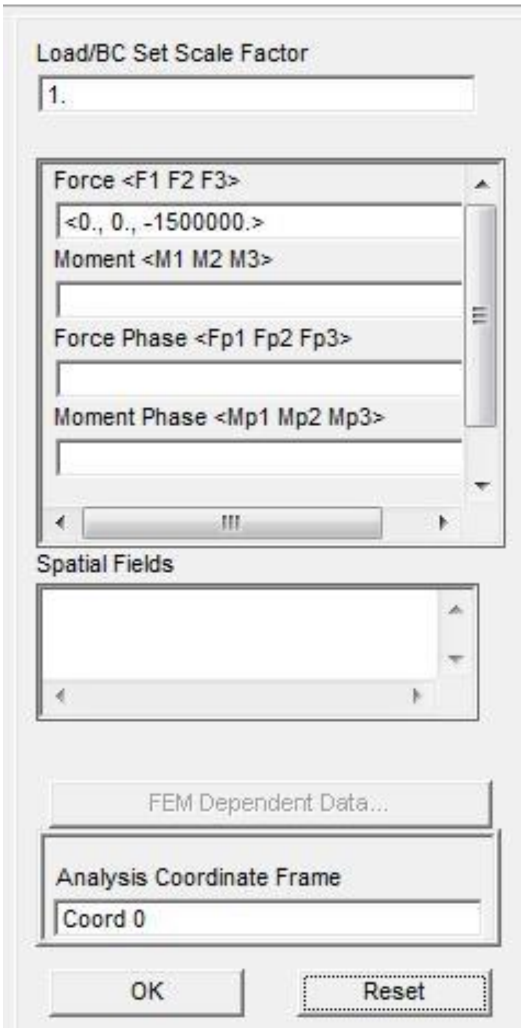


Fig.A29 Input Form for one load

Click OK and then Apply on the Loads/BC form. Get the Mat Foundation in Load. (Fig.A30)

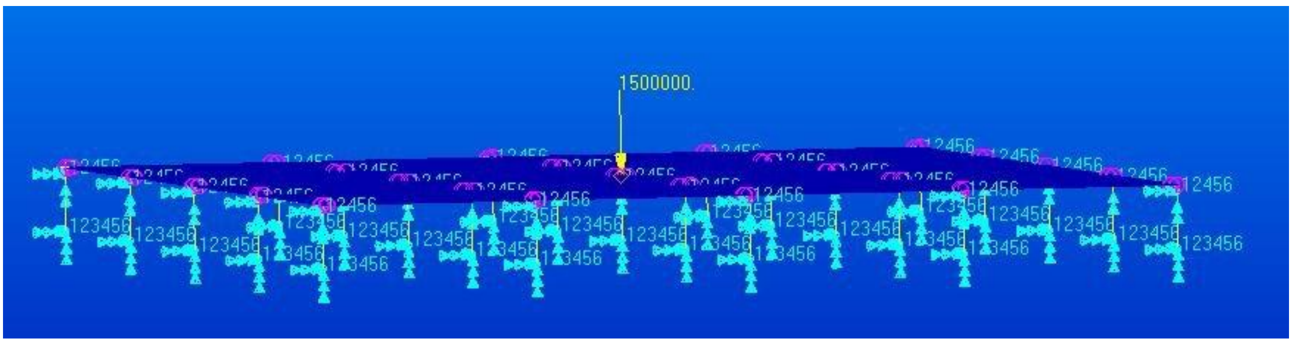


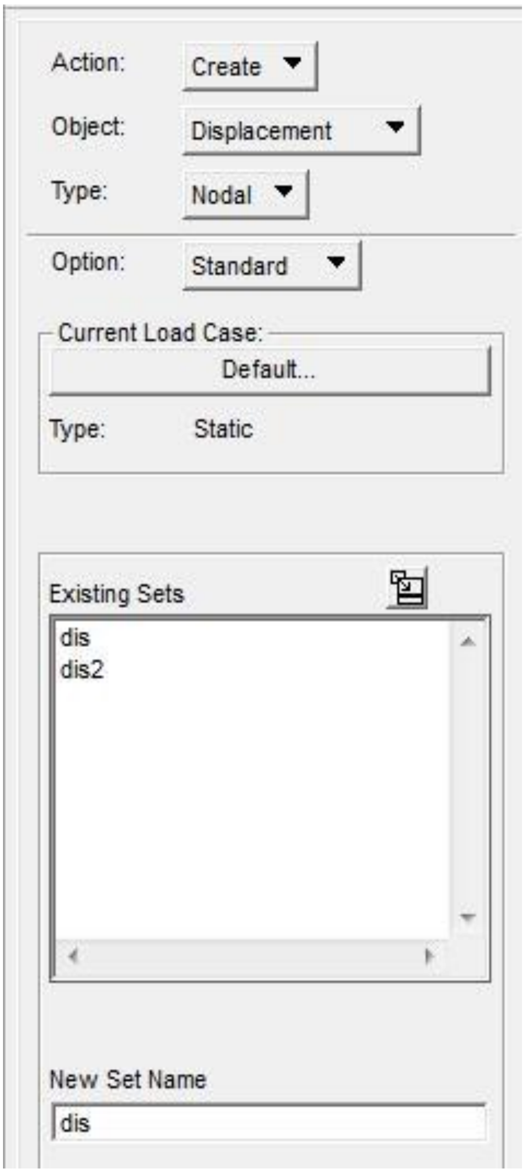
Fig.A30 the Mat Foundation in one load

Create a Constraining Condition

On the Patran Main Menu, click on the Loads/BCs Application button. (Fig.A27)

On the top of the Loads/BCs form, select Action >> Create, Object >> Displacement, Type >> Nodal. Option>> Standard. In the New Set Name textbox, enter dis. Click on the Input Data button. (Fig.A31)

Fig.A31 Loads/BCs Form for one load



The Input Data form, enter <0,0,0> for Translations and leave the Rotations field blank. Click OK. (Fig.A32)

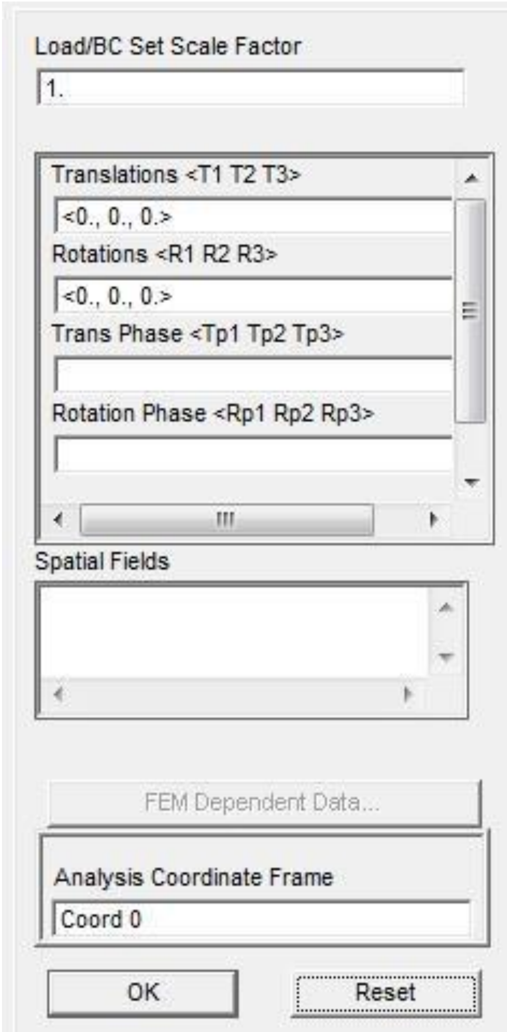


Fig.A32 Input Form for one load

Apply on the Loads/BC form. Get the Mat Foundation in Constraining Condition. (Fig.A33)

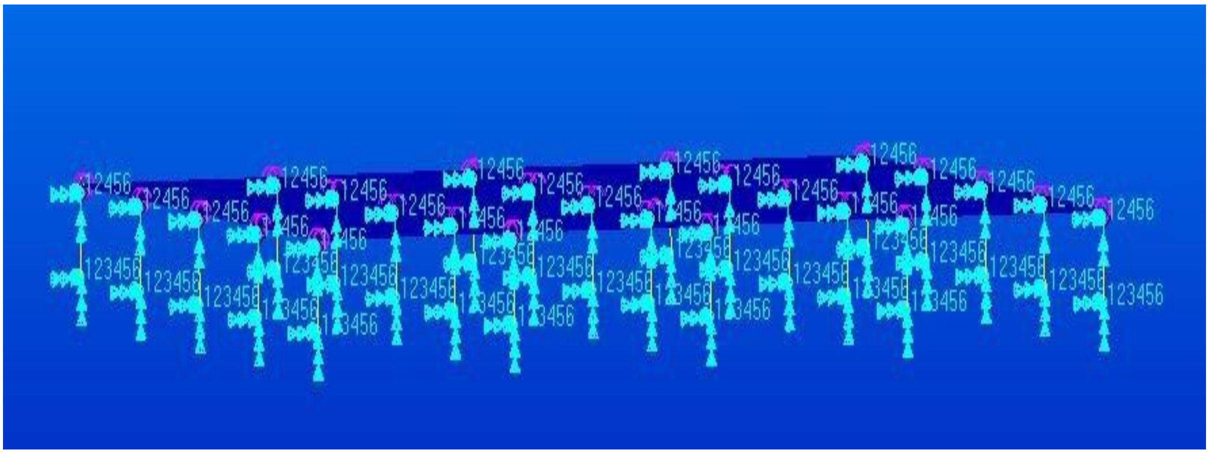


Fig. A33 Mat Foundation in one load Constraining Condition

A.6 Running the Finite Element Analysis

On the Patran Main Menu, click on the Analysis Application button. (Fig.A34)

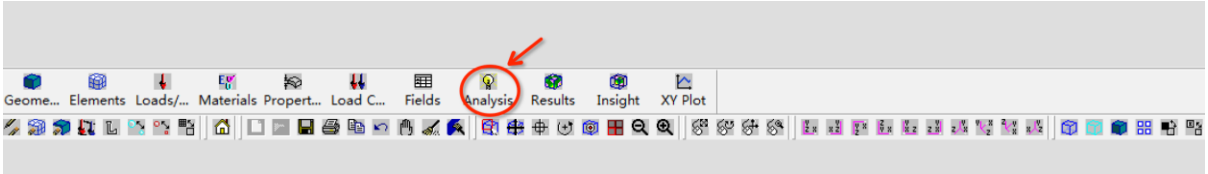


Fig.A34 Patran Main Menu

On the top of the Analysis form, select Action >> Analyze, Object >> Entire Model, Method >> Full Run. Code: MSC Nastran, Type: Structural. (Fig.A35)

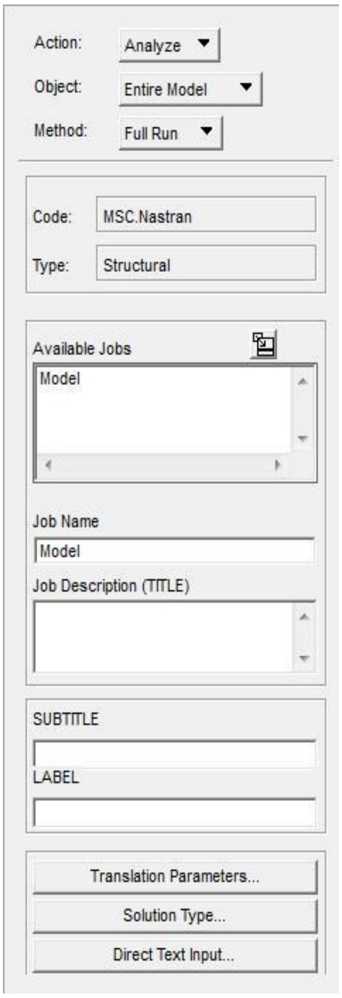


Fig.A35 Analysis Form for one load in analyze

Click in Solution Type, Chose Linear Static then click OK. (Fig.A36)

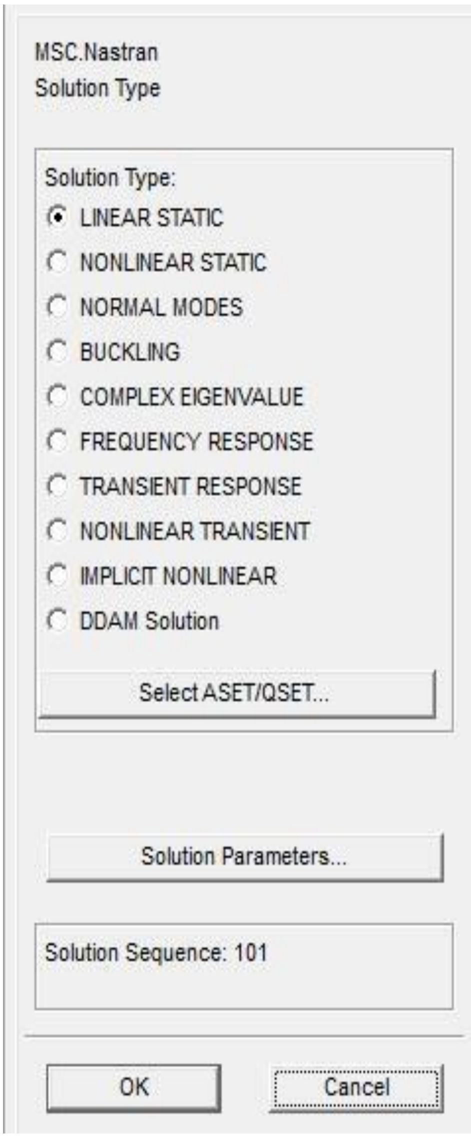


Fig.A36 Solution type for one load

After that click Apply. Then click on the Analysis again. On the top of the Analysis form, select Action >> Access Results, Object >>Attach XDB, Method >>Result Entities. Code: MSC Nastran, Type: Structural. (Fig.A37)

Click Apply, get the result.

Fig.A37 Analysis Form for one load in result

A.7 Compiling the Analysis Results

There are two different kinds of results: Force and Displacement.

Result of Displacement

On the Patran Main Menu, click on the Result Application button. (Fig.A38)

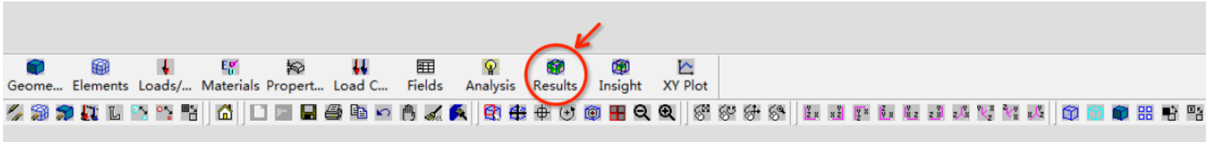


Fig.A38 Patran Main Menu for result

On the top of the Result form, select Action >> Create, Object >>Quick Plot, click Default, A2: Static Subcase in Select Result Cases, click Displacement, Translational in Select Fringe Result, select Quantity>>Magnitude and click Displacement, Translational in Select Deformation Result. (Fig.A39)

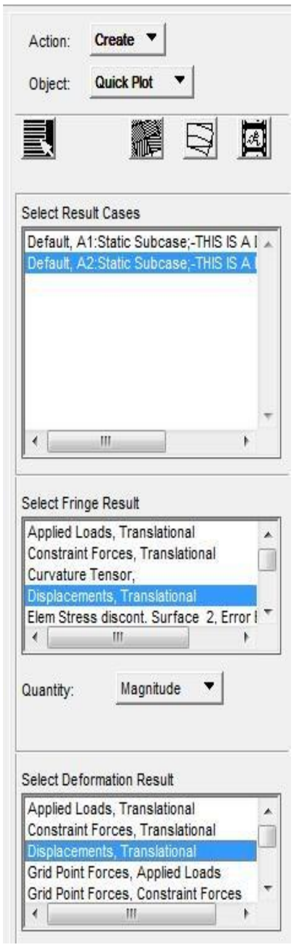


Fig.A39 Result form of displacement for one load

Click Apply, then get the result model. (Fig.A40)

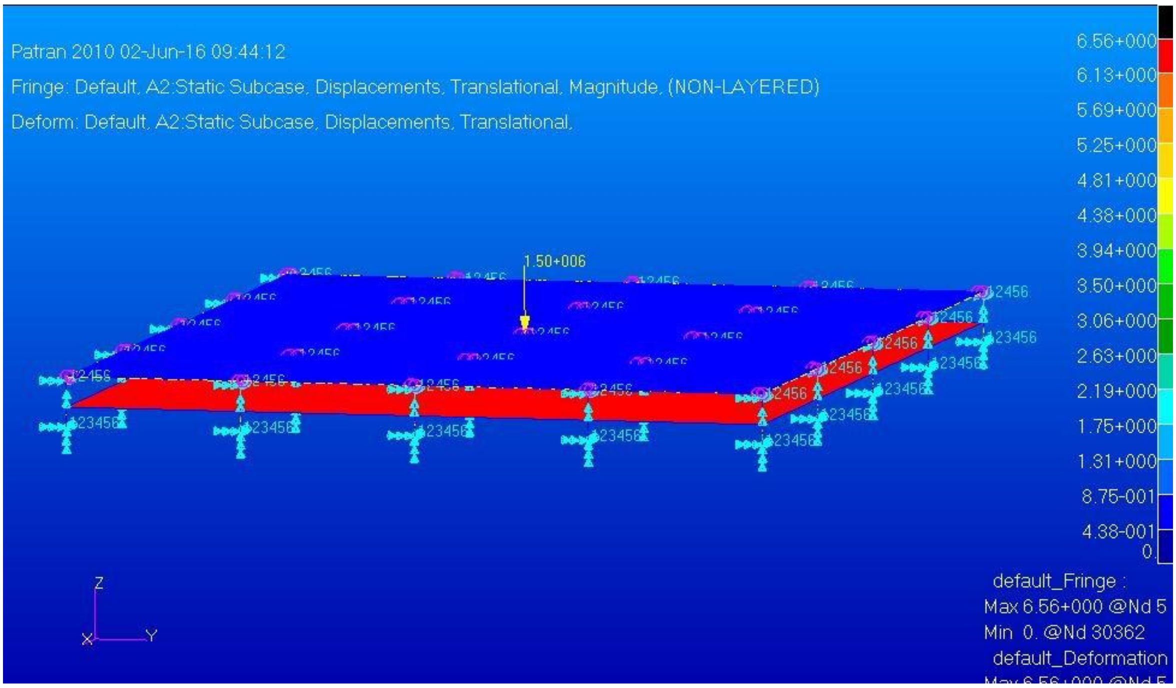


Fig.A40 Result Model of Displacement for one load

Result of Force

On the Patran Main Menu, click on the Result Application button. (Fig.A38)

On the top of the Result form, select Action >> Create, Object >>Quick Plot, click Default, A5: Static Subcase in Select Result Cases, click Stress Tensor in Select Fringe Result, select Quantity>>Magnitude and click Displacement, Translational in Select Deformation Result. (Fig.A41)

Click Apply, then get the result model. (Fig.A42)

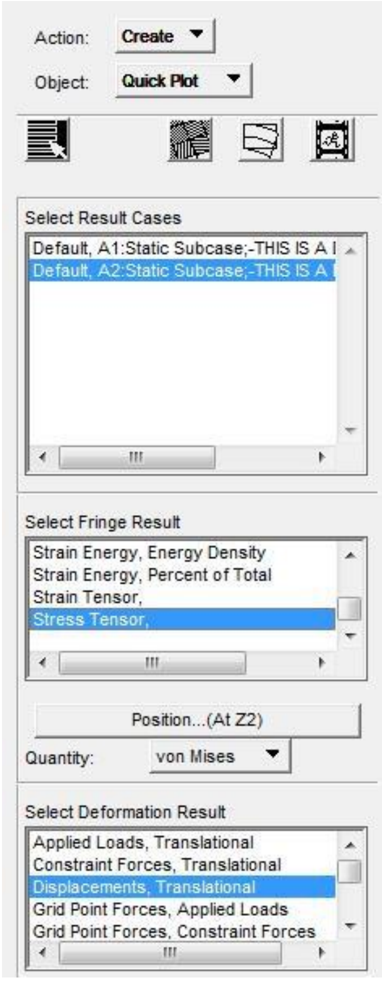


Fig.A41 Result form of Force for one load

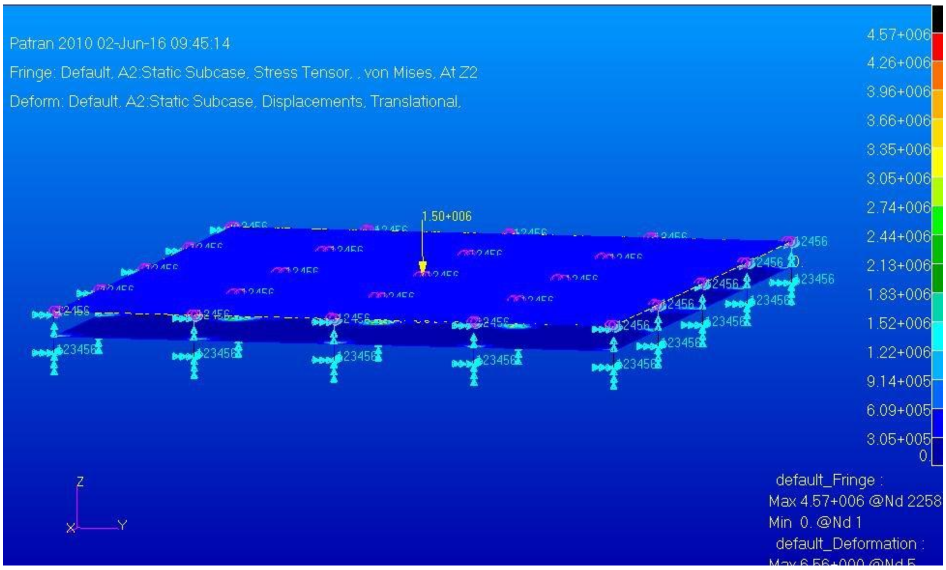


Fig.A42 Result Model of Force for one load

VITA

Yang Zhao was born on July 27th 1990, in China. Has received his Bachelor of Engineering in Civil Engineering in June 2013 From Xi'an University of Science and Technology.