Available at www.ijsred.com

RESEARCH ARTICLE OPEN ACCESS

Finite Element Analysis of Rectangular RC Wall Subjected to Monotonic Load

Tsenat Befkadu Bekele*

*Lecturer, Civil Engineering Department, Madda Walabu University, Bale Robe (Ethiopia)
tsenat.befkadu0203@gmail.com

Abstract

Finite element analysis of shear wall in linear elastic material and non-linear material behavior, to check the behavior in each node , the analysis mainly concern compared plane stress elements (Q4, Q8) and solid elements of the sameand different mesh size, of each element with theoretical results. Here a shear wall is designed for both elastic and in elastic states,so as to recommend suitable element size and to determine the stress and strain relationship of the wall. The software used in this paper is finite element software through ABACUS.

Keyword: **Finite element, mesh size, plane stress Q4 and Q8 element and Abaqus**

1.INTRODUCTION

finite element method (FEM),or finite element analysis is based on the idea of building acomplicated object with simple blocks or dividing acomplicated object into small and manageable pieces.application of this idea can be found everywhere in everyday life ,as well as in engineering.

FEM in structural analysis (the procedure)

- Divide structure into pieces(elements with nodes)
- Describe the behavior of the physical quantities on each element
- Connect (assemble) the element at nodes to form an approximate system of equations for the whole structure
- Solve the system of equation involving unknown quantities at nodes(e.g. displacement)
- Calculate desired quantities(e.g. stresses and strain) at selected element

2. STATMENT OF THE PROBLEM

Dividing acomplicated object into small and manageable pieces, whose behavior is readily understood and then rebuild the original system from such components to study its behavior. The aim of this paper is to prepare a suitable FE model for the shear wall and interpret (evaluate) the quality of the results means to determine the behavior of the shear wall in the perspective of stress ,strain and displacement of each discretized element so as to check the behavior of the whole shear wall.

3. FINITE ELEMENT METHOD

In this paper a discretization technic uses in order to study the behavior of shear wall. First idealize a model of two dimensional shear walls which has 1.2m length and 1.18mwidth. Then select some representative discrete technic this are bilinear rectangle (Q4) and quadratic rectangle (Q8/C3D8R). By using this technics will discretize the given shear wall with each technics separately and study the behavior of shear wall. Discretization is performing by using general purpose FE-program (ABAQUS).

The study of the behavior mainly depends by changing the number of discretization and evaluating the value of strain for each case. Nowcompare the discretization technic using their degree of freedom.

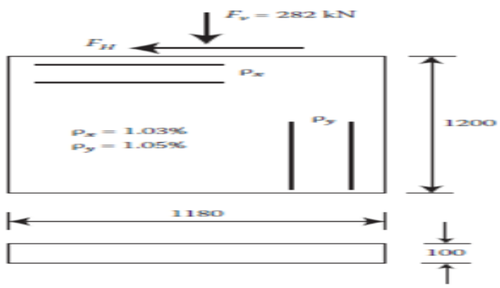


Fig 1. Rectangular Rc wall

3.1*Simple overview of the finite element method*

Suppose to solve a physical problem such as finding the stresses in an object when some prescribed forces are applied. This is a typical problem for FEA: some type of force is applied to an object and the response calculated subject to specified constraints.

Finite element method is a numerical method that generates approximate solutions to engineering problems which are usually expressed in terms of differential equations used for stress analysis, heat transfer, fluid flow, electromagnetic etc. One resorts to a numerical solution, the best of which is the FEM.

- Structure is partitioned into finite elements –that are joined to each other at limited number of NODES
- Behavior of an individual element can be described with a simple set of equations
- Assembling the element equations, to a large set, is supposed to describe the behavior of the whole structure.

3.2 Abaqus procedure

A step-by-step procedure for modeling and analysis of RC Shear Wall in ABAQUS is Explained here

Table 1. Basic Dimensions

Dimension	(mm)
Height	1200
Width	1180
Thickness	100

The material properties used are:

Compressive strength of concrete-C-50 mpa

Tensile strength of reinforcements Fe-365mpa

In order to assemblefirst to calculate the length of reinforcement bars in both vertical and horizontal directions. So by giving a25mm clean cover in both ends,

3.3Area of reinforcement in the horizontal direction;

As total=ρ x h x t

As total=0.0103 x(1200-50) x 100=1184.5mm^2

As for one bar=Π x $12^2/4$=113.1mm2 taking diameter 12mm

Number of bars in the horizontal direction;

N_h=total area of bar/area of one bar

$N_{h=}$1184.5/113.1mm2=10.47=11

Spacing between bars;

S= (1200-50-11 x120/10) =101.8mm

3.4 Area of reinforcement in the VERTICAL direction;

As total=ρ x h x t

As total=0.0105 x(1180-50) x 100=1186.5mm^2

As for one bar=Π x 12^2 /4=113.1mm^2 taking diameter 12mm

Number of bars in the vertical direction;

N_h=total area of bar/area of one bar

$N_{h=}$1186.5/113.1mm^2=10.47=11

Spacing between bars;

S=(1180-50-11 x120/10) = 99.8mm

4.ANALYSIS

4.1 ABAQUS/CAE:- This product is a Complete Abaqus Environment that provides a simple, consistentinterface for creating, submitting, monitoring, and evaluating results from Abacus simulations. Abacus/CAE is divided into modules, where each module defines a logical aspect of the modeling process; for example, defining the geometry, defining material properties, generating a mesh, submitting analysis jobs, and interpreting results.Starting abacus is a method that prepares your selves to use abaqus software to solve finite element problems.

4.2 Create Model and parts:-Defining the part geometry is generally the first step in the development of a finite element model. Here, the plate geometry is sketched and extruded to generate a solid part. In creating the model, the shear wall is composed of concrete element and reinforcement rebars.so created two parts for the concrete and reinforcement.

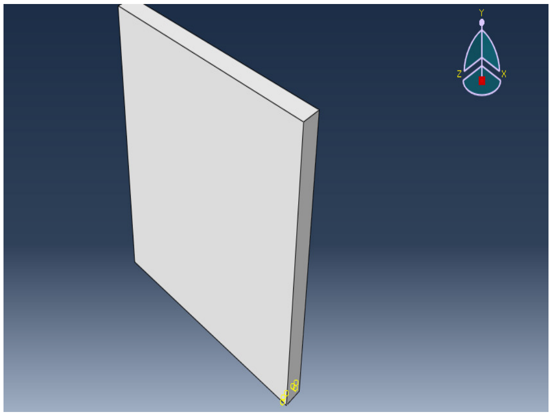


Fig 2: Concrete part

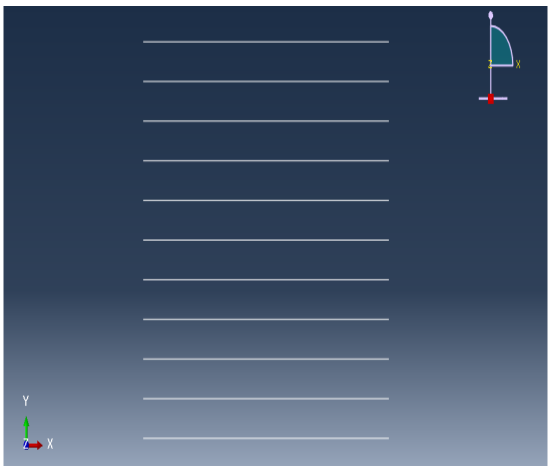


Fig 3:Horizontal rebar

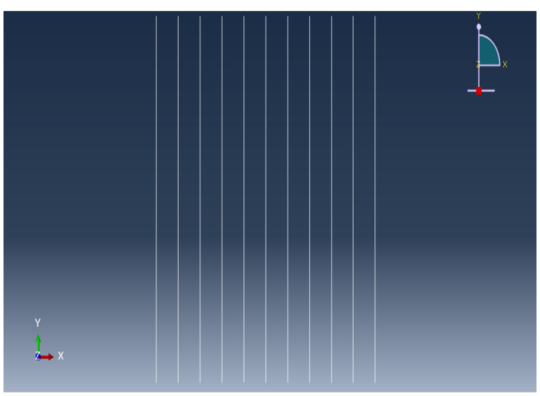


Fig 4 : Vertical rebar

4.3 Material property used:-Here the created parts are described by their behavior. The elastic and plastic material properties are inserted on each part. The properties are as follows.

ELASTIC property;

Concrete;

Table 2. Elastic properties

Elastic modules	Poisons ratio
38.4 x10^3	0.2

Reinforcement bar;

Young'smodulus	Poison's ratio
210 x 10^3	0.3

Plastic properties;

Concrete

Table 3. Plastic properties of Concrete damage plasticity parameters

dilation angle	31
Flow potential eccentricity	0.1
Biaxial/uniaxial compression plastic strain ratio	1.16
Invariant stress ratio kc	0.66667
Viscosity	0
compression yield stress	50.5N/mm2
Inelastic strain	0
Tensile yield stress	2.25N/mm2
Tensile cracking strain	0

Reinforcement

Yield stress	325Nmm2
Plastic strain	0

4.4Defining and Assigning Section Properties:

After the part and the material have been created, a composite layup section can be created. The composite layup editor is used to create plies and to assign materials and orientations to these plies. In this step, a composite layup that represents the plate layup is created and defined. Here both parts are defined and assigned.

4.5Assembling the Model;In order to apply boundary conditions, loads, etc., an instance of the part must be added to the assembly. Add a dependent instance of the part to the assembly.To assemble the model to define a datum were the reinforcement bars needed to be embedded. Here since the reinforcements bars should be imbeddedinside the concrete, a clean cover of 25mm is given.

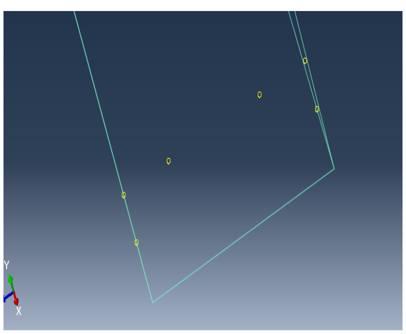


Fig 5: Datum fixing

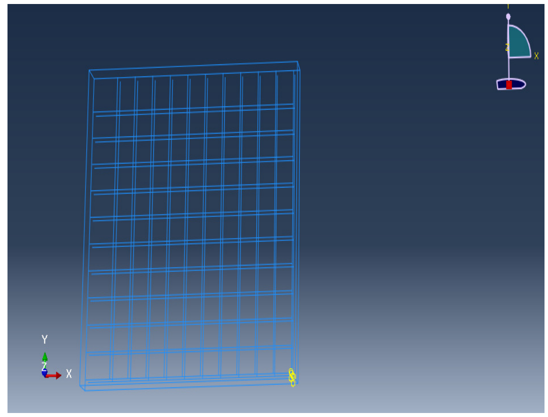


Fig6: Assembling

4.6 Defining Analysis Steps:- Here, a step is defined that allows boundary conditions and output requests to be added to the model.

4.7 Applying Loads and Boundary Conditions:-This step create boundary conditions that minimally constrain the movement of the bottom surface of the plate and prevent rigid body motion. A second boundary condition is created to impose a vertical displacement along the top surface of the plate and horizontal displacement. The plate is loaded using applied forces. When a simple structure, such as this composite shear wall, begins to fail under the action of applied forces, the structure fails rapidly because the load continues to increase as the load carrying capacity of the structure decreases. With displacement controlled loading, the load carried by the structure decreases as the structure fails which allows for a slower rate of failure here loads are applied to the shear wall both horizontally and vertically. And the boundary condition is fixed at the bottom part only. The maximum amount of load the computer can sustain is 500KN .

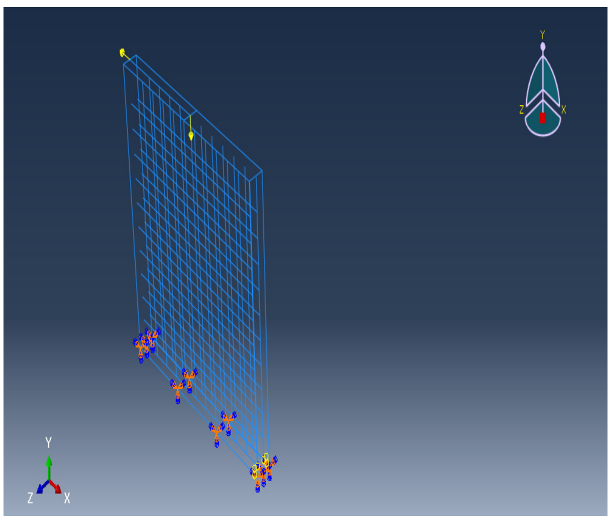


Fig 7:Loading and boundary fixing

Meshing: meshing is the main part of FEM such that the whole shear wall is sub divided in to parts so as to have good analyzing of the parts.Since a layered element is used in this modeling, a swept mesh is used to define consistent top and bottom element faces. Also, edge seeding is employed to conserve model size.There are many meshing methods constant strain triangle(CST),linear strain triangle(LST),Q4 and Q8(C3D8R).in this modeling used C3D8R and Q4 elements type.

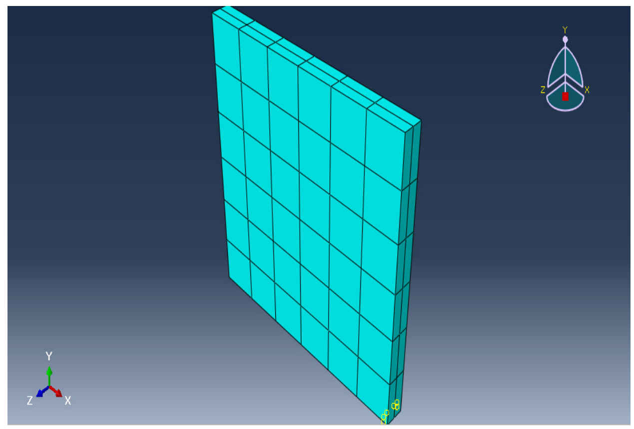


Fig8:Meshing

5.Analysis Results

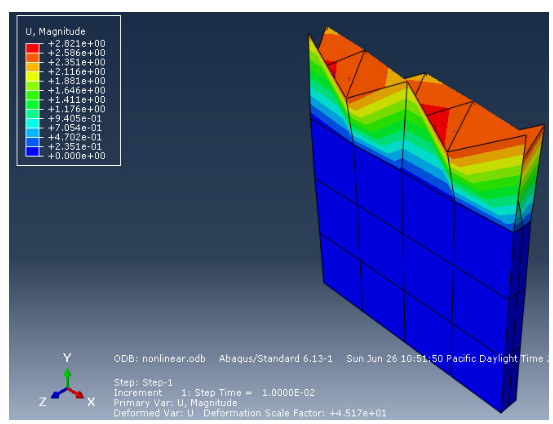
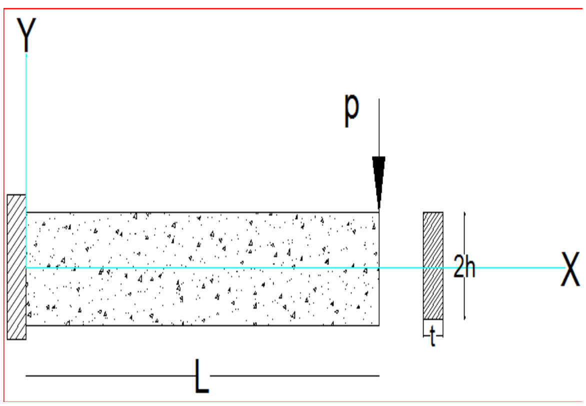


Fig 9 :nonlinear displacements

6. Analytical method

The exact solution for the deflection and stresses in an end-loaded cantilever is widely used to demonstrate the capabilities of adaptive procedures, in finiteelements, mesh less methods and other numerical techniques

Problem definition Fig. below shows a cantilever beam of depth h=1.18m length L=1.2m and 0.1m thickness, which is fully fixed to a support at x = 0 and carries an end load P=100KN.



The situation described may be regarded as a case of plane stress, $\sigma z=\tau xz=\tau yz=0$, if the plane stresses $\sigma x=\sigma y=\tau xy$ are expressed in a thin plane: which significantly has very small thickness, t.

Boundary conditions

$(\sigma y)y=\pm$□$=0$

$(\tau xy)y=\pm$□$=0$ ………………….. (a)

Since the bending moment varies linearly with x and σx at any section depends on y. it is reasonable to assume the general expression of the form

$\sigma x=\partial^2\emptyset/(\partial y^2)$=c1xy

C1 – constant and integrating σx twice wrt y

$\Phi=\frac{1}{6}c_1xy^3 + yf_1(x) + f_2(x)$ …………………….. (b)

Where f1 and f2 are functions of x.

Introducing Φ for compatibility, $\nabla^4\Phi=0$ and we have

$y(d^4f1)/(dx^4) + (d^4f2)/dx^4 = 0$ ……………(c)

Since f1 and f2 are independent of y for all x and y $(d^4f1)/(dx^4) = 0$ and $(d^4f2)/dx^4=0$ integrating and solve functions f1 and f2

f1(x) = c2x3 + c3x2 c4x + c5

f2(x) = c6x3 + c7x2 c8x + c9 ………..……….(d)

Substitute equation (d) in to (b)

$\Phi=\frac{1}{6}c_1xy^3+ (c_2x^3 + c_3x^2\ c_4x + c_5)\ y + c_6x^3 + c_7x^2\ c_8x + c_9$

$\sigma y = \partial^2\Phi / (\partial x^2) = 6(c_2y + c_6)x + 2(c_3y+c_7)$

$\tau xy = -(\partial^2\ \Phi)/(\partial x\partial y) = -\frac{1}{2}c_1y^2-3c_2x^3-2c_3x-c_4$…….(e)

Apply boundary conditions; substitute equation (a) in to (e) provided that

$c2 = c3=c6=c7=0$ and $c4= -\frac{1}{6}c_1h^2$

$-\int_{-h}^{h}\tau xy = P = -\int_{-h}^{h}(-\frac{1}{2}c_1y^2-c^4)tdy$

$P = \int_{-h}^{h}(\frac{1}{2}c_1y^2 - \frac{1}{6}c_1h^2)tdy$

$\frac{1}{2}$c1t($\int_{-h}^{h} y^2$dy – h$^2\int_{-h}^{h} dy$ = Ƥ solve the integraton gives

C1 = -3Ƥ/(2th^3)

σ x (x ,y) =(-3Ƥ/2th^3)xy = $\frac{-pxy}{I}$ and Ʈxy= $\frac{-p}{2I}$ (h^2- y^2) , I= t*h^3/12 = 0.0137m^4

σ x (1.2 ,0.59) =$\frac{-500000*1.2*0.59}{0.0137}$ =-258.339 *10^{5}N/m^{2}tension

σ x (1.2 ,-0.59) = 258.339*10^{5}N/m^{2}compression

Ɛx (x ,y) =$\frac{1}{E}$ (σ x - ν(σ x + σ y)) =$\frac{\sigma x}{E}$ (1- ν) = $\frac{Ƥxy}{EI}$ (1 –ν)

Ɛy(x,y) = $\frac{1}{E}(\sigma y - v\ (\sigma x +\ \sigma y)$

Ɛy(x,y) = $\frac{-v(\sigma x)}{E}$ = $\frac{-vpxy}{EI}$ = -1.635 X 10^{-6}xy

$Uy \int Ɛ\, ydy$ =-3.8 X 10^{-5}X$\int_{-0.59}^{0.59} ydy = 0$

$Ʈxy = \frac{-p}{2I}$ (h^2 – y^2)

$Ʈxy(0)\ =\ \frac{-500000}{2*0.0137}$(0.59^2-0^2) = -37.45X10^{5}N/m^2

$Ʈxy(0.59)\ =\ \frac{-500000}{2*0.3375}$(0.59^2-0.59^2) = 0

$$Uxy = \frac{ux + uy}{2} = 0.00016m$$

RESPONSE		MAXIMUM DISPLACMENT TO ACTUAL DISPLACMENT RATIO		
DOF(ELEMENTS)		8	32	72
ELEMENT	Q4(LINEAR)	4.95	8.23	18.62
	Q4(NONLINEAR)	4.000	8.2445	10.25
	Q8(LINEAR)	15.9	5.32	3.179
	Q8(NONLINEAR)	4.006	4.367	7.14
	Q4(WITH OUT RF	4.59	5.87	8.10
	Q8(WITH0UT RF)	3.537	4.622	9.327

Ɛx (x ,y) = -0.00076xy

$$UX = \int Ɛxdx\ =\ -0.00076y\int_{0}^{1.20} xdx$$
$$=\ -0.00055y$$

@y =0.59 ,ux = 0.00032m and ux = -0.00032m @ y= -0.59

7. Results

In this section clearly discuss the result of FEM and the analytical method of rectangular RC shear wall.

Table 4 maximum displacement to actual displacement ratio

LOAD DISPLACMENT RELATION SHIP:

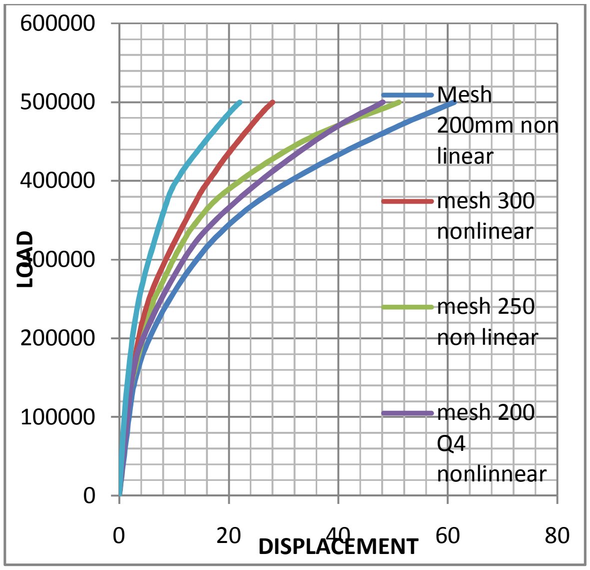


Fig 9: load vs dispersant

Conclusion

- Mesh size increase the elements are finer and the results are more accurate.
- Also conclude that Q8 (C3D8R) elements are nearest to the actual result than Q4 elements.
- Finite element results have good agreement to the actual calculated results.

REFERENCES:-

[1] EBCS - 2, M of W. & U. D.(1997). "Structural use of Concrete ", Addis Ababa Ethiopia.

[2] Gonzalez-Vidosa. (1990). Analytical Investigation Based on Nonlinear Finite Element Modeling. *ACI Structural Journal*.

[3] Alfarah , B ., Lopez-Almansa , F., Oller ,S. (2017). New methodology for calculating damage variables evolution in Plastic Damage Model for RC structures. Engineering Structures.

[4] Comi and Perego (2001), fracture energy based bi-dissipative model for concrete, Int.j.solids structer.38:6427-6454.

