INVESTIGATION OF STEEL COLUMN BASE PLATE UNDER MULTIAXIAL LOADING CONDITIONS

FINAL YEAR PROJECT UG 2016

By

Wajahat Saadat – 194896 (G-L)
Muhammad Salman Ahmed Abbasi - 195148
Jahangir Alam - 183076
Ferozudin Sumro - 182541

NUST Institute of Civil Engineering
School of Civil and Environmental Engineering
National University of Sciences and Technology, Islamabad, Pakistan

2020

This is to certify that the

Final Year Project Titled

INVESTIGATION OF STEEL COLUMN BASE PLATE UNDER MULTIAXIAL LOADING CONDITIONS

submitted by

Wajahat Saadat – 194896 (G-L)
Muhammad Salman Ahmed Abbasi - 195148
Jahangir Alam - 183076
Ferozudin Sumro - 182541

has been accepted towards the requirements
for the undergraduate degree

in

CIVIL ENGINEERING

____

Samiullah Khan Bangash
Lecturer
NUST Institute of Civil Engineering (NICE)
School of Civil and Environmental Engineering (SCEE)
National University of Sciences and Technology, Islamabad, Pakistan

ACKNOWLEDGEMENT

We would like to extend our deepest thanks to all those who had given us the opportunity to complete this report. Special thanks, given to our final year project advisor Mr. Sami Ullah Bangash. His contribution to suggestions and advocacy, has helped me to plan my project and especially for writing this report. Without his guidance the completion of this report would not have been possible.

TABLE OF CONTENTS

ABSTRACT

A technique is developed in this research which permits the derivation of the volume of the base of the column under the uniaxial and bi-axial combination of the forces. The process of obtaining a column base stiffness under an axis which is considered weak, bending is obtained by changing the structural path that already exists to fit such loading. The curve at the contact is formed using the major axis bending resistance. The model is built on software that was integrated with the base plate, column and base. Basically, two things were used i.e. concrete and steel. Subsequently these materials are combined together to form a single model using the mode. This model is provided by a set of standard conditions called Encastre in software where all six levels of freedom are limited. The luggage is then placed in a vertical model with a vertical seating or integrated load. This loading is placed in the column and its reaction to the metal plate and the concrete base is noted. The contact area is built between the two surfaces because the conditions only cover the arms but do not form any interaction between them. For this purpose, two types of land are used, one is called a bounded arch and the other is called an earth working tool which is also assigned a shooting factor of 0.3. For the accuracy of the results, all model items are transported and their size is as small as 0.04.

OBJECTIVES

The structural performance of the column bases under biaxial loading conditions is the main purpose of this analysis. So, an attempt was made to figure out the process, the use of which will permit the designers to find the biaxial joint bending moment. Therefore, the function is being divided in to the functions stated below:

- Review of the literature on column foundations and method (rigid axis);
- Search for available tests on column bases that are ready to be compared to object method.
- The development of a numerical model, which is verified in accordance with the test results, and compares the results which are obtained numerically with those which are being extracted by a method called the object method
- The creation of a collar of contact between the current forces, providing the resistance is the main purpose for the base of the column of any bend;
- By comparing the results with the numerical values, an analytical method can be validated.

SCOPE

Interaction between a number of components is the main problem in the metal structures, including the joints of the base as well as the beam to beam joints. A significant impact is caused by the mechanical properties of different joints on the behavior of the imposed structure, for this reason, the analysis of the structural framework is greatly impacted by their appearances and also the process of fabrication bears some impact as well.

With regard to the foundations of columns, they are the major components as their purpose is to move the loads of the material of the main structure into the system of foundation. However, still they the most studied building blocks. When compared with the column to column junction, where they exist in the form of tests which are more than one thousand, the tests performed on column bases are very limited.

It is quite impossible to predict beforehand that how the bases of the columns will perform if the panes are not bent. It is very wise to think in a way that the FEM or finite element modeling is used to integrate this model and at the end analyze it. These models which are based on FEM take much time as well as they are very complicated as well. So, the basic objective of this work is to develop an analytical approach which can inculcates the column bases resistance even during biaxial bending.

CHAPTER 1

INTRODUCTION

1.1 GENERAL

One of the most important properties of structures is the base plate in the column, especially when it is being designated to withstand moments cause by bending. The column loads should be passed through bolts and strategically mounted on the corresponding base. Besides its essential role, their design is not much desired. This may proceed to the expensiveness and inappropriate decisions. Improper solutions can also pose a serious threat to structural safety.

The foundation plate design is an integral part of any building project. All available biblical drawings related to the analysis and subsequent construction of the base plates look at the undesirable behavior. However, since most structural models are three-dimensional, the reactions obtained from the structure have moments in both axes, as shown in Figure 1. In this article the nonlinear methods are extended to deal with biaxial problems.

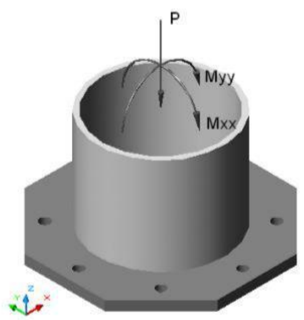


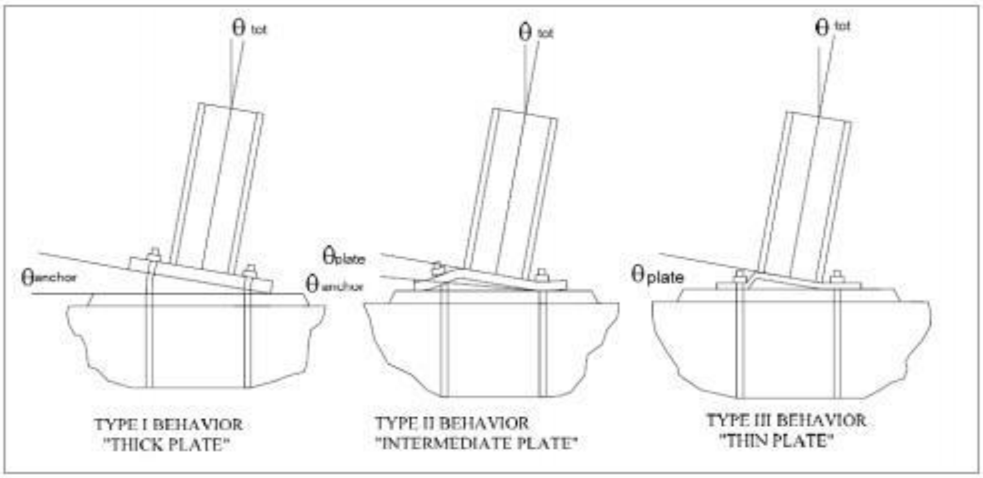
Figure 1. Biaxially loaded base plate

To solve biaxial problems a system of nonlinear equations is needed, since concrete can only handle compressive forces and only anchor rods can handle tension. Generally, the system variables are: high compressive concrete pressure, high anchor tension, neutral axis and bore length. The standard practice to fix high-pressure concrete compression is the approved construction code.

1.1.1 BASE PLATE

A metal sheet that has its adjustment to the column is called as the base plate. It is crucial for enhancing the area of the connection that exists between the column and the bridge made of concrete, which will reduce reflections in compression and prevent the pull of the concrete. Another purpose is the transit if the possible column differences to the milking sheets.

As far as their strength is concerned, the base plates are arranged according to size, or more than that which is generally required for a plastic hinge formation, on the plate. The figure shows 3 base plate types and also their distorted form.

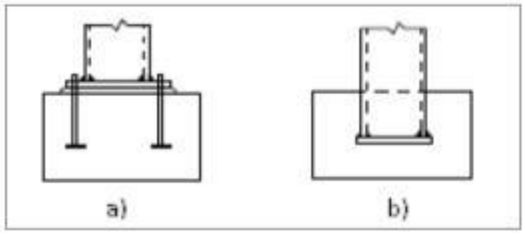


1.1.2 ANCHOR BOLTS

Anchor bolts are used for transferring the rigid loads to the base and in this way holding the column down to its place. These kinds of loads appear to be pure friction or tension which is caused by the one side bending of the column.

1.1.3 COLUMN BASES

It is a special kind of connection which has the responsibility to connect the base and the steel column and is performing the function of load transition between the supporting member and the supported member. Exposed base plate is the most common type of connection but it can also be used in accordance with concrete.

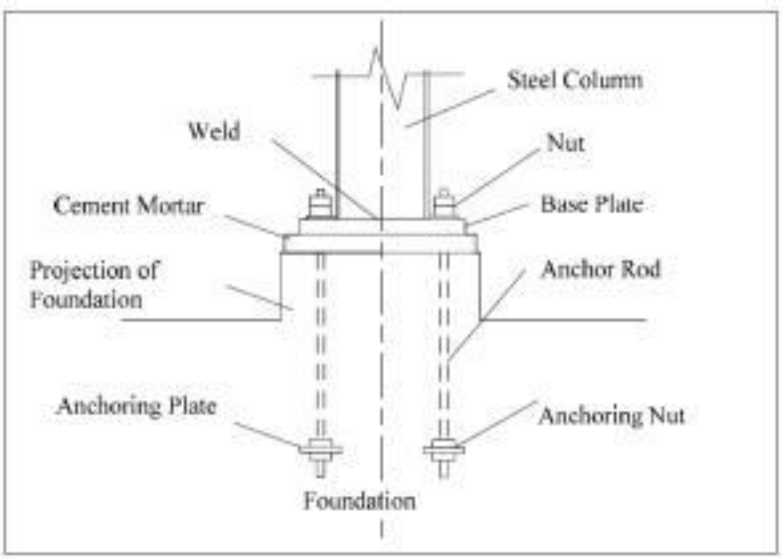


Column bases: a) Exposed column base, b) Embedded column [2]

The main features of column-supported plates, also called anchored base plate,

- column foot;
- base plate brought to the foot of the column;
- a layer of mud;
- holes with bases;
- concrete block (foundation).

Sometimes the combinations can be tightened using stiffeners. In addition, if necessary, the joint can be added with a shear lug combat key.



Chapter 2

LITERATURE REVIEW

2.1 DESIGN GUIDE

This work on the behavior of basic plates under biaxial bending includes many ideas proposed by different people but the basic guide for this study was the AISC design guide for the construction of basic plates and anchor rods. Other research papers were also submitted and provided raw information relevant to the project.

The column bases are generally designed to transfer the energy from the column to the base plate in a straight line. Grounding systems are designed to tighten the column during construction, and resist any lift in the threaded joints.

When the base plate is provided at the bottom of the column, the load is dispersed to a large area and subsequently when transferred to the concrete base so the system is safe

The base of the column briefly is used to distribute the load.

Table 2.1. Base Plate Materials

Thickness (t_p)	Plate Availability
$t_p \leq 4$ in.	ASTM A36 [a] ASTM A572 Gr 42 or 50 ASTM A588 Gr 42 or 50
4 in. < $t_p \leq 6$ in.	ASTM A36 [a] ASTM A572 Gr 42 ASTM A588 Gr 42
$t_p > 6$ in.	ASTM A36

[a] Preferred material specification

The size and dimensions of the base plate are specified by ACI.

In addition to the base of the plate other elements of the base column system are grout, anchor rods, nut and washer etc.

The general features of the column base are shown

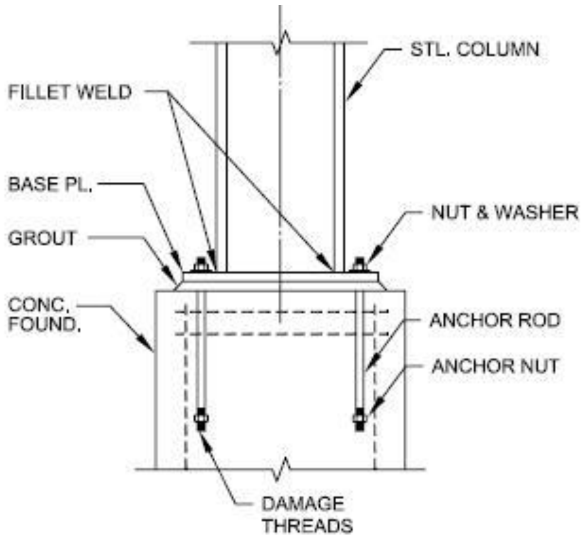


Figure 1.1. Column base connection components.

2.1.1 ANCHOR ROD:

Anchor rods, also called anchor bolts, concrete ribbons or base supports, are mounted on a concrete base to support structural steel columns, light beams, road signs, signposts, industrial equipment and many other systems.

2.1.1.1 Anchor Rod Sizing

Designed on the basis of ASTM F1554, with most grade 36 steel. Areas where more power is required, the rod size has increased to two inches before switching to a new one. The anchor rod layout should be as precise as possible in both directions. The standard dimensions of a stick are four rods of a square pattern.

2.1.1.2 Anchor Rod Placement and Tolerance

Anchor rods are used for safe, rich and fast steel frame construction. The placement begins with a drawing of the placements made by an experienced financial auditor. When the anchor rods are placed a tolerance, check is performed. An 11-inch precision tolerance, latest compatibility with the level. Tolerance is defined by ACI.

2.1.2 COLUMN ERRECTION:

2.1.2.1 Procedure

There are three methods of setting elevation.

- Nuts and washers setting
- Plates setting
- Shim-stakes

In the first method we use four rods as it is an independent environment. This method is simple and does not work effectively. There is less chance of interference once it is set. This method is used when the column is loaded slowly during creation.

In the second method we use a placement plate, whose size is limited to 24 inches. It is more expensive than nuts and washers. Setting plates are usually ¼” firm and slightly larger than the base plate. After placing the anchor rods, the mounting plate is removed.

While the latter method is the traditional way of setting the plate height. The size of the metal shim is 4 "wide at the four ends of the base plate.

2.1.3 REPAIR OF ANCHOR RODS:

Anchor rods need to be fixed when there is a problem in its assembly and some problems are discussed below.

2.1.3.1 Anchor rods bend or not vertical

The band is limited to 45 degrees or less. The walls up to 1 inch can rot in cold and those over 1 inch are heated up to 1200-degree Fahrenheit to make the bend.

2.1.3.2 Anchor rods projections too long or too short

When the prediction is too short the solution digs another hole and releases another anchor of the epoxy type. And if it is too far to embed it may not be enough to develop stagnant energy.

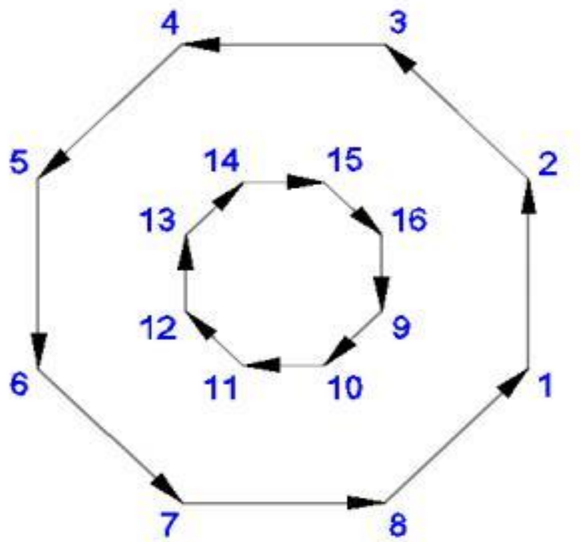
Chapter 3

METHODOLOGY

3.1 METHOD DEVELOPMENT

To develop the methodology, we will take as an example an octagonal base plate with an inscribed circle radius of 10 in (8.28 +in per side approx.). An opening is made, also octagonal, with an inscribed radius of 4 in (3.31 in per side approx.). 8 anchor rods of 5/8” are included symmetrically in a circular placement with an 8.5 in circle radius.

An important step is the borders definition. The external borders are defined counterclockwise and the openings clockwise. Then, each line segment is taken in the same way for all defined borders.



Linear models are proposed for concrete compressive stresses and anchor rods tension forces. The next step for numerical solution is to transform the base plate border and anchorage geometry to a local coordinate system.

where,

T = maximum tension force on anchor rods.

Ơc = maximum compressive stress = neutral axis (NA) angle.

Y = bearing length.

f = maximum anchor rod distance from NA.

The three variables of the biaxial problem are: bearing length, maximum anchor rod tension and the neutral axis angle. A simplification is to consider that the maximum concrete stress reaches the maximum concrete bearing strength by the adopted design code.

$$\phi f_{p(max)} = \phi 0.85 f_c' \sqrt{\frac{A_2}{A_1}}$$

where,

fp(max) = maximum design bearing strength,

Section 10.14, ACI 318-08 = 0.65 is the compression resistance factor which is taken by the section 9.3 of ACI 318_08

Fc'= compressive strength of concrete

A1 = Base plate area

A2 = supporting surface maximum area which is exactly alike in geometry to the loaded area.

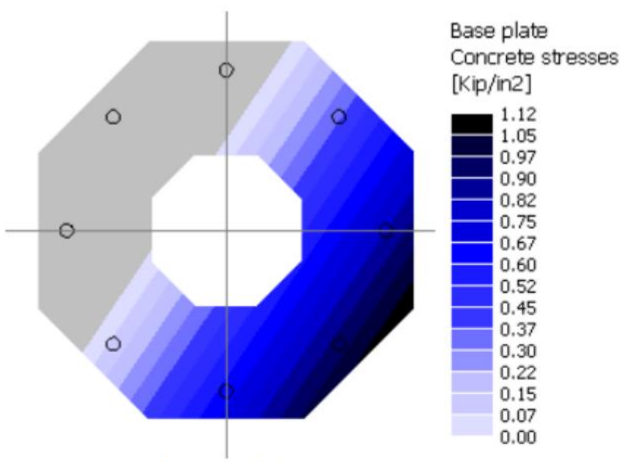


Figure 4. Concrete stresses

3.2 DESIGN OF BASE PLATE:

There are five types of loads acting on base plate.

3.2.1 CONCENTRIC COMPRESSIVE AXIAL LOAD

Where the base of the column is opposed to the vertical axial loads only, the base plate must be large enough to withstand the carrying force transferred to the base plate (concrete limit), and the base plate must be of sufficient size (base plate release limit

The concrete strength design is defined in ACI 318-02, Section 10.17, as φ (0.85fc'A1) where the support surface is no larger than the base plate. Where the supporting area is wider on all sides than the loaded area, the above-ground capacity is allowed to be increased by

(A2 / A1) ^ 1/2 ≤ 2.

Pp = 0.85fc′A1 on support of concrete full area

3.2.2 DESIGN FOR SHEAR

There are three main ways to transfer a piece from the base plates to the concrete:

Filling between base plate and grout or concrete.

The shear strength can be calculated in accordance with ACI criteria,

$$\phi V_n = \phi \mu P_u \leq 0.2 f_c' A_c$$

The friction coefficient μ is 0.55 for steel on grout, and 0.7 for steel on concrete.

Column and base plate bearing against the surface of the concrete:

In summary, the lateral resistance can be expressed as

$$\phi P_n = 0.80 f_c' A_l + 1.2(N_y - P_a) \text{ for shear lugs}$$

$$\phi P_n = 0.55 f_c' A_{brg} + 1.2(N_y - P_a) \text{ for bearing on a column or the side of a base plate}$$

For tightly loaded structures, when the force is greater than the output is not necessary and it is acceptable to build in the limited or shear strength of the anchor rod group which controls the formation. Frames designed for lateral seismic load resistance are expected to behave in a discreet manner and in this case, it may be necessary to design the base and connection of the base plate so that the center limits the frictional nature and the anchor rod group's core does not control construction.

Shear strength can be transferred in accordance with the use of shear lugs or by attaching a column to the base.

The shear contribution should be based on the most undesirable arrangement of the compressed load in accordance with the lateral forces being evaluated.

3.3 METHOD OF CODE DEVELOPMENT

The code actually deals with 3 kinds of Column Sections:

- W Sections
- Rectangular HSS
- Circular HSS

There is the number of parameters which should be obtained directly from the user and these parameters will be

- Breadth
- Depth
- Thickness
- Anchor Rod
- coordinates
- Depth of web
- Breadth of flange
- Moment of inertia
- Thickness of flange

The input data regarding the loading conditions will be

- Applied Load (Pu)
- Eccentricity (e)

Now, when all the dimensions and other parameters were being defined by the user, other parameters were calculated:

- Area of the box plate = B*N
- Area of the anchor rod = Pi*(d2/2)
- Area of the mesh block=(B/100) *(N/100)

Considering the value of applied load:

a) If the applied is negative then it will be the case of pure tension.

b) If the applied is positive then it will be one of the following 3 cases:

   i) Pure compression

   ii) Uniaxial bending

      (1) Small moment

      (2) Large moment

   iii) Biaxial bending

3.3.1 PURE TENSION

In pure tension case we have to verify the two checks one check is of concrete and the other one is for the anchor rods as:

3.3.1.1 Anchor Rod yielding

If;

Pu < 4*ϕ*fy*Anchor rod

Where ϕ = 0.75,

Then the check satisfies otherwise it fails.

3.3.1.2 Concrete pullout strength

If;

Pu < ϕ*8*fc'*A bearing

Where ϕ = 0.75,

Then the check satisfies otherwise it fails.

3.3.2 PURE COMPRESSION

If eccentricity entered by the user is 0 then that would be the case of pure compression. So, in case of Pure Compression 3 cases were being checked:

3.3.2.1 Concrete Bearing Fail

If;

Pu < ϕ*0.85*fc'*A Plate

Where ϕ = 0.6,

Then the check satisfies otherwise it fails.

3.3.2.2 Base Plate Yielding

If;

Pu < ϕ*fc'* A Plate

Where ϕ = 0.9,

Then the check satisfies otherwise it fails.

3.3.2.3 Minimum thickness

Thickness of the plate i.e.

t > l (2*Pu/ (ϕ*fy* A Plate))1/2

Where;

ϕ = 0.9,

l = max (m, n, λn'), where values of m and n are different for different sections and λn' can be calculated by the formula as:

λn' = (λ*(d*bf)1/2)/4

Λ = 2x1/2/(1+(1-x)1/2) <= 1

Then the check satisfies otherwise it fails.

3.3.3 UNIAXIAL BENDING

In uniaxial bending, the bending would only be in one direction only and there would be only one moment in that particular direction.

For small moments we have to apply a condition on eccentricity which is:

If;

e < N/2 - Pu/ (2*qmax)

then the structure is experiencing the small moment uniaxial bending.

Where,

qmax = Fp*B;

Fp = Pu/(B*Y);

Y = N -2*e;

For simplification the above-mentioned code is also explained in the flow charts in appendix.

3.4 NEWTON-RAPHSON METHOD

The main purpose to include the Newton Raphson Method in our project is that this method is used by many of the popular software like RAM Connections use this method for finding out the most efficient values of different variable which are very important for analysis of any structure. We have studied this method in detail and now we'll define how this method can be used to find the values of the important variables for analysis of any structure.

3.4.1 INTRODUCTION

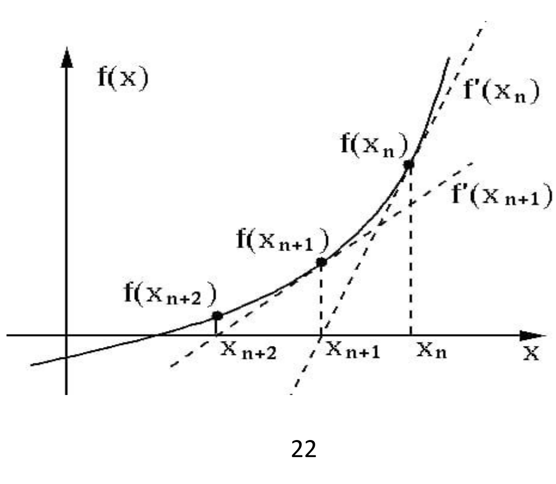
Newton Raphson method is a mathematical method by which we get closer and closer to the root of a given function by applying the following formula (with each iteration)

$$x_{n+1} = x_n - \frac{f(x_n)}{f'(x_n)}$$

Where;	Xn	=	The x value of nth iteration
	Xn+1	=	The value of x in the (n+1) th iteration
	f(Xn)	=	The value of function at the value Xn
	f '(Xn)	=	The value of derivative of function at X = Xn

3.4.2 GRAPHICAL INTERPRETATION

So above expression gives us the value of Xn+1 which is the value of x at which the line tangent to the original curve at X=Xn intersects at X-axis. Then the process is repeated to fin the tangent line at X=Xn+1 and it intersects X axis at X=Xn+2 and so on giving us the value of X closer and closer to the root of the original equation (where it intersects x-axis) as shown in the figure below.



3.4.3 THE MATLAB CODE

The MATLAB code of Newton Raphson Method for the function of y=cos(x) from 0 to pi is given below:

```
% Newton-Raphson Algorithm
% Find the root of y=cos(x) from o to pi.

f = @(x) (cos(x));
fd = @(x) (-sin(x));
p0 = input('Enter initial approximaation: ');
n = input('Enter no. of iterations, n: ');
tol = input('Enter tolerance, tol:  ');

i = 1;
while i <= n
    d=f(p0)/fd(p0);
    p0 = p0 - d;
    if abs(d) < tol
        fprintf('\nApproximate solution xn= %11.8f \n\n',p0);
        break;
    else
        i = i+1;
    end
end
```

3.4.4 NEWTON RAPHSON IN MULTI-DIMENSIONAL CASE:

The newton Raphson Method is applied in three-dimensional case with similar concept but this time the calculation takes us to matrices and we calculate Jacobian matrix to solve for the variables.

Multidimensional case: $\mathbf{f}(\mathbf{x}) = \mathbf{0}$

$$\mathbf{x}^{(k+1)} = \mathbf{x}^{(k)} - \left[\mathbf{J}\left(\mathbf{x}^{(k)}\right)\right]^{-1}\mathbf{f}\left(\mathbf{x}^{(k)}\right)$$

where:

$$\mathbf{J}(\mathbf{x}) = \begin{bmatrix} \frac{\partial f_1(\mathbf{x})}{\partial x_1} & \frac{\partial f_1(\mathbf{x})}{\partial x_2} & \cdots & \frac{\partial f_1(\mathbf{x})}{\partial x_n} \\ \frac{\partial f_2(\mathbf{x})}{\partial x_1} & \frac{\partial f_2(\mathbf{x})}{\partial x_2} & \cdots & \frac{\partial f_2(\mathbf{x})}{\partial x_n} \\ & & \cdots & \\ \frac{\partial f_n(\mathbf{x})}{\partial x_1} & \frac{\partial f_n(\mathbf{x})}{\partial x_2} & \cdots & \frac{\partial f_n(\mathbf{x})}{\partial x_n} \end{bmatrix} \Rightarrow \text{JACOBIAN MATRIX}$$

$$\mathbf{J}\left(\mathbf{x}^{(k)}\right) = \mathbf{J}(\mathbf{x})\big|_{\mathbf{x}=\mathbf{x}^{(k)}}$$

3.4.5 USE OF NEWTON RAPHSON METHOD TO FIND STRESSES AND MOMENTS

So now we will look into the type of matrix formed when we are talking about bi axial bending in base plates and are discussing about forces and moments. The below equations are valid when a base plate is acted upon by an applied force P and which induces tensional force in anchor rods, stresses and moments about x and y axis. (taken from the paper published by Mr. Gonzallo Orallana).

$$P - \sum_{i=1}^{n} P_i - \sum_{i=1}^{m} T_i = 0 \qquad (2$$

$$M_{xx} - \sum_{i=1}^{n} M_{ixx} - \sum_{i=1}^{m} T_i Y_i = 0 \qquad (3$$

$$M_{yy} - \sum_{i=1}^{n} M_{iyy} - \sum_{i=1}^{m} T_i X_i = 0 \qquad (4$$

where,

P	=	applied axial force.
M_{xx}	=	applied moment about X global axis.
M_{yy}	=	applied moment about Y global axis.
P_i	=	axial force for segment i.
M_{ixx}	=	moment about X global axis for segment i.
M_{iyy}	=	moment about Y global axis for segment i.
n	=	total segment number.
T_i	=	force tension in anchor rod i.
(X_i, Y_i)	=	global coordinates of anchor rod i.
m	=	total anchor rods number.

These equations can further be derived into the following form with some assumptions and can be solved by using numerical methods:

$$\begin{bmatrix} \frac{\partial P}{\partial \sigma_c} & \frac{\partial M_{xx}}{\partial \sigma_c} & \frac{\partial M_{yy}}{\partial \sigma_c} & \frac{\partial \sigma}{\partial \sigma_c} \\ \frac{\partial P}{\partial \theta} & \frac{\partial M_{xx}}{\partial \theta} & \frac{\partial M_{yy}}{\partial \theta} & \frac{\partial \sigma}{\partial \theta} \\ \frac{\partial P}{\partial Y} & \frac{\partial M_{xx}}{\partial Y} & \frac{\partial M_{yy}}{\partial Y} & \frac{\partial \sigma}{\partial Y} \\ \frac{\partial P}{\partial T} & \frac{\partial M_{xx}}{\partial T} & \frac{\partial M_{yy}}{\partial T} & \frac{\partial \sigma}{\partial T} \end{bmatrix} \begin{bmatrix} \delta\sigma_c \\ \delta\theta \\ \delta Y \\ \delta T \end{bmatrix} = - \begin{bmatrix} P \\ M_{xx} \\ M_{yy} \\ \sigma \end{bmatrix}$$

This was the whole concept of how Newton Raphson Method is being used by top software to find the values of variables like moments, stresses and forces.

*Pictures are taken from paper by Orallana and Google images.

3.5 ABAQUS MODELLING AND ANALYSIS

Nowadays, a number of different software are available in the market in order to do the modelling as well as the analysis at the later stages like ETABs, SAP2000, SOLIDWORKS, ANSYS etc. but we chose ABAQUS for this particular job this is because we are dealing with baseplates and baseplates are analyzed perfectly in ABAQUS with its meshing size as predefined and it gives an intensive detail of even a single element of the model.

3.5.1 ABAQUS

ABAQUS is a software system for software analysis and computer-assisted engineering, first released in 1978. The name and logo of this software is based on the abacus calculation tool. The Abaqus product site contains five basic program products.

- Abaqus / CAE
- Abaqus / General
- Abaqus / Specified
- Abaqus / CFD
- Abaqus / Electromagnetic

The complete content analysis consists of 3 different sections:

- **Preprocessing or Modeling:** This section includes the creation of an input file that has a developer design for input analysis (also called "solver").
- **Processes or Processes Analysis:** This section presents a visual output file.

- **Postage or old report, photo, animation, etc. In Output File:** This section is the visual output category.

3.5.2 MODELLING

The first stage in any project is the modeling stage. In this stage a number of steps have been followed which will be described with respect to the model of the steel base plate, steel column, concrete foundation and the steel anchor bolts which we had constructed and organized using different tools of the software.

3.5.2.1 Parts

The model constitutes many individual parts including the steel base plate, steel column, concrete foundation and the steel anchor bolts. All of these can be made on ABAQUS or can be imported from the CAD file which in our case were created on the software. The dimensions of the parts were according to the design file which was being discussed in the literature review part.

All of these parts are made using the certain properties, the details of which are described as under:

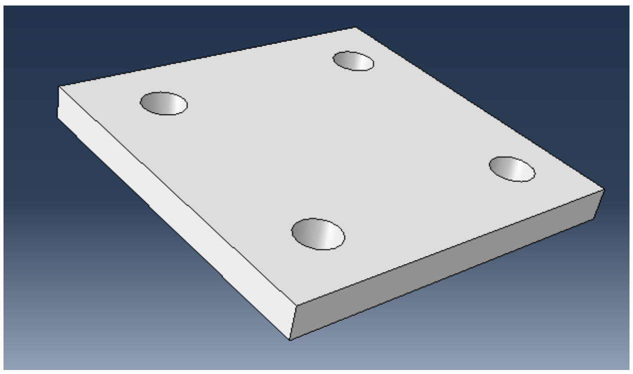
- Modelling space 3-Dimensional, Type: deformable
- Base Features Shape: solid, Type: extrusion

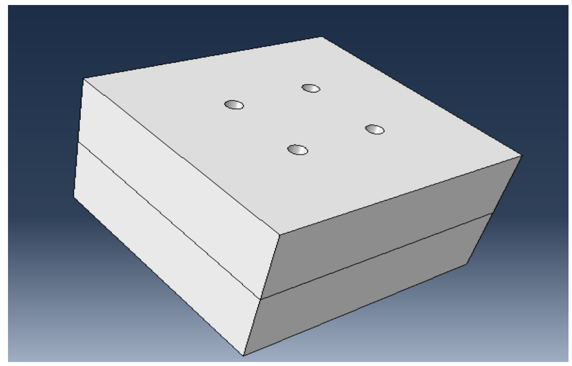
Tools which have been used in the creation of these parts are described as:

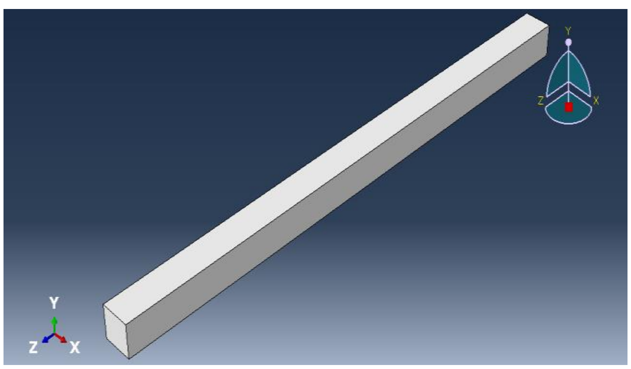
- Create Lines: Connected and Rectangles (4 lines)
- Create Circle: Centre and perimeter
- Auto-Trim
- Linear Pattern Array

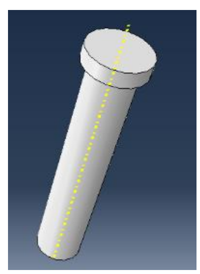
- Add constraint (Equal length)
- Add Dimension
- Extrusion tool

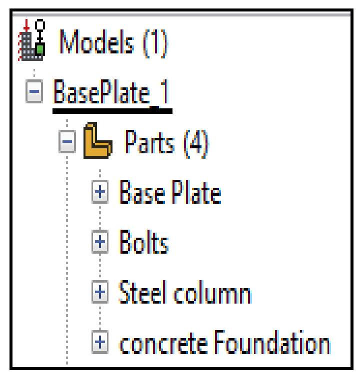
So, in this way all the 4 parts have been formed which are shown in the figures below.

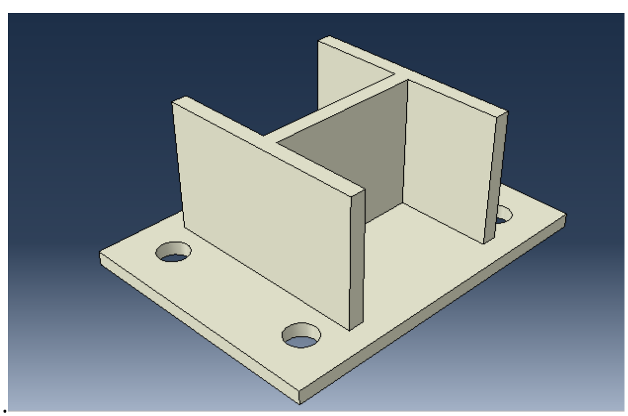




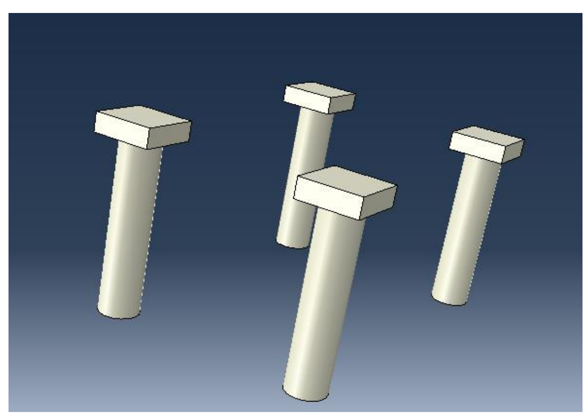








.



3.5.2.2 Materials

In this stage of the model formulation, materials are being introduced. These are the materials which are main ingredient of the model parts. Our project constitutes mainly two materials which are concrete and the steel. The properties assigned to these materials are:

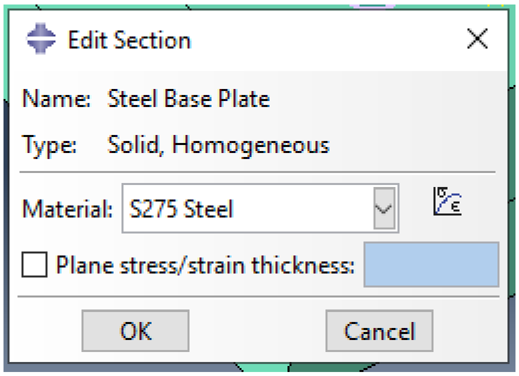
- General Properties
- Elastic Properties
- Plastic Properties

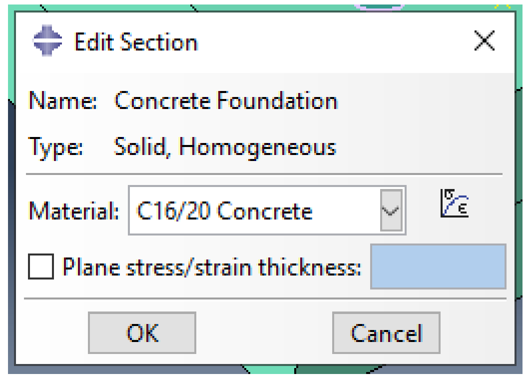
Values provided to such properties are according to the system international unit values which are being used globally.

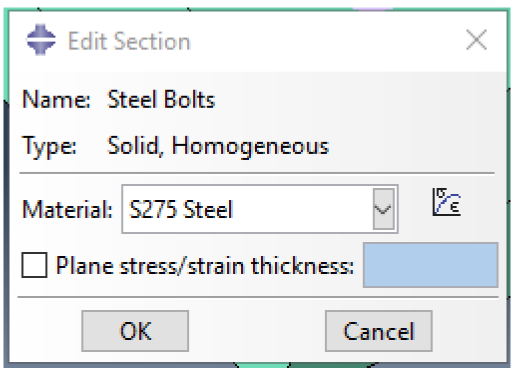
3.5.2.3 Sections

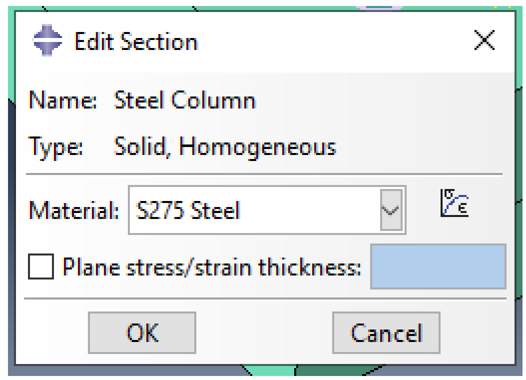
A section contains properties component information or specific part’s region. The information required in the description of the section depends on the type of region in question. For example, if a region is a corrosive wire, shell, or three-dimensional solid, you must provide a section in that region that provides data on the geometry of the circuit. Similarly, a solid region requires a classification that describes its large structures. Most components must look at the name of the material. Beam components should also check the profile name. In this stage different sections of the model are made like in our model we have distributed the model in four sections which are already been discussed in the previous paragraphs. This is done in order to analyze each part individually and most importantly if we intend to assign different parts different materials then these sections give us the liberty to perform such tasks and hence the same model comprise different materials forming a composite model which also was required in our project. The properties assigned were discussed as under:

- **Foundation** Type: Solid, Homogeneous, Concrete
- **Base Plate, Anchor Bolts and Column** Type: Solid, Homogeneous, Steel









3.5.2.4 Section Assignment

In this stage, the sections made in the prevous steps are being assigned to the parts created. This is being done in the parts module. So, we have made four parts and there are four sections being created so each of the part is assigned to its respective part. Because of this, materials formed when this component is meshed to the conditions will have the properties described in that section.

Components of any region are created independently. One segment can be assigned to as many different regions as possible. The material module can be used to make rigid sections, shell sections, beam sections, water sections, and other sections.

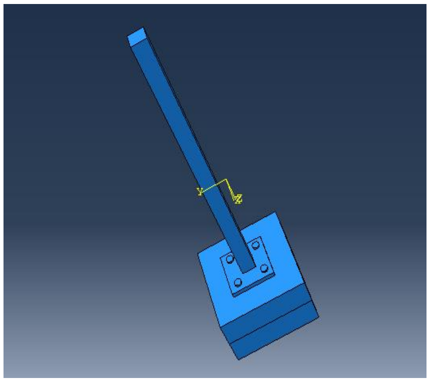
3.5.2.5 Instances

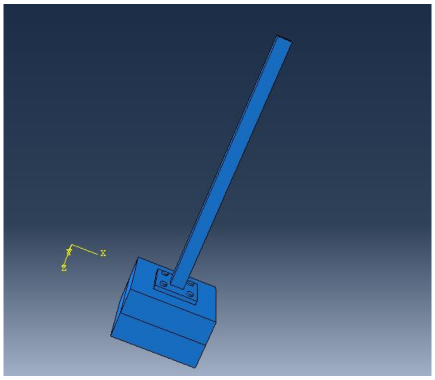
In this step different instances have been made and a model is formed. Instances are arranged in a manner that they directly fit into one another.

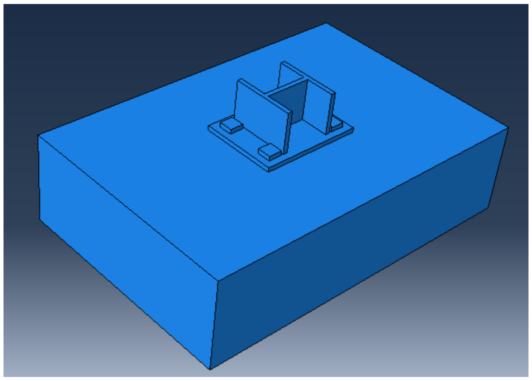
When a component is created, it becomes available in its integrated system, which is does not depend on the other components of model. The Assembly module is used to set related conditions to each other in the global coordinate system, hence creating assembly. Certain conditions are classified according to work conditions that vertices or faces or by using simple translation and rotation.

There are two types of example component:

- A reliable model component, for which the mesh component will be used.
- An independent component of the component, for which the mesh will be used for example.







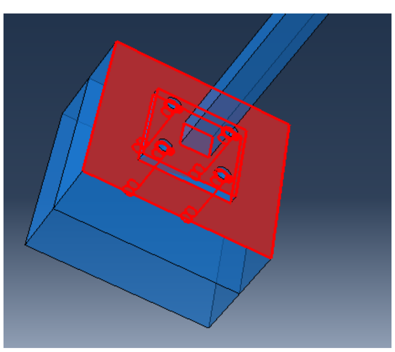
3.5.2.6 Surfaces

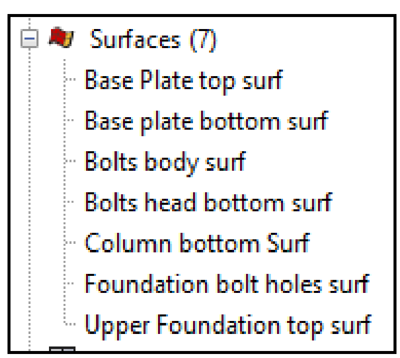
This option is used to determine the surface area of the contact, bonding, bonding, and bonding, as well as the properties of the distributed surface loads, acoustic radiation, body mass and surface aggregate output. In Abaqus / Standard it is also used to describe the area of analysis for cavity radiation and assembly loads.

Surfaces are created by selecting entities in the viewport that one wants to include in the surface. If a geometry surface is intended to be created, Abaqus/CAE allows us to select only faces and edges. Similarly, if mesh surface is intended to be created, Abaqus/CAE permits us to click only the element faces and element edges. Interior entities can be selected to include in the surface.

The surfaces which were introduced in the project are:

- Top surface of base plate
- Bottom surface of base plate
- Bolts body upper surface
- Bolts head bottom surface
- Column bottom surface
- Foundation bolt holes surface
- Foundation top surface

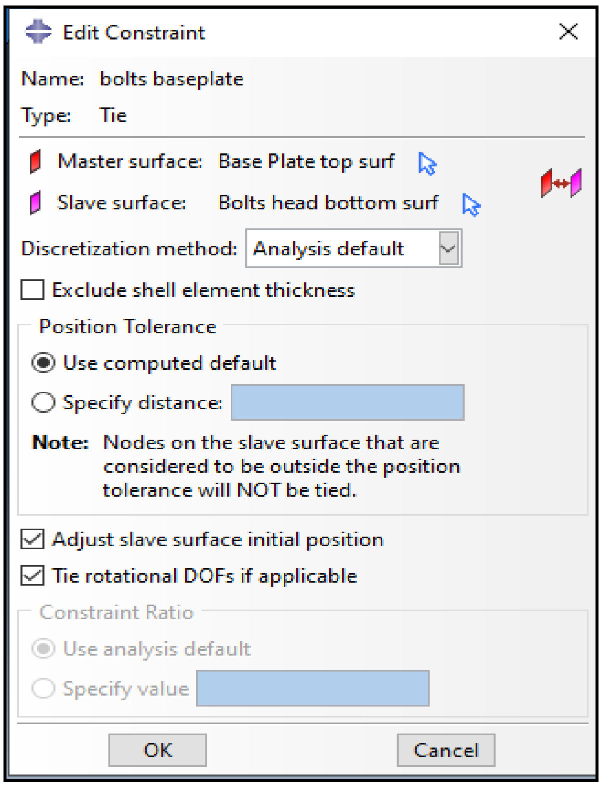


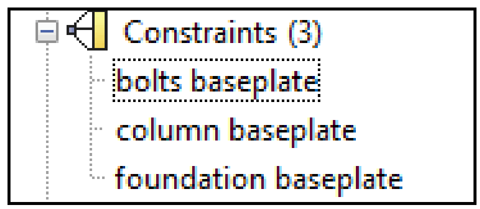


3.5.2.7 Constraints

- The constraints described in the Communication modules define constraints on the levels of freedom analysis, while constraints defined in the Assembly module define only constraints on the first instance positions. In the Interaction Module, degrees can be masked by freedom between model states, and constraints can compress and restart to modify the analytical model. At present, it often causes the following types of problems.
- **Tie:** The binding barrier allows the binding of two regions even though the radiation generated in the region is uneven.
- **Rigid body:** The rigid body interference allows for the compression of the circuit of the assembly points along the reference point. The relative positions of regions that are part of a solid body are constant in the analysis
- **Coupling:** An coupling obstacle allows for the forcing of the earth's motion at one point.
- **MPC constraint:** An MPC obstacle allows for the coercion of the movement of regional slave states in a single point continuum.
- **Shell-to-solid coupling:** Blocking the fastening of the joint tightly allows the movement of the shell edges to move along the adjacent solid surface.
- **Embedded region:** An embedded region block allows embedding a model region within a "catch" region of the model or within the entire model.
- **Equation:** Equality is an equal barrier, of many points that allow for defining equal barriers between degrees of freedom.

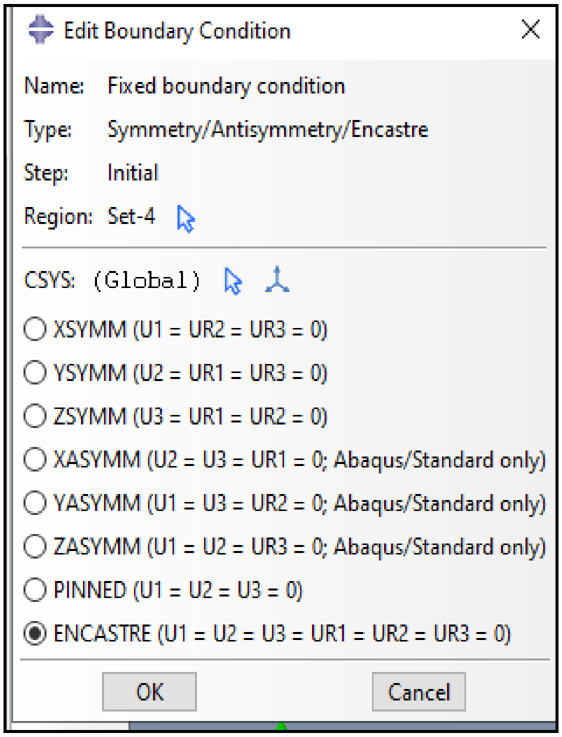
In this project, tie constraint is being utilized. This is because, this type of constraint genereally fixes the two surfaces which is required in this type of connection. Firstly, the base plate top and the base of the column has been tied in such a way that it is fixed. For this reason, tie constraint has been used. Secondly, the bottom of the bolts head and the top of the base plate are being tied together. Thirdly the body of the bolts and the holes made in the concrete are tied together but there was a friction factor being introduced in the constaint which has a value of 0.9. This is because there is friction between the anchor bolts and the concrete foundation and this has been catered in the friction factor compartment.

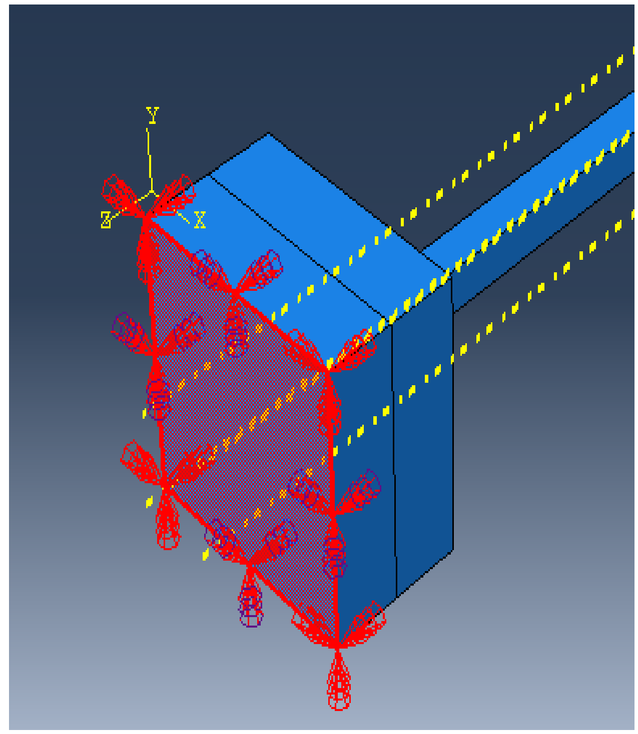




3.5.2.8 Boundary Conditions

In this stage, boundary constraints are set to the model so that it is restrained in any of the six degrees of freedom i.e. reaction in 3 axis as well as rotation in the 3 axis. The boundary condition used in the project is encastre which restrains all the 6 degrees of freedom. This boundary condition is being applied to the bottom surface of the foundation.



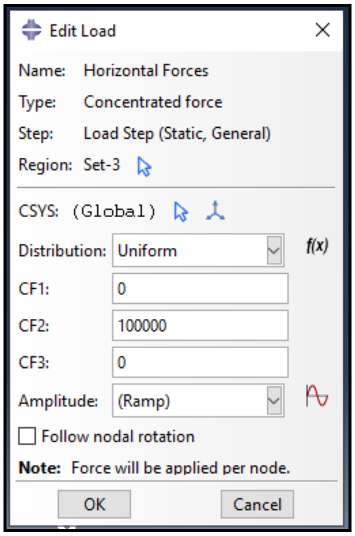


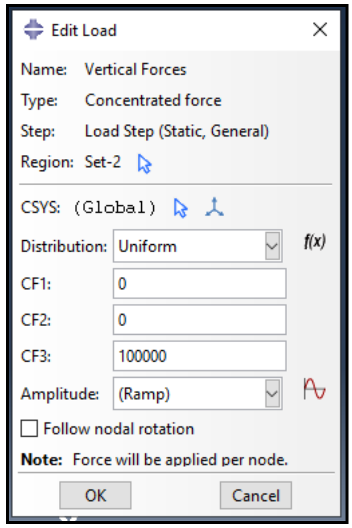
3.5.2.9 Loads

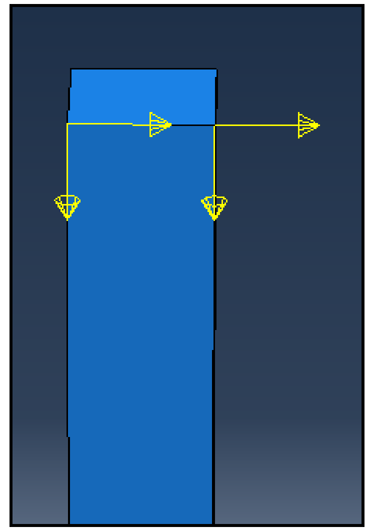
In this modeling stage, external load is being applied to the model so that this loading will create the stresses and the model can be analysed accordingly. An external loading can be performed in many forms but the most common are:

- Concentrated tractions
- Concentrated fluxes
- Incident wave loads

Distributed loads of many types is applied; they are element type dependent and are stated in the library of the element. Two loads are being applied to the model one is the horizontal load and the other is vertical load. Both of these contribute together to form the resulting changes in the model.







In some cases, concentrated loads and other widely distributed loads (such as pressure applied overhead) may rotate when analyzing a non-geometric surface. Such loads are known as loads of fans. The follower loads may result in an unlimited contribution to the stiffness matrix, often referred to as load elasticity.

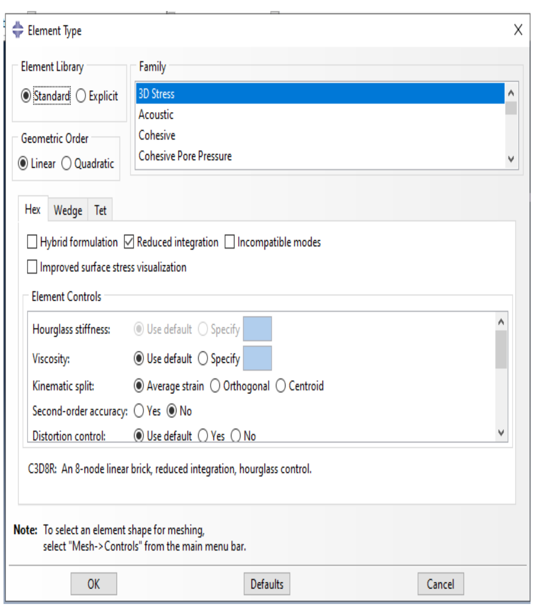
There are two ways to specify the loads distributed in Abaqus: the base-based distributed loads and the ground-based distribution loads. The loads used based on the Element can be placed on the bodies of the object, top, or edges of the object. The over-distributed loads can be placed over the geometric surface or the geometric edges. In Abaqus / CAE the distribution area and the maximum loads can be based on an element or based on a surface, while the distributed loads are determined by geometric or object bodies.

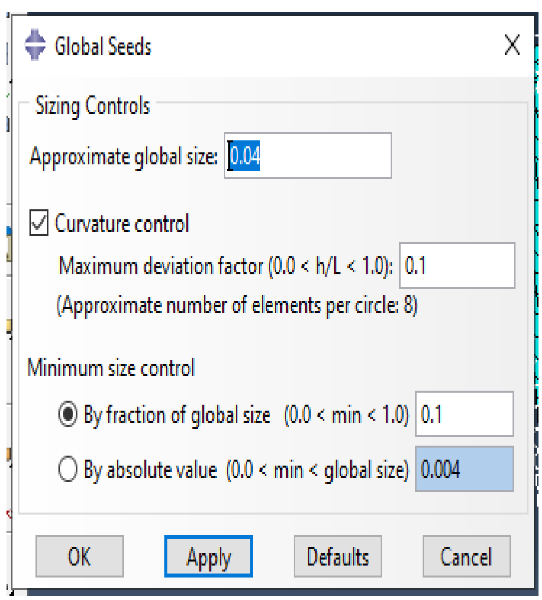
- **Element-based loads:** Use element-based loads to define the loads distributed on the surface of an element, the edges of an object, and the bodies of an object. For an element-based load the element element must be assigned (or a set name) and a distributed load type label. The load label label identifies the type of luggage and face of the item or edge on which the load is limited. This method of specifying distributed loads is very common and can be applied to all types of distributed load.

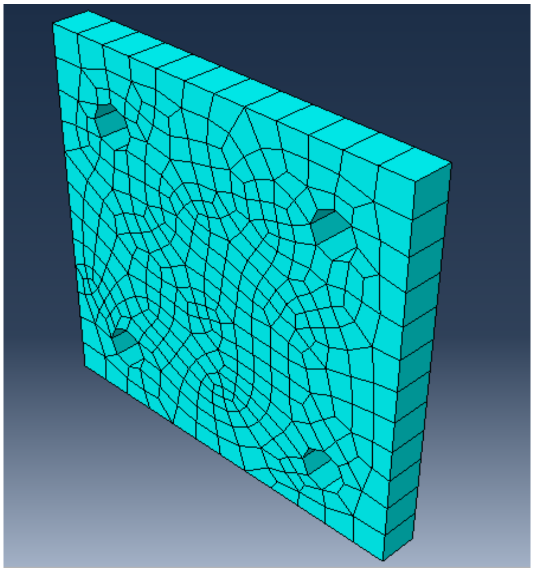
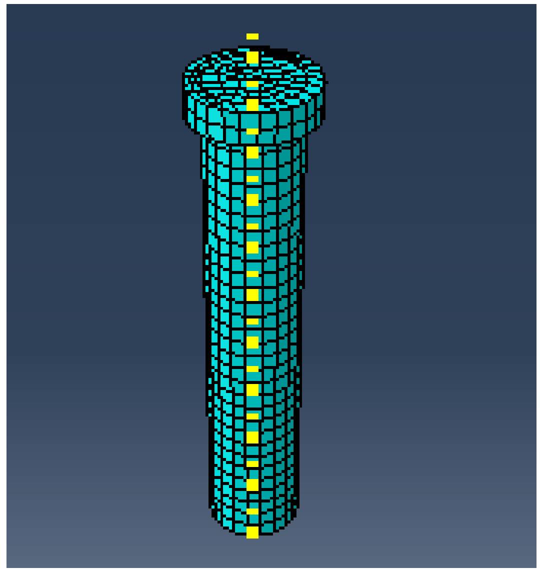
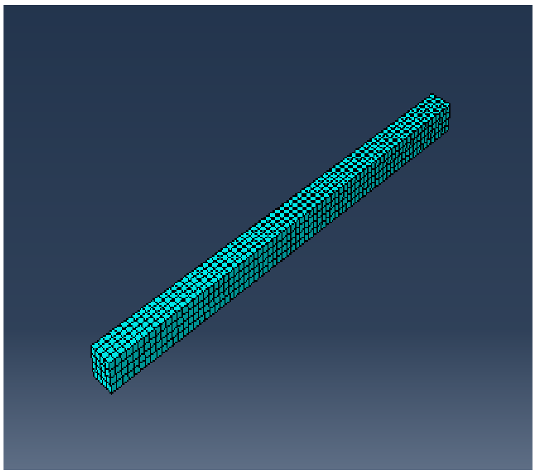
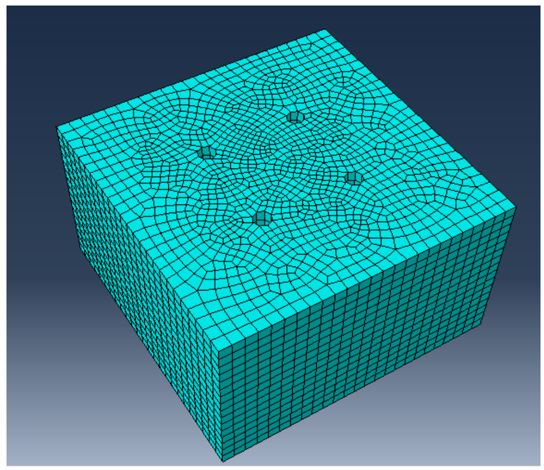
- **Surface-based loads:** Use surface-based loads to determine the load distributed on a geometric surface or on a geometric edge. For surface-based loads, the upper or lower term and the type of load distributed must be specified. The surface or edge, which contains feature and face information. In the Abaqus / CAE walls it can be described as a collection of geometric faces and ends or a collection of faces and ends. This method of determining a distributed load enables the user to enter complex models. It can be used for many types of elements with which a valid site can be defined. You can specify in the location description it can be specified similar to how the distributed load is applied to the boundary of the variable mesh domain in Abaqus / Exposed.

3.5.2.10 Meshing

Abaqus/CAE gives a number of many tecjniques for meshing in order to mesh different topologies modes. The meshing details used in the project are:





The following duifferent kinds of techniques for mesh generation are available:

- **Structured meshing:** Structured meshing works with mesh patterns established in a specific model environment. Complex models must be split into regions that are easy to use.
- **Swept meshing:** Swept meshing remove internally generated radiation by striking or rotating the axis of transition. Like formal meshing, swept meshing is restricted to models with special topologies and geometries.

- **Free meshing:** Free meshing method is the most flexible method. It does not use pre-configured mesh curtains and can work in any model environment.
- **Bottom-up meshing:** Uses a standard process for constructing a hexahedral or hex-controlled hexagonal solid or hard-to-rotate region using the default top meshing techniques. The bottom-up mesving is not discussed in any of the examples in this guide.

3.5.2.11 Job

In order to do the analysis a job must be created. The function of this job is to compile the whole work and after analysing each factor of constraints, loads and meshing of the model, results are provided by the software using the finite element modeling.

3.5.3 ANALYSIS

The second and final phase of the project is the analysis phase. At this stage, the model is analyzed after all the loadings applied in the observation section. AbaQUS provides comprehensive care of analytical methods. These techniques provide powerful tools for efficient and effective analysis.

3.5.3.1 Analysis continuation techniques

In many cases the results of the analysis represent a significant investment of financial effort. As a result, the cost of accounting is often reduced by using the results from the analysis already performed. In some cases the entire analysis history will have different WAQUS functions, each representing part of the model's response history. AbaQUS offers strategies for continuous analysis:

3.5.3.2 Modeling abstractions

All types of ABAQUS include some releases. In addition to the traditional deduction that comes with a good approach, you can incorporate techniques into your model to find the most affordable solutions. ABAQUS offers the following strategies for excessive filtering:

- **Sub-modeling:** The geographical locations of the model can be analyzed in great detail and interpret the results of the solution from the large global coarser model.
- **Meshed beam cross-sections:** A section of a complex beam section can be defined by adding multiple elements and complexes, and automatically generating the structures of the beam section. Cross sections:
  a) sent to ABAQUS/ General with three-dimensional combat features, which can be safely out of the aircraft as an end to freedom;
  b) produce beam section structures that can be used in the analysis of the next beam object than ABAQUS/ Standard or ABAQUS/ Exposed;
- **Special-purpose techniques:** Some of the analytical techniques don't come under general distribution and are listed as special purpose methods. Strategies offered by ABAQUS are as follows:

  a) Inertia relief method can be used as an inexpensive alternative to the full processing of a relaxed or partially loaded body carried by the accelerated loads of a powerful body.
  b) The components of the model can be removed selectively, and then reproduced later.
  c) Minor imperfections are introduced into the model, usually for bidding analysis.
  d) In ABAQUS/ Specified, the size estimation process can be used to control the fixed extension time.

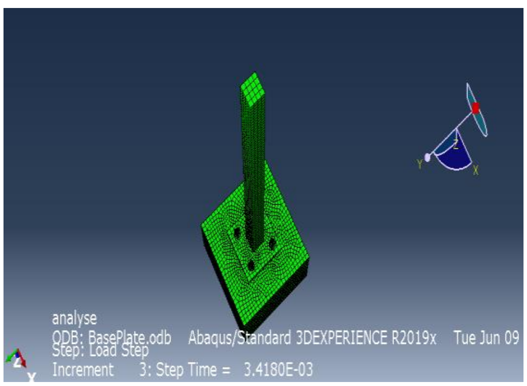
- **<u>Adaptivity techniques:</u>** Adapaction techniques allow mesh conversion to get a better solution. AbaQUS offers the following useful strategies:
  a) **<u>ALE adaptive meshing:</u>** The corresponding ALE adapter can be used to control radiation interference or to model the loss factor. The dynamic mesol of ALE in ABAQUS does not change the topology (elements and connections) of the figure, suggesting a certain limit to the ability of this high-quality statistical method to account for excessive collisions.
  b) **<u>Mesh-to-mesh solution mapping:</u>** Mesh-to-mesh solution mapping can be used as part of a mesh recovery plan for distortion control. Finding a solution from one cell to another is a standard process for regeneration analysis, where the mesh that has been severely damaged from its original layout is replaced with a better quality mesh and analysis is ongoing. The mapping method is used when elements are severely distorted during the analysis so that they do not give a good solution to the problem. It places the solution from an old, broken space into a new equation for the analysis to continue; and can only be used with continuous objects.

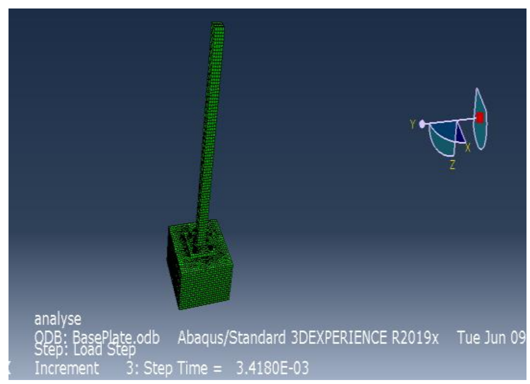
3.5.3.3 Tools used in the anlysis of the abaqus

There are number of different tools which are being utilized in the analysis process of abaqus. Some of the tools are discussed as under:

a) **<u>Plot undeformed shape:</u>** In this tool undeformed shape of the model can be seen with the values obtained because of the analysis. These values can be obtained by creating extra sets so that the values at particular points can be obtained. This section describes the unparalleled formation of structures and superimposed sites.

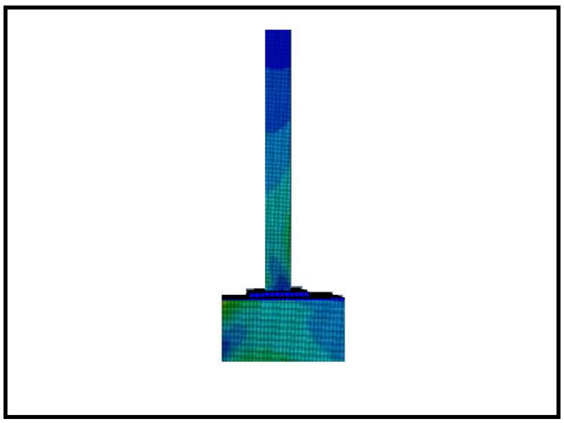
As soon as he opens the output database, Abaqus / CAE shows a vague build of the model. In conventional analytical methods, the unstructured structure shows the formation of the model without deviation. In the line deviation steps, the unpaired plot indicates the base condition of your model. The deformed structure shows the shape of your model depending on the values of the variance variables such as displacement. The shallow and deformed conditions are also used to indicate each of the structural elements found in Abaqus / CAE. The depth, features, or tendency of the material can be viewed from some undisclosed or damaged form or is very flattering to see the difference it has made.

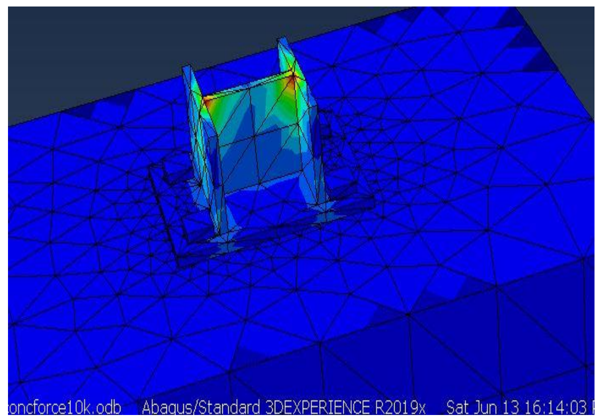


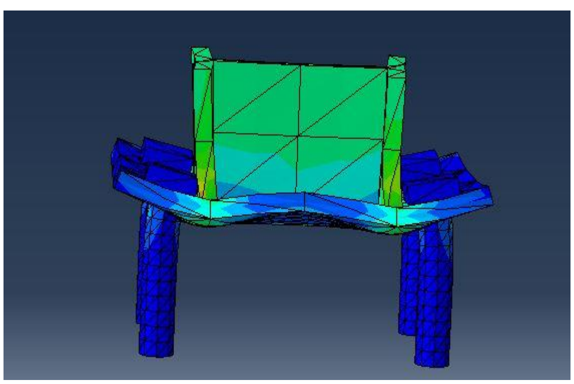


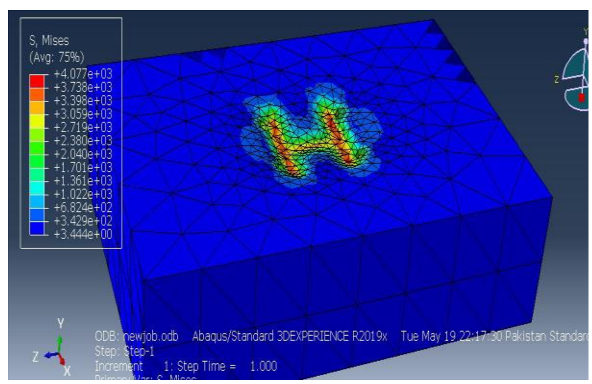
b) **Plot deformed shape:** shows the structure of the model at a specific location and the outline of results. Most processors in Abaqus / Standard or Abaqus / Exprehensive encrypt the output data automatically and choose to move through the nodal vein mass to use it as a damaged variable. When Abaqus reads the output data, it uses a non-corrupted variable to determine the structure of the corrupted model. In the elastomeric blocking example, the user requested a debug output for all models after 10 increments, and the export was selected as the default variable. If the output field variance is not corrupted, Abaqus / CAE attempts to select the default. Most process in Abaqus / Standard and

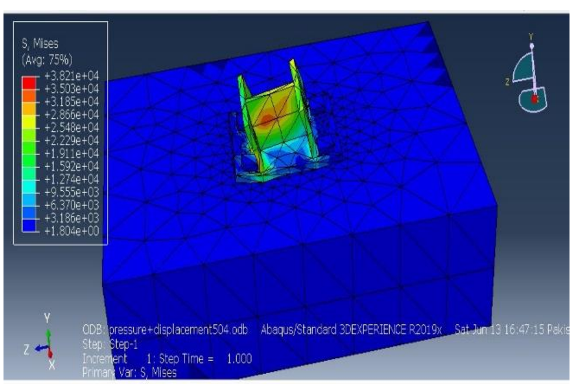
Abaqus / Write specifications in this automatically extracted database; in these cases, Abaqus / CAE chooses to be deployed as the nodal vein to use in the default correlation variable. Some processes - for example, heat transfer - do not record transfers to an automated repository; Therefore, Abaqus / CAE does not select the error-prone variable. Abaqus / CAE cannot display a corrupt structure if the output database does not contain any variables that can be used to calculate the corrupt state.

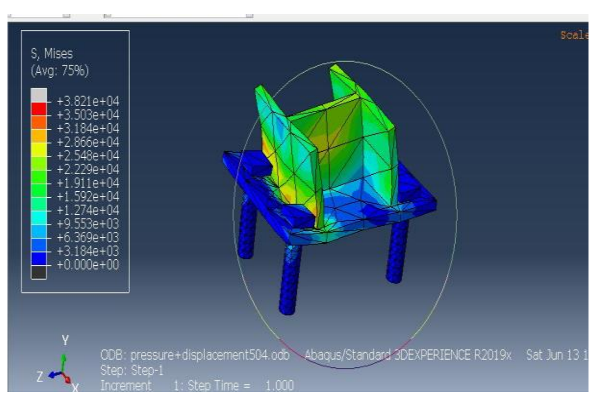












3.5.4 RESULTS

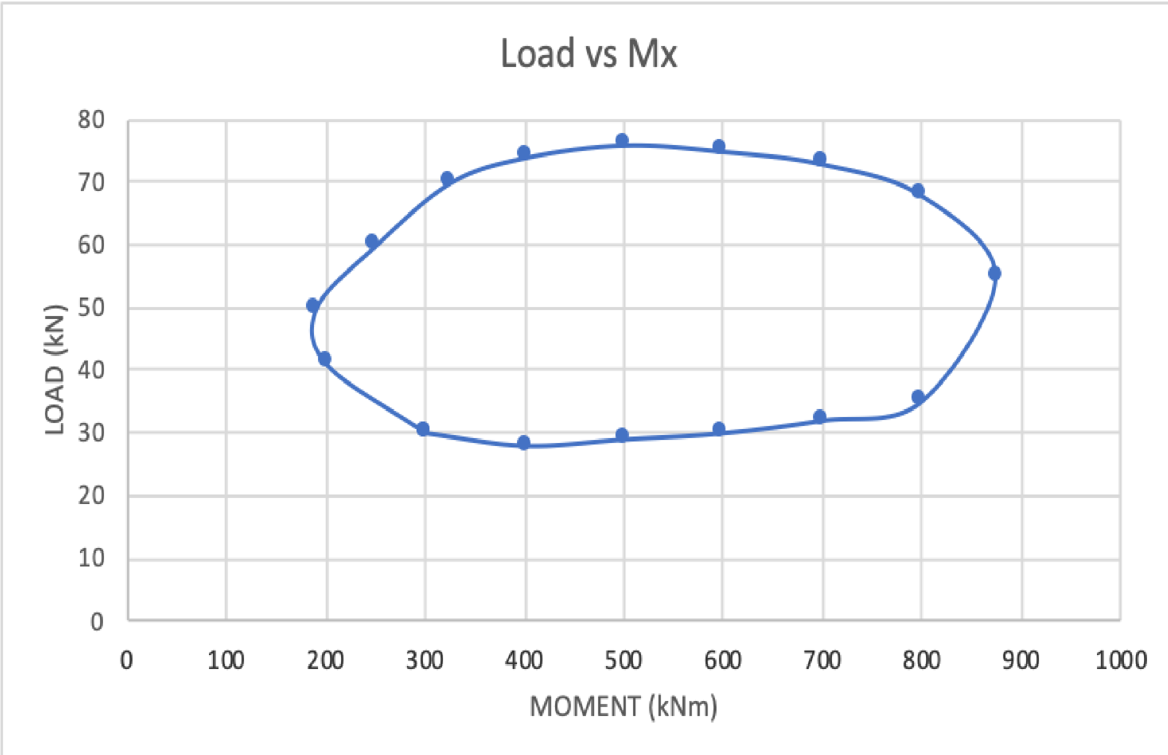
A series of loads have been applied to the model. This is done in order to form an interaction surface which is specifically for this squared 0.5m by 0.5m base plate. As a result of these series of loading, a series of moments are obtained. This loads and moments data is being plotted on an excel sheet and two ellipse are formed. First ellipse is drawn graphically between the load and the moment along x-axis and the second ellipse is drawn graphically between the load and the moment along y-axis.

3.5.4.1 Load vs Mx

Data

Load	Mx
30	300
41	200
50	190
60	250
70	325
74	400
76	500
75	600
73	700
68	800
55	875
35	800
32	700
30	600
29	500
28	400
30	300

Graph

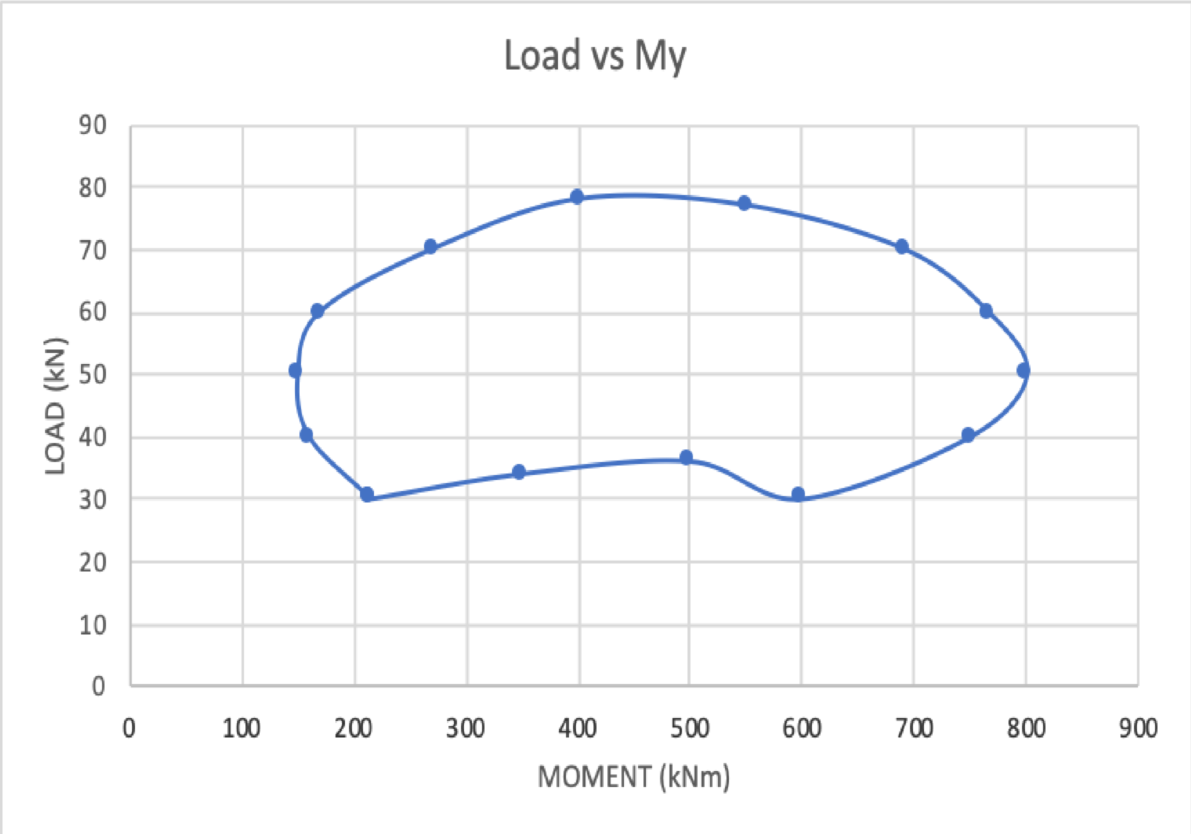


3.5.4.2 Load vs My

Data

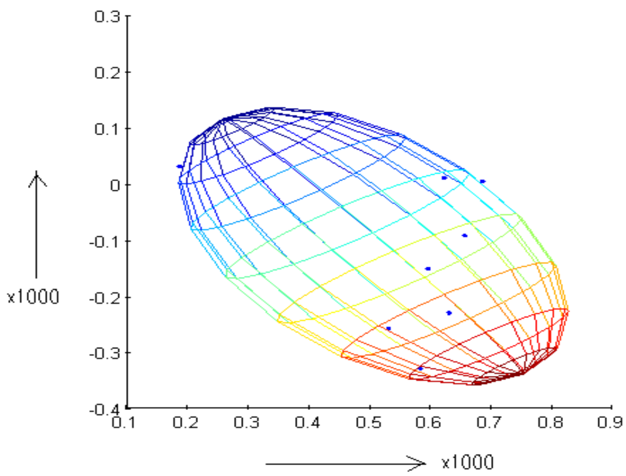
Load	Moments
30	215
40	160
50	150
60	170
70	270
78	400
77	550
70	690
60	765
50	800
40	750
30	600
36	500
34	350

Graph



So, there are two ellipses which are being formed between the load and the moment across x-axis and moment across y-axis. In order to form the ellipsoid or in other words the interaction surface for the base plate which is actutally the objective of all this proceedings , both the ellipses are combined together in such away that it will form a 3D surface by combining two 2D surfaces such that one ellipse is in x-y plane and the other ellipse in the y-z plane so the resulting surface would be in the x-y-z plane making a 3D one. So, it is actually only two ellipses which are being drawn using the data, rest of the ellipses are formed using the sequence which the two ellipses are projecting.

This can be done in the ABAQUS as well as the number of different softwares but because of some problems this couldn't be done on ABAQUS but the interaction surface is being formed in the matlab. This exception of introducing matlab at the end doesn't include any error because it is just showing the results of the points which are being created using the data which is initially obtained from abaqus. Also, it forms a very smooth ellipsoid because we have used the best fit curve for this purpose so the final form of the ellipsoid or simply called the interaction curve is as follows:



Chapter 4

CONCLUSIONS

The developed procedure is an extension of uniaxial methods, widely accepted for the analysis of base plates, it also gives accurate results for moments about one axis. Another advantage, of the presented biaxial procedure calculation, is to be an alternative for FEM also called finite-element method for the analysis of special projects with arbitrary base plate shapes with openings. Statically loaded structures, if the strength is much larger than the demand the ductility is not necessary and it is acceptable to design with the limit state of tensile or shear strength of anchor rod group governing the design.

Frames designed for seismic lateral load resistance are expected to behave in a ductile manner and in this case, it may be necessary to design the foundation and the column base plate connection so that the center limit state of tensile and shear of the anchor rod group don’t govern the design.

Shear forces can be transferred in bearing by the use of shear lugs or by embedding the column in the foundation.Since the developed procedure is an extension of uniaxial methods, widely accepted for the analysis of base plates, it also gives accurate results for moments about one axis.Another advantage, of the presented biaxial procedure calculation, is to be an alternative to the finite element method (FEM) for the analysis of special projects with arbitrary base plate shapes with openings.

There are actually two main methods which are adopted in our study. The first one includes the software called the matlab and the second one uses the ABAQUS. Interaction surfaces are being formed in both these cases. Matlab uses the code to formulate the data and that data is then used to form the interaction surface while the ABAQUS uses the data obtained by using the finite element method in order to form the interaction surface. After the formation of both of these surfaces for the same base plate, the results are being compared and find out the accuracy of using matlab and abaqus. Also, there is no defined method for the biaxial analysis of the structures so in our final year project we have given a heed on the biaxial analysis of the columns and the base plate.

The application of all the loads on the particular base plate form the interaction surface which can also be called as the capacity of the base plate which means that if a point lies outside of the interaction surface then the base plate will fail and hence we have a wild guess in the start of the application of the load that whether the base plate is able to take the load and the moments originated by the application of these loads or not.

Therr are a number of ways to proceed in it but abaqus is chosen because of its ability to operate between the Implicit solver, generally used for stress/strain, and the highly dynamic Explicit solver, for a seamless Dynamic-Static co-simulation. Basically the purpose of doing all this is to create an interaction surface which will also be called as the capacity diagram and it will tell the user what will be the optimum value for the combination of moments can be applied for a particular load so that it may not fail and give a strong resistance against the applied forces and moments. Using the abaqus, a set of different loads are applied at an angle (and the angle is 45 degrees so that the moment in both of the axis can be obtained) and for that particular load and same base plate a number of different values of Mx and My are obtained and when all of these are arranged on a plot they made an approximate ellipse surface which is obtained by using best fit curve.

Now, there are two types of cuves or ellipses that are obtained using the data given by abaqus and those ellipses are in xy plane and yz plane so the resulting surface would be in the xyz plane and hence is called the ellipsoid of the capacity or the interaction surface which gives the optimum value. The graph showed the values in which we have Mx at x-axis axis and My at y-axis so if we cut the plane for a particular value of load then we will obtain an ellipse and that approximate ellipse will give us the values of maximum Mx and maximum My for that particular load so that any moment which is greater then that applied moment will cause the failure of the base plate.

The main inaccuracy exists while defining the connection of the surfaces because there were a number of constraint types like the general and tie constraints etc. both have the limits if their own like the tie constraint fix the two surfaces and have a great rigidity where as in general constraint value, a friction factor is used whose value is taken as 0.3 but it can deviate from this value as well. Sometimes, 0.2 is used and mostly 0.3 is preferred depending upon the friction between the two surfaces. A friction factor of 0.3 is used in our project for the friction between bolts and the base plate and bolts and the concrete foundation whereas the tie constraint is used in the connection between the steel base plate and the steel column.

Another factor which contributes for the inaccuracy of the results is the meshing of different parts which have been created during the project. This is because for the regular straight objects like cubes etc. the meshing is just in the form of squares or rectangles so there is an easy distribution for the meshing part but when a curve is introduced into the model then in this case it could not be meshed so easily and first the model is being separated in to many parts and then the meshing is done or else the Abaqus doesn't allow to mesh the circular or curved shaped objects.

Chapter 5

REFERENCES

Steel column bases under bi-axial bending loading conditions by Patricia Medeiros Amaral

Elasto- Plastic Analysis of Plate Using Abaqus by Rohan Gourav Ray

Bi-axial analysis of General shaped base plate by R. Gonozalo Orellana

ACI. (2014). "Building Code Requirements for Structural Concrete (ACI 318-14)". Farmington Hill, MI: ACI.

AISC. (2006). "Steel Design Guide 1: Base Plate and Anchor Rod Design". Chicago, IL: AISC.

AISC 360-16. (2016). Specification for Structural Steel Buildings. Chicago, IL: AISC.

Bresler, B. (1960). "Design Criteria for Reinforced Concrete Columns Under Axial Load and Biaxial Loading". Journal of the American Concrete Institute, Vol. 57, pp. 481-490.

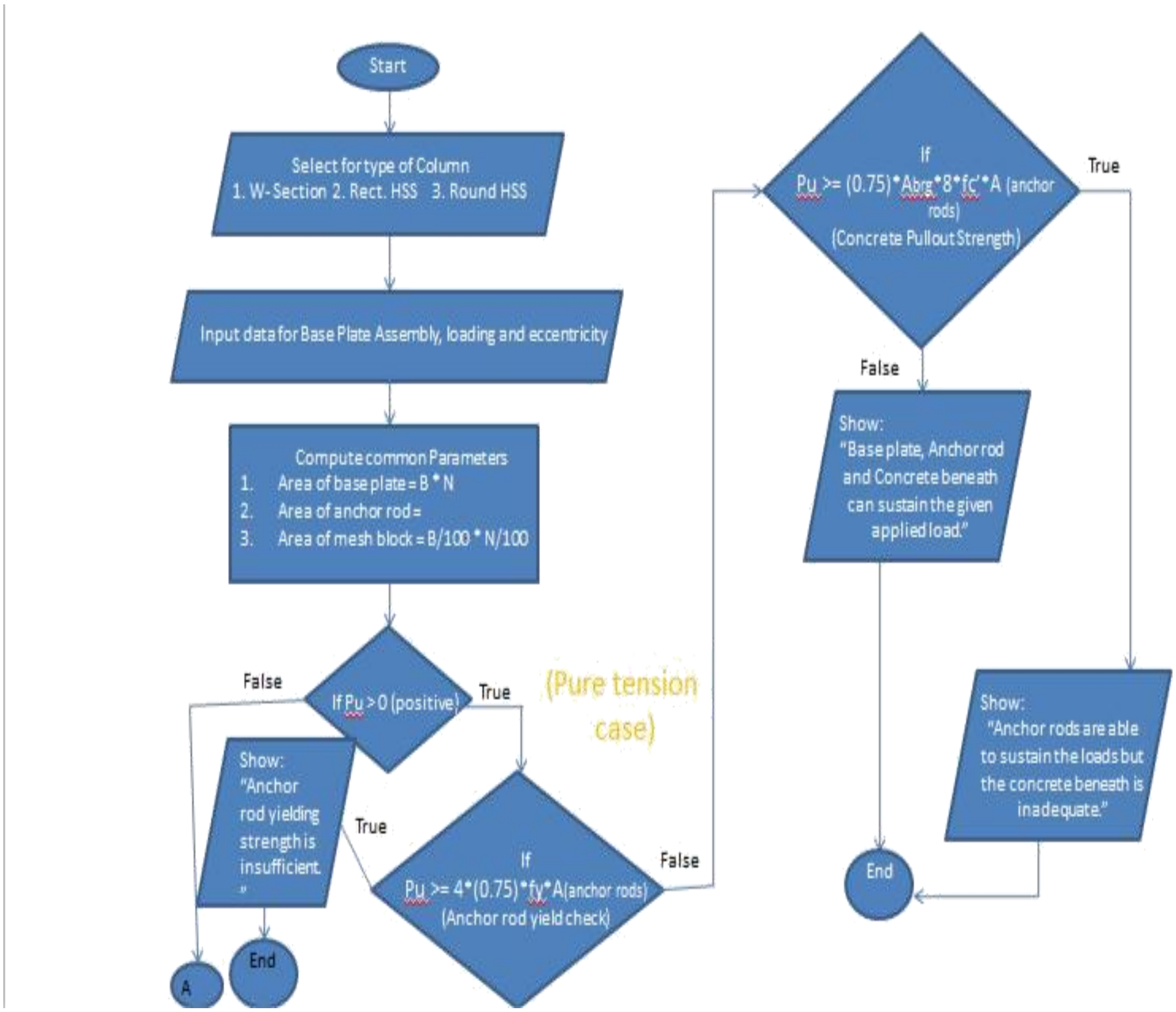
Gourav, R. (2015). Elasto-Plastic Analysis of Plates using Abaqus.

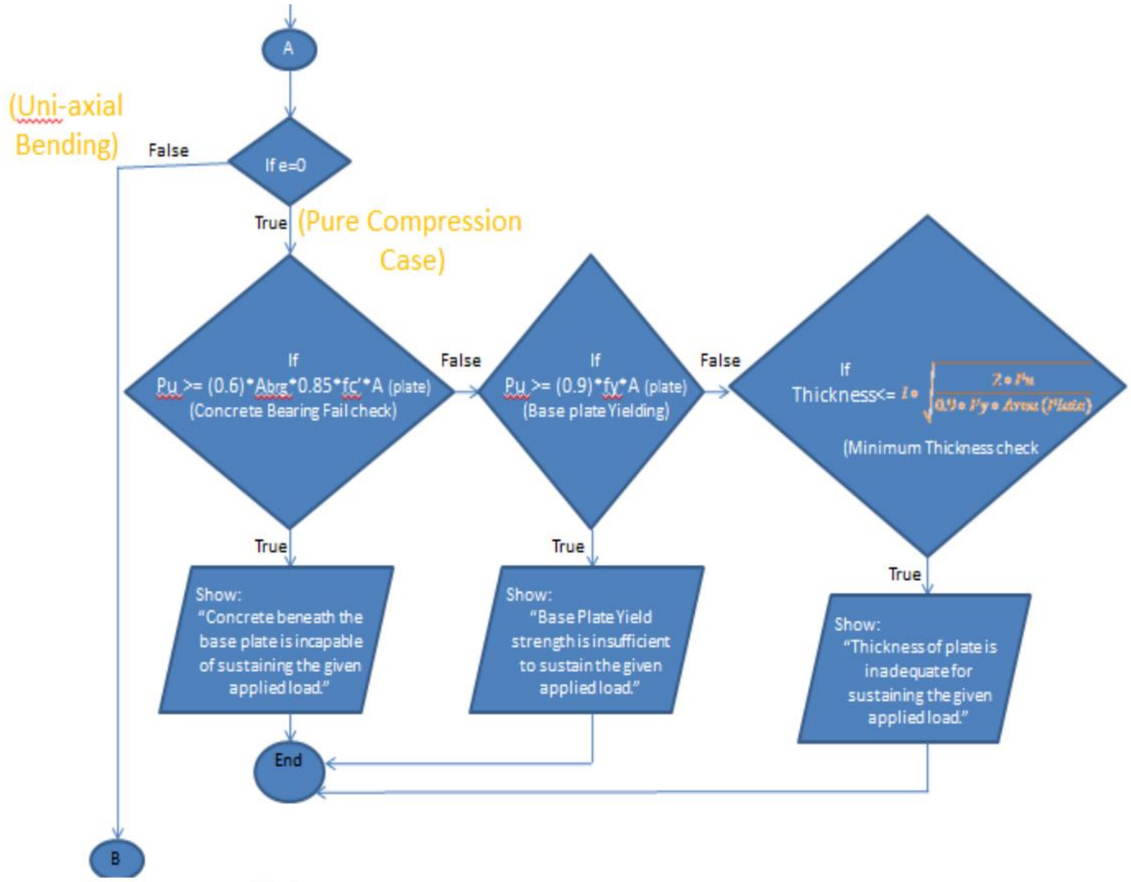
Medeiros, P. (2014). "Steel Column Bases under Biaxial Loading Condition". Porto, Portugal: University of Porto.

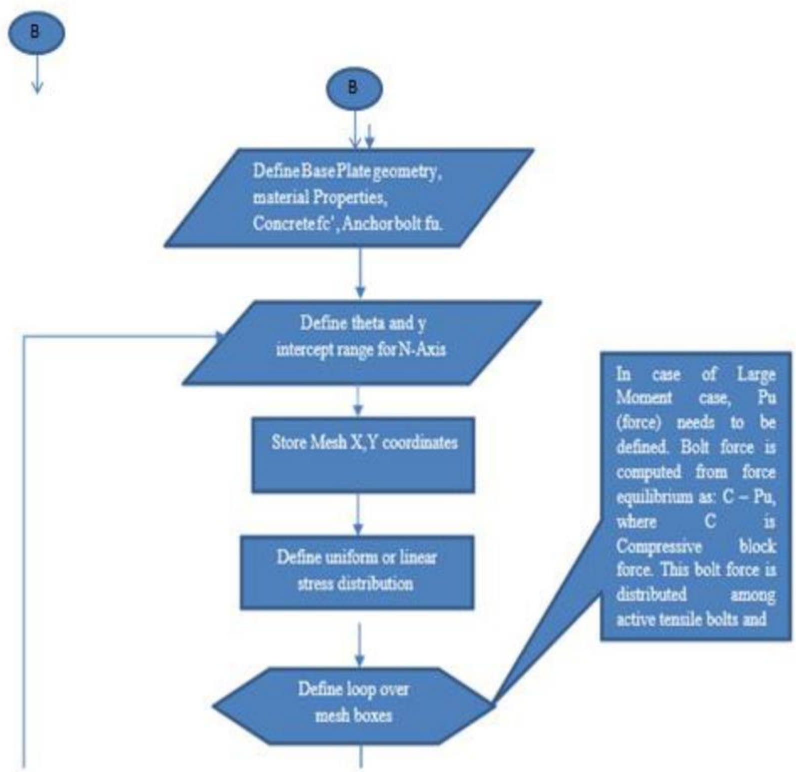
Orellana, R. G. (n.d.). Biaxial Analysis General Shaped Base Plates.

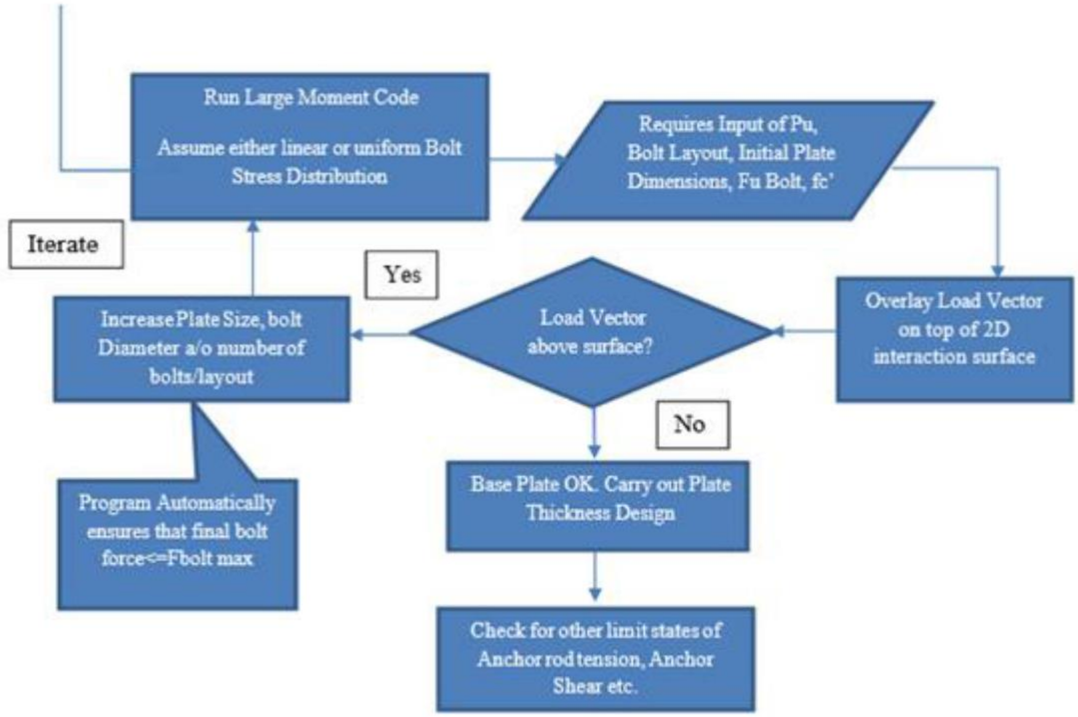
APPENDIX

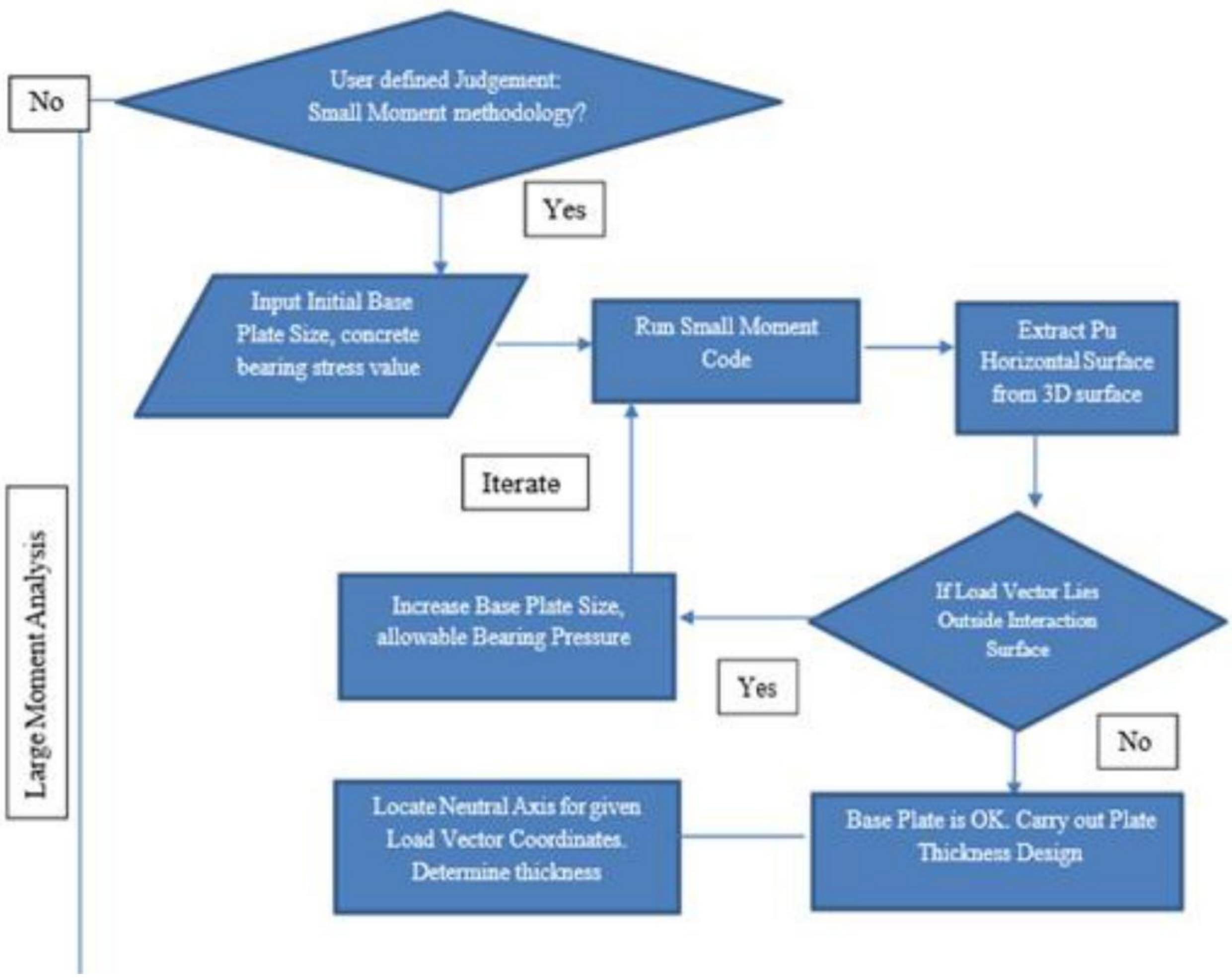
Small Moment Code

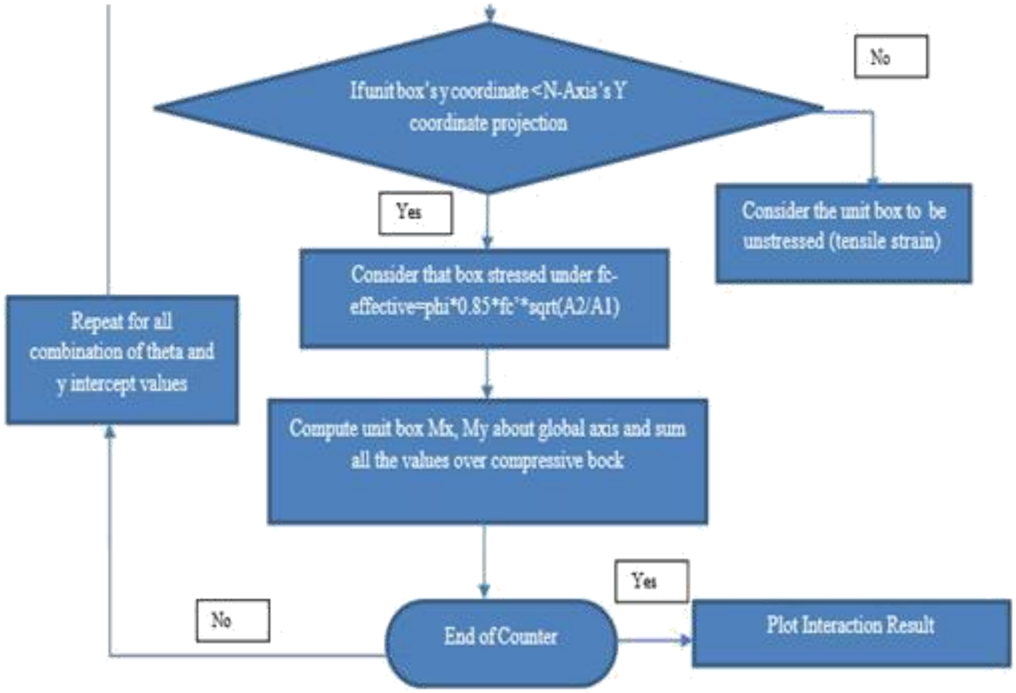



A
(Uni-axial Bending)
False
If e=0
True
(Pure Compression Case)
If
Pu >= (0.6)*Abrg*0.85*fc'*A (plate)
(Concrete Bearing Fail check)
False
If
Pu >= (0.9)*fy*A (plate)
(Base plate Yielding)
False
If
Thickness<=
(Minimum Thickness check
True
Show:
"Concrete beneath the base plate is incapable of sustaining the given applied load."
True
Show:
"Base Plate Yield strength is insufficient to sustain the given applied load."
True
Show:
"Thickness of plate is inadequate for sustaining the given applied load."
End
B


B
B
Define Base Plate geometry, material Properties, Concrete fc', Anchor bolt fu.
Define theta and y intercept range for N-Axis
Store Mesh X,Y coordinates
Define uniform or linear stress distribution
Define loop over mesh boxes
In case of Large Moment case, Pu (force) needs to be defined. Bolt force is computed from force equilibrium as: C – Pu, where C is Compressive block force. This bolt force is distributed among active tensile bolts and


Run Large Moment Code
Assume either linear or uniform Bolt Stress Distribution
Requires Input of Pu, Bolt Layout, Initial Plate Dimensions, Fu Bolt, fc'
Iterate
Yes
Increase Plate Size, bolt Diameter a/o number of bolts/layout
Load Vector above surface?
Overlay Load Vector on top of 2D interaction surface
No
Program Automatically ensures that final bolt force<=Fbolt max
Base Plate OK. Carry out Plate Thickness Design
Check for other limit states of Anchor rod tension, Anchor Shear etc.


No
User defined Judgement: Small Moment methodology?
Yes
Input Initial Base Plate Size, concrete bearing stress value
Run Small Moment Code
Extract Pu Horizontal Surface from 3D surface
Iterate
Large Moment Analysis
Increase Base Plate Size, allowable Bearing Pressure
If Load Vector Lies Outside Interaction Surface
Yes
No
Locate Neutral Axis for given Load Vector Coordinates. Determine thickness
Base Plate is OK. Carry out Plate Thickness Design



No
If unit box's y coordinate < N-Axis's Y coordinate projection
Yes
Consider the unit box to be unstressed (tensile strain)
Consider that box stressed under fc-effective=phi*0.85*fc'*sqrt(A2/A1)
Repeat for all combination of theta and y intercept values
Compute unit box Mx, My about global axis and sum all the values over compressive bock
Yes
No
End of Counter
Plot Interaction Result